PV Elite

User's Guide

Version 2014 (16.0)

November 2013

DICAS-PE-200108E

PART OF HEXAGON

INTERGRAPH®

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.

b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, http://www,bis.doc.gov/complianceandenforcement/liststocheck.htm, the U.S. Department of Treasury Specially Designated Nationals List, www.treas.gov/offices/enforcement/ofac/http://www.pmddtc.state.gov/compliance/debar.html, and the U.S. Department of State Debarred List.

c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, Intergraph Smart, PDS, SmartPlant, SmartMarine, FrameWorks, I-Sketch, IntelliShip, ISOGEN, SmartSketch, SPOOLGEN, SupportManager, and SupportModeler are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Microsoft and Windows are registered trademarks of Microsoft Corporation. MicroStation is a registered trademark of Bentley Systems, Inc. Other brands and product names are trademarks of their respective owners.

Contents

What's New in PV Elite and CodeCalc

Below are the new features for Version 2014 (16.0) of PV Elite and CodeCalc. New features and improvements come directly from your comments, as well as updates to the previous version. This version of the software includes any new features or revisions made to the software for service packs on the 2013 version.

Code Updates and Analysis Changes

- ASME 2013 VIII-1 Edition updates incorporated
- ASME 2013 VIII-2 Edition updates incorporated
- ASME 2013 Section II Part D Edition updates added (included hundreds of changes and additions)
- PD 5500 2013 Addenda changes added
- BS EN 13445-3:2009+A1:2012 updates incorporated
- ASCE-2010 Wind/Seismic codes added
- IBC 2012 Wind/Seismic codes added
- NBC 2010 Wind/Seismic codes added

Internationalization Updates

- EN 1092 Flange Pressure Temperature Ratings updated (including PN 2.5 series)
- EN 1092 Flange Dimensional Data updated
- South African Wind Code added
- South African Seismic Code added
- User customizable G-loading option added

Productivity Enhancements

- Automatic File Backups added
- A new testing Program is included
- Smart Selections for ANSI/EN-1092 Flange Class/Grade Groups
- Updated Help documentation

Analysis Features

- TEMA Exchanger Analysis added back in CodeCalc software
- ASME Appendix 9 Type 2 Jackets added
- PD 5500 Type 2 Jackets added
- Nozzle FEA via NozzlePRO added back in CodeCalc software
- Direct UI support for EN-13445 Cold Spun Heads

QA Validation and Verification Additions

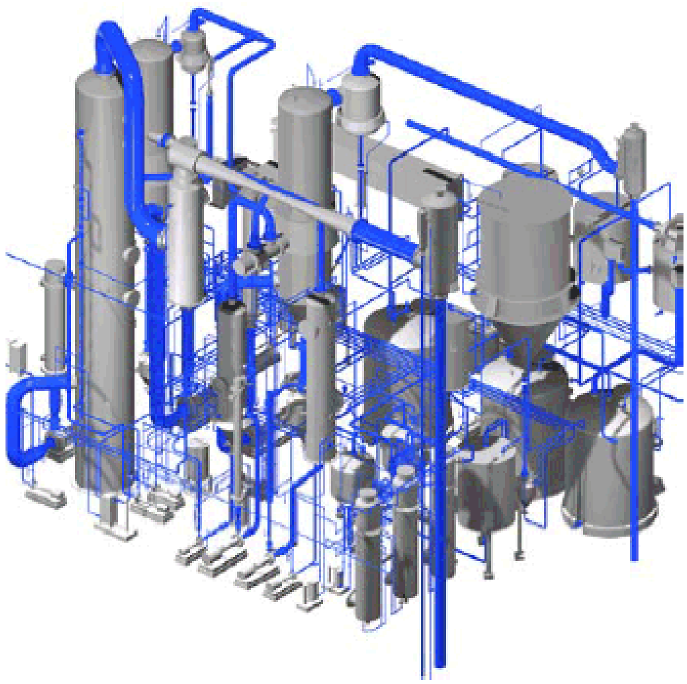
- The PV Elite development team continues to add new QA sample benchmark problems to the benchmark solution set. In addition, QA Test now includes many new EN nozzle tests. PV Elite 2014 was benchmarked against ASME PTB-4 (ASME Section VIII-Division 1 Example Problem Manual). The Verification and Quality Assurance Manual was updated to include tables of comparison for various sample problems.

SECTION 1

PV Elite Overview

PV Elite consists of nineteen modules for the design and analysis of pressure vessels and heat exchangers, and assessment of fitness for service. The software provides the mechanical engineer with easy-to-use, technically sound, well-documented reports. The reports contain detailed calculations and supporting comments that speed and simplify the task of vessel design, re-rating, or fitness for service. The popularity of PV Elite is a reflection of Intergraph CADWorx and Analysis Solutions' expertise in programming and engineering, and dedication to service and quality.

Calculations in PV Elite are based on the latest editions of national codes such as the ASME Boiler and Pressure Vessel Code, or industry standards such as the Zick analysis method for horizontal drums. PV Elite offers exceptional ease of use that results in dramatic improvement in efficiency for both design and re-rating.



PV Elite features include:

- A graphical user interface allowing you to add model data while seeing the vessel elements as they are added.
- Horizontal and vertical vessels of cylinders, conical sections, and body flanges, as well as elliptical, torispherical, hemispherical, conical, and flat heads.
- Saddle supports for horizontal vessels.
- Leg and skirt supports at any location for vertical vessels.
- Extensive on-line help.

- Dead weight calculation from vessel details such as nozzles, lugs, rings, trays, insulation, packing, and lining.
- Wall thickness calculations for internal and external pressure according to the rules of ASME Section VIII Divisions 1 and 2, PD 5500, and EN-13445.
- Stiffener ring evaluation for external pressure.
- Wind and seismic data using the American Society of Civil Engineers (ASCE) standard, the Uniform Building Code (UBC), the National (Canadian) Building Code, India standards, as well as British, Mexican, Australian, Japanese, and European standards.
- A user-defined unit system.
- A complete examination of vessel structural loads, combining the effects of pressure, dead weight, and live loads in the empty, operating, and hydrotest conditions.
- Logic to automatically increase wall thickness to satisfy requirements for pressure and structural loads, and introduce stiffener rings to address external pressure rules.
- Structural load evaluation in terms of both tensile and compressive stress ratios to the allowable limits.
- Detailed analysis of nozzles, flanges, and base rings.
- Material libraries for all three design standards.
- Component libraries containing pipe diameter and wall thickness, ANSI B16.5 flange pressure vs. temperature charts, and section properties for AISC, British, Indian, Japanese, Korean, Australian and South African structural shapes.
- Thorough and complete printed analysis reports, with definable headings on each page. Comments and additions may be inserted at any point in the output.

What Distinguishes PV Elite From our Competitors?

Our staff of experienced pressure vessel engineers are involved in day-to-day software development, software support, and training. This approach has produced software that closely fits today's requirements of the pressure vessel industry. Data entry is simple and straightforward through annotated input fields. PV Elite provides the widest range of modeling and analysis capabilities without becoming too complicated for simple system analysis. You can tailor PV Elite through default settings and customized databases. Comprehensive input graphics confirm model construction before analysis is made. The software's interactive output processor presents results on the monitor for quick review or sends complete reports to a file, printer or Word document. PV Elite is an up-to-date package that not only uses standard analysis guidelines, but also provides the latest recognized opinions for these analyses.

PV Elite is a field-proven engineering analysis program and is a widely recognized product with a large customer base and an excellent support and development record.

What Can Be Designed?

PV Elite can design and analyze:

General Vessels

Enables users to perform wall thickness design and analysis of any vessel for realistic combinations of pressure, deadweight, nozzle, wind and seismic loads in accordance with ASME Section VIII Division 1 rules, Division 2 rules, PD 5500, and EN-13445. These calculations address minimum wall thickness for pressure and allowable longitudinal stress (both tension and compression) in the vessel wall for the expected structural load combinations.

Complete Vertical Vessels

Enables users to define vessels supported by skirts, legs or lugs for complete dead load and live load analysis. Stacked vessels with liquid are also addressed. Enables users to specify Hydrotest conditions for either vertical or horizontal test positions. Vessel MAWP includes hydrostatic head and ANSI B16.5 flange pressure limitations.

Complete Horizontal Vessels

Enables stress analysis of horizontal drums on saddle supports using the method of L. P. Zick. Results include stresses at the saddles, the midpoint of the vessel and in the heads.

SECTION 2

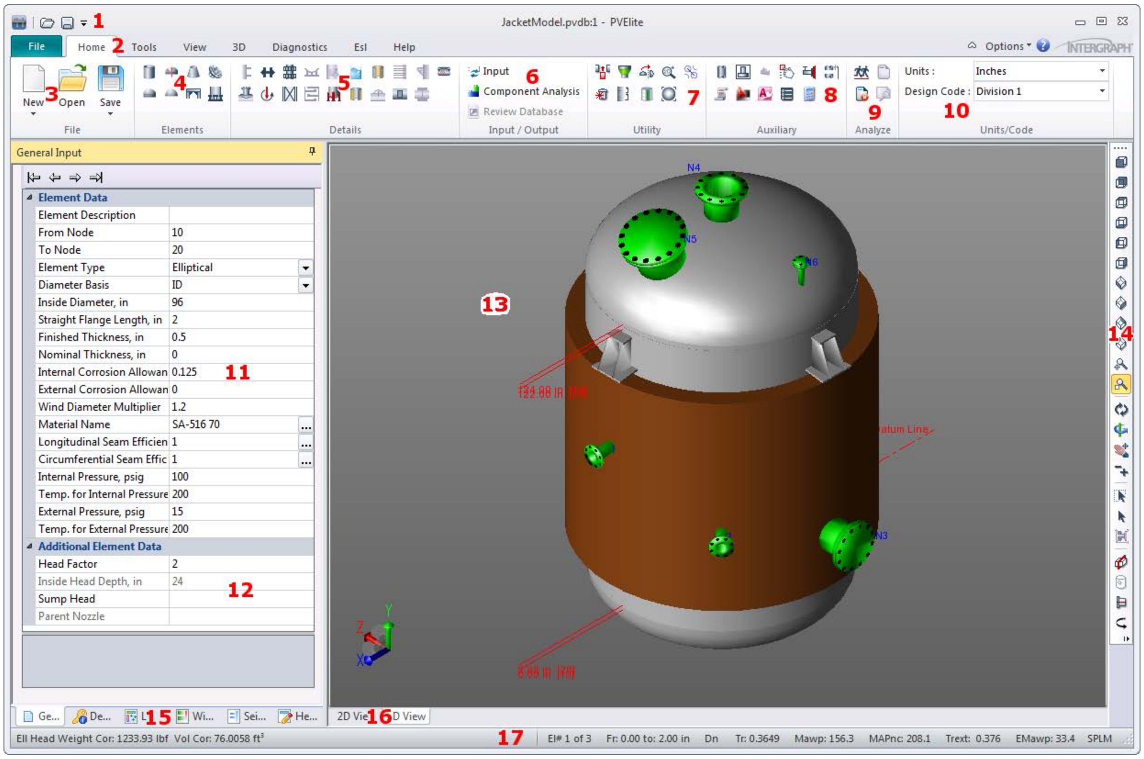
Getting Started and Workflows

This section focuses on getting started and the fundamental workflows PV Elite.

- **Input** - Enter information required to define the vessel, its service requirements, and its design guidelines.
- **Analysis** - Translate the input data with design and analysis algorithms, apply the rules of the appropriate code or standard, and generate results.
- **Output** - Present a comprehensive final report of the results.

Understanding the Interface

The main PV Elite window has a quick access toolbar in the top left corner (indicated with a red 1 in the picture below), which you can customize with the commands that you use most. Directly below the quick access toolbar is a series of ribbons (indicated by the red 2) and panels. On the Home tab, are data file command (3 *File Panel* (on page 48)), add elements (4 *Elements Panel* (on page 48)), and add details (5 *Details Panel* (on page 49)) to the current element.



PV Elite breaks a vessel into an assemblage of individual elements—heads, shells, cones, body flanges, and/or a skirt—and the components on these elements. A quick look at the screen (above) shows the data (near red 11) defining one element in the graphic view (red 13). Except for **From Node** and **To Node**, the data is common to all vessel wall thickness calculations. The

From Node and **To Node** inputs are necessary to assemble the individual elements into the complete vessel and are automatically assigned by PV Elite. A complete vessel is required if all dead and live loads are to be included in the design or analysis. However, PV Elite will run wall thickness calculations on elements without constructing the entire vessel.

The body of the screen contains either two or three areas - a table of the Element Data (red 11 above), a table of the Element Additional Data (red 12 above when required) and the graphic area which contains an image of the current status of the entire vessel or the current element (red 13 above). A status bar displays (red 17) across the bottom of the screen and displays the element count, the position and orientation of the current element, quick internal pressure calculations for the current element.

When you click in the data areas (red 11 and 12), the **Tab** key moves the highlight (and control) through its input cells. In most element data areas, press **Enter** to register the data and move the focus to the next field. The exception is at combo boxes where clicking the arrow displays the available choices. Throughout the program, **F1** displays help for the highlighted data item. After you are familiar with these screen controls, a combination of mouse and keystroke commands will provide the most efficient navigation through the program.

Some of the data input in PV Elite is controlled through a data grid (red 11 and 12). To enter the data, click the mouse on the data text, such as **Inside Diameter**, and type the input value. The cursor will not blink over the numeric/alphanumeric values until typing has begun. After the data is entered, press **Enter** or **Tab** to proceed. The arrow keys can also be used to navigate between the input fields.

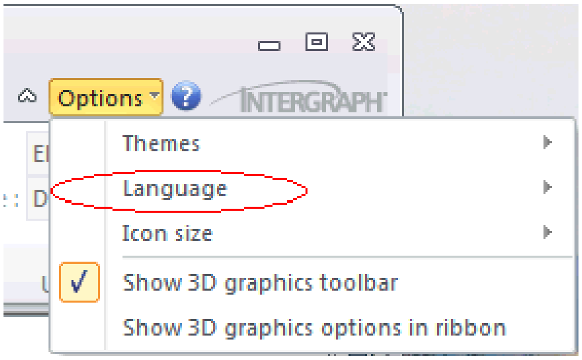
NOTE The right mouse button is used to select vessel details on the vessel graphic. Combo boxes have the down arrow button at the right end of the input cell.

When the 3D View (red 13) is active, a few more keys are available. No special highlight appears, but the string PgUp/PgDn/Home/End displays at the bottom graphics area. This indicates that these keys are now active. The image in the graphics area shows the current state of the input for the vessel model with its elements and the details on these elements. Switch between 2D and 3D views using the tabs at the bottom of the screen (red 16).

When the 2D View is active, one of the elements is highlighted. The element data (**Element Data** 11 and **Element Additional Data** 12) shown on the screen defines this element. By pressing **Page Up** or **Page Down**, the highlight changes from one element to the next through the vessel. Press the **Home** and **End** keys to move the highlight to the first and last elements in the vessel. Also, you can click the left mouse button while selecting the element to highlight it. After an element is highlighted, detail information for that element may be accessed. With the mouse, click the right mouse button for the existing detail image to display. To add details to the current element, click the appropriate detail on the **Details** panel (red 5) and enter the necessary data.

Set Interface Language

The PV Elite interface is available in several languages. Use **Options > Language** to select the language that you want to use.



La interfaz de PV Elite está disponible en varios idiomas. Utilice **Opciones> Idioma** para seleccionar el idioma que desea utilizar.

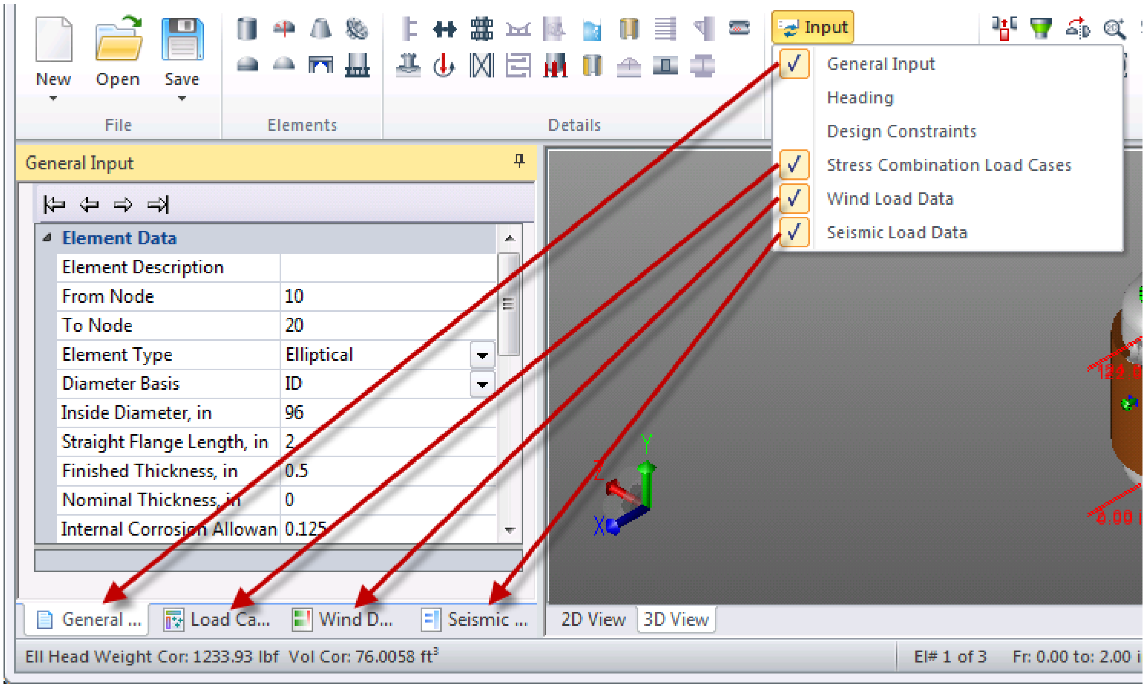
L'interface PV Elite est disponible en plusieurs langues. Utilisez **Options> Langue** pour sélectionner la langue que vous souhaitez utiliser.

L'interfaccia PV Elite è disponibile in diverse lingue. Utilizzare **Opzioni> Lingua** per selezionare la lingua che si desidera utilizzare.

O interface de PV Elite está disponível em vários idiomas. Use **Opções> Idioma** para selecionar o idioma que você deseja usar.

Input Processors

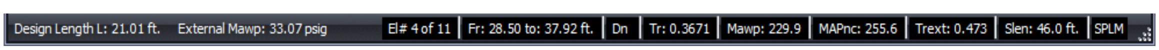
The **Input** items are used to define the other types of data that might be necessary for an analysis: design constraints, report headings, general input data, and live (**Wind Loads** and **Seismic Loads**) load definitions. Each command activates an input data tab in the bottom-left corner of the window. You can re-order the tabs as you like. The **Design Constraints** data is important because this is where the overall analysis for this vessel is defined and controlled.



The **Component Analysis** option allows you to enter data and analyze without building a vessel. These are Intergraph's CodeCalc analysis modules, some of which are not incorporated directly into PV Elite. CodeCalc, Intergraph's popular vessel component analysis package is included in PV Elite.

Status Bar

The status bar across the bottom of the window displays information about your vessel in real time. Values that display in red need attention. You can show or hide the status bar using the Toggle Status Bar option on the **View** tab.



Wgt Cor - Displays the corroded weight.

Vol Cor - Displays the corroded volume.

t/L - Displays the thickness to length ratio.

OutDepth - Displays the head depth for a torispherical head.

El# - Displays the current element location relative to the start of the vessel. For example, a value of **4 of 11** means the active element is the fourth element from the start of the vessel and that there are eleven elements in total that define the vessel.

Fr - Displays the starting and ending distances from the Datum point for the current element.

Left / Right / Up / Down - Displays the element orientation.

Tr - Displays the computed thickness for the internal pressure.

Mawp - Displays the maximum allowable working pressure of the vessel and flange.

MAPnc - Displays the maximum allowable pressure in a new and cold condition.

Trext - Displays the computed thickness for the external pressure.

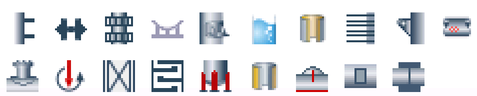
EMawp - Displays the external maximum allowable working pressure.

Slen - Displays the maximum length between ring stiffeners. When **>>>** displays, no reinforcing ring is required for this element for external pressure purposes. For more information, see Stiffening Ring.

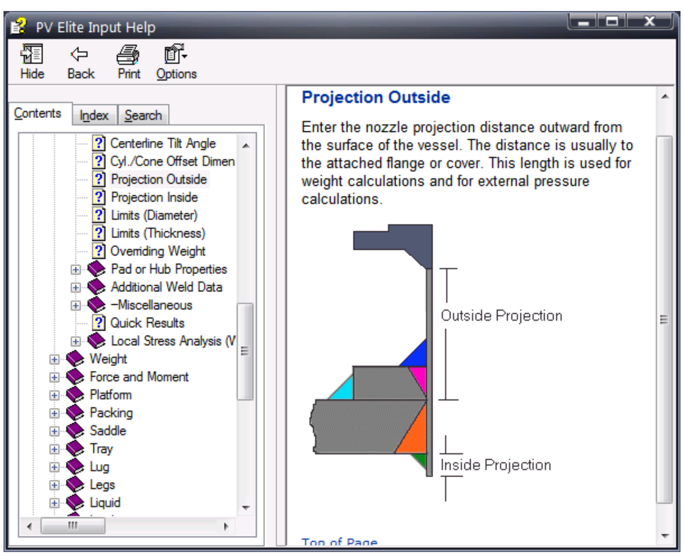
Adding Details

With the elements defined, enough information exists to run through the pressure calculations but the total vessel weight is not yet set. Much of this information is specified as element details. Nozzles, insulation, operating fluid, platforms and the like are all entered as details on the various elements. PV Elite calculates the weight of each of these items and account for them in the various analyses. Details such as saddles, lugs and legs are also used to locate support points on the vessel—important data for load calculations.

Details can only be specified on the current element. To enter the first detail, highlight (make current) the element that will hold the detail, and click the appropriate **DETAIL** command. Allowing the cursor to rest on the toolbar button displays a tool tip definition of the button. Select the detail and enter the data in the screen that follows. For more information, see *Details Panel* (on page 49).



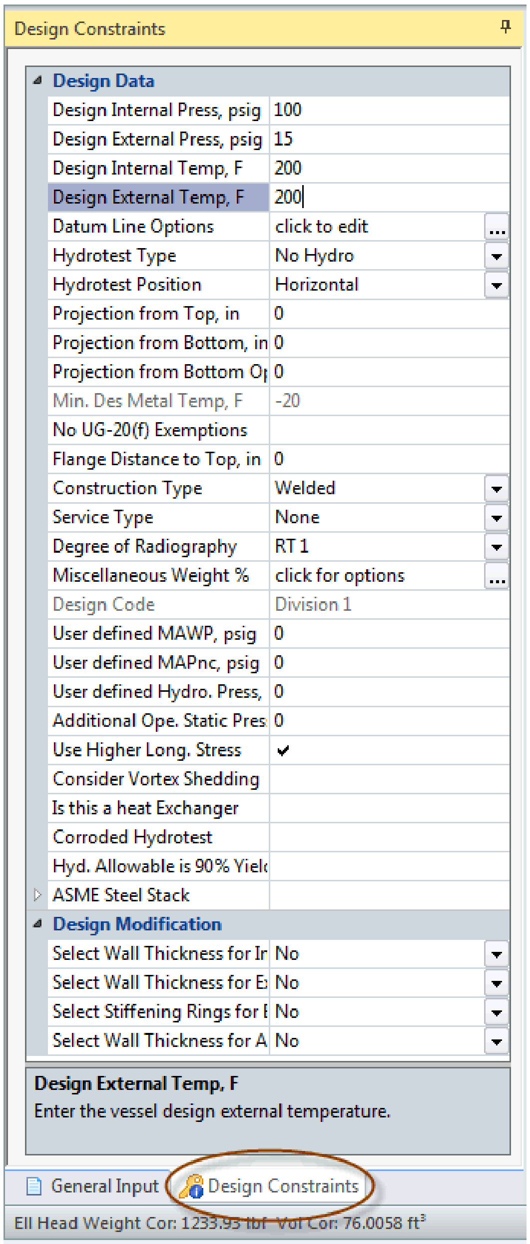
Use the **Help** button on the detail screen or press [**F1**] to learn more about the requested data. Define all details necessary to develop the proper total vessel load.



Specifying Global Data - Loads and Design Constraints

Although default values allow the analysis to proceed, other data should be set before the analysis continues. These data are the required live loads and design constraints, and the optional vessel identification and report headings. These data are accessed and entered through the **Input** panel on the **Home** tab. The **Headings** input allows the specification of three lines of data, which appears at the top of each page in the printed output. The heading data also includes title page entry, which appears at the beginning of the input echo report.

Select **Input > Design Constraints** and then select the **Design Constraints** tab to display the design data.



Design Data includes vessel identification along with items that affect the design and analysis of the vessel; items such as type of hydrostatic testing and degree of radiographic examination appear here. It is important to note that this is where the design code is set - Division 1, Division 2, PD:5500 or EN 13445.

The **Design Modification** area holds four inputs that control the redesign of the vessel should the user-entered wall thickness be insufficient for the analyzed loads. If a box is checked, the software increase the element's wall thickness so that it meets or exceeds the requirements for that load category. There are four boxes for three load types: one box for internal pressure, two boxes for external pressure (either increase the wall thickness or locate stiffener rings along the vessel to satisfy the buckling requirements), and one box for the variety of structural loads that develop longitudinal stresses in the vessel wall. The software provides the option of rounding up a required thickness to a nominal value (such as the next 1/16 inch or 1 mm) in the *Configuration* (on page 200) dialog box.

The **Load Cases** tab displays nineteen default structural load cases for the analysis. These cases cover the extent of structural loads on the vessel wall. Each case contains a pressure component (axial) 1, a weight component (both axial and bending), and a live load component (bending). The axial stresses are combined with the bending stresses to produce a total stress in the vessel wall. Both tensile and compressive stresses are compared to their allowable limits. Refer to the table below for a definition of terms used in the **Load Case** input.

NOTE These pressure calculations should not be confused with those used for the wall thickness requirements defined in ASME Section VIII and PD:5500. Here, internal and hydrostatic pressures are used to calculate a longitudinal, tensile stress in the vessel wall and the external pressure a similar compressive stress in the wall.

Pressure	**Weight**	**Live Load**
NP - No Pressure	EW - Empty Weight	WI - Wind
IP - Internal Pressure	OW - Operating Weight	EQ - Earthquake
EP - External Pressure	HW - Hydrostatic Weight	HI - Wind at Hydrostatic Weight conditions
HP - Hydrostatic Pressure	CW - Empty Weight No CA	HE - Earthquake at Hydrostatic Weight conditions
		VF - Vortex Shedding Filled
		VO - Vortex Shedding Operating
		VE - Vortex Shedding Empty
		WE - Wind Bending Empty New and Cold
		WF - Wind Bend Filled New and Cold
		CW - Axial Weight Stress New and Cold
		FS - Axial Stress, Seismic
		PW - Axial Stress Wind

Nozzle Design Options are used to set the overall pressure requirements for the nozzles on this vessel and also to include the maximum allowable pressure - new and cold (MAP nc) case in the nozzle checks. The **Installation | Misc. Options** option displays a screen to specify where certain vessel details will be added - either at the fabrication shop or in the field. This data is used to properly set the detail weights for the empty and operating conditions.

Wind and earthquake information is entered on the **Wind Data** or **Seismic Data** tabs. PV Elite generates live loads based on the criteria established by one of many standards, including the American Society of Civil Engineers (ASCE), the Uniform Building Code (UBC), the (Canadian) National Building Code (NBC), and the Indian National Standard. Wind loads can also be specified directly as a wind pressure profile. PV Elite references these codes for live loads only. ASME Section VIII or PD:5500 rules apply for all other calculations. The screen below shows the data required for the default codes. PV Elite uses these criteria to set the magnitude of the live load and bending moment on each element of the vessel.

After the element, detail, and global data is entered and checked, the model is ready for error processing and analysis.

Error Checking

The input processor makes many data consistency checks during the input session. For example, the processor creates an error message if you try to specify a nozzle 20 feet from the bottom of a 10-foot shell element. However, not all data can be confirmed on input so a general error processor is run prior to the analysis. This error processor can be run in a stand-alone from the **Analyze** panel, *Error Check Only* (on page 196).

In addition to the notes that are presented on the screen during error checking, these error messages appear in the output report and are accessible through the output review processor.

NOTE As with all engineering and designing, the vessel analyst must use common sense to insure the model is basically correct. This is a great advantage of the 3D graphics as it reveals obvious errors.

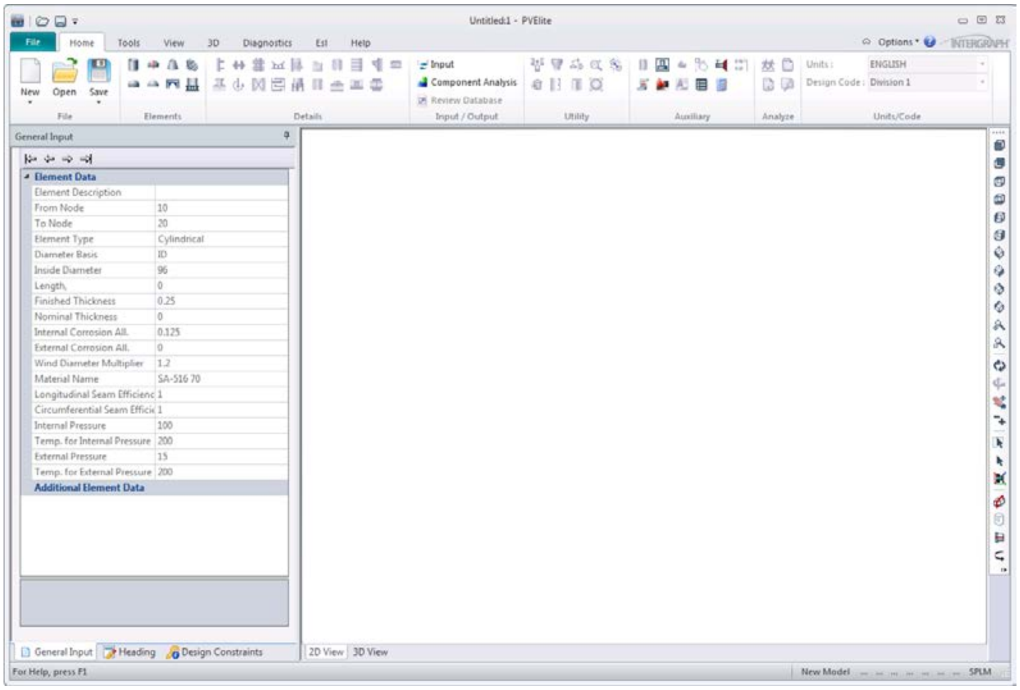
Modeling Basics

PV Elite breaks a vessel into an assemblage of individual elements—heads, shells, cones, body flanges, and/or a skirt—and the components on these elements. Vessels are defined one element to the next - from bottom to top for vertical vessels and from left to right for horizontal vessels.

Before starting PV Elite, most users collect the necessary data for the vessel design or analysis. Collecting data to define these elements before starting the program is not required but it will make the most efficient use of the designer's time. Typical input items include actual or proposed values for vessel material, inside diameter, operating temperatures and pressures, wind and seismic site data, nozzle and ring location to name a few. If necessary, the input processor can be terminated at any time and restarted later if any missing data need be collected. With the program's graphic display of the vessel input, it is easy to recall the current state of an unfinished model or identify where data is missing or incomplete.

1. Start PV Elite by clicking the icon on the desktop or selecting the item from **Start** > **All Programs > Intergraph CAS > PV Elite > PV Elite**.

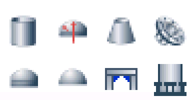
*PV Elite starts with a **Vessel Input** screen for the job called "Untitled."*



Defining the Basic Vessel

You should build vertical vessels from bottom to top and horizontal vessels from left to right. It is not necessary to build an entire vessel if only the thickness for pressure is needed. The Element Data (the grid on the left side of the interface) must be specified before the first element can be placed on the screen.

1. Start with the **Inside Diameter**, as both the **Node Numbers** and the **Element Type** are set by using the **Element** toolbar.



2. After the Element Data is entered, elements are quickly assembled one after another by clicking the **Element** toolbar and making any changes to the Element Data.

The complete vessel is created from the following elements (in their toolbar order):

- Shell
- Elliptical Head
- Torispherical Head
- Spherical Head
- Cone
- Welded Flat Head

- ANSI/Bolted Flange
- Skirt

If the vessel begins with a skirt element, it is a vertical vessel. Vertical vessels on legs and horizontal vessels start with a head element.

NOTE If that first head element is improperly oriented for the vessel in mind (horizontal or vertical), click **Flip Element Orientation** on the **Home** tab, **Utility** panel to correct the orientation. Later, if heads, body flanges, or cone elements show incorrect orientation, click **Flip Element Orientation** to fix the orientation.

After the second element is added, use the **Flip Model Orientation** on the **Tools** tab to flip the entire model flip between horizontal and vertical.

From Nodes and **To Nodes** values are automatically assigned by the software; they start with node 10 and are incremented by 10 throughout the model. The element data set at the beginning of the session carries forward from one element to the next. Any data changes on the last element carry forward onto any new elements that are added. The element data displayed belongs to the highlighted element in the vessel image.

1. Use the mouse to change the highlighted and displayed element by clicking on the element of interest.

   Data may be updated one element at a time but there are more efficient ways to change an item through several elements; for example, if the circumferential weld joint efficiency for the skirt (from node 10 to 20) is set at 0.7. If this value was not changed to 1.0 on the bottom head as it was created, this (incorrect) value is carried from one element to the next in the Build Mode to the top of the vessel element (such as, From Node 50 and To Node 60). In this situation, it is easiest to change the data on the bottom head element (20 to 30), and then click **Share** to "share" this item through the elements in the list with From Node 30 through From Node 50. Certain data is automatically "shared". Inside diameter, for example is automatically changed for all elements (stopping at cones) attached to the element where the change occurs. Some changes to the element data do not immediately appear on the vessel image.

2. To refresh the image, press **F5**.

Building a Heat Exchanger

This section provides the workflow of how to build a heat exchanger using PV Elite.

1. Launch **PV Elite**.

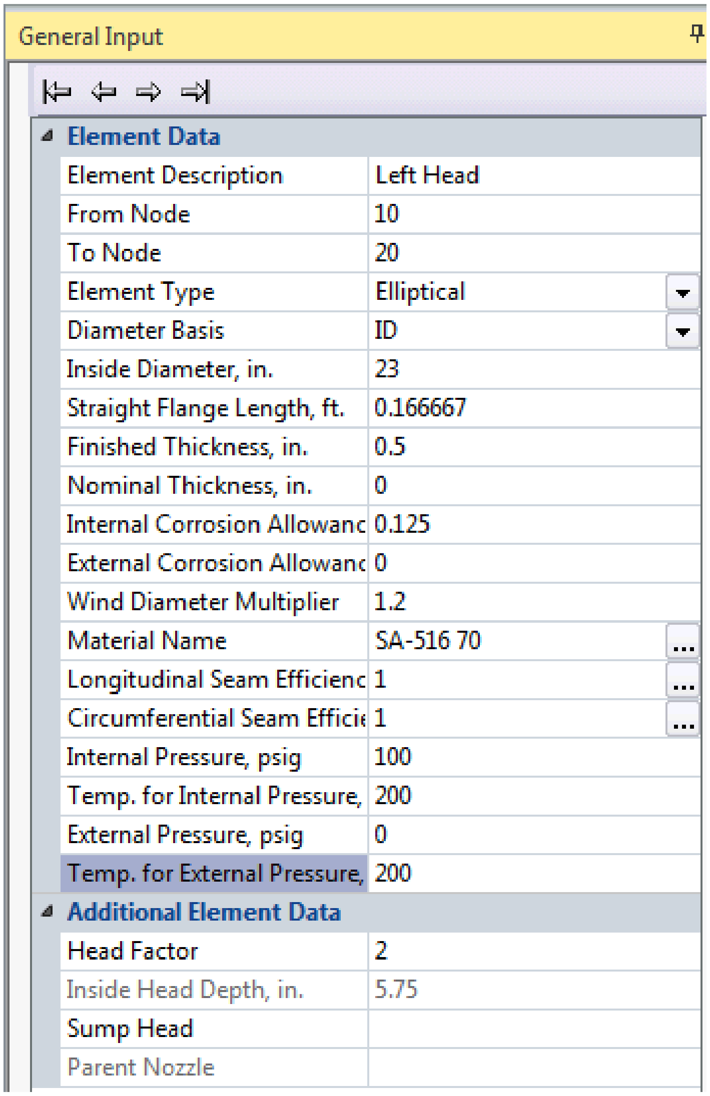
   *The **General Input** screen displays.*

2. Select an ellipsoidal head by clicking **Ellipse** .

3. Click **Flip Elements Orientation** on the **Home** tab to build the heat exchanger in the horizontal orientation.

4. Enter the information for the head exactly as shown below. Check your input before you move on. Remember to click on the text in the left column and then start typing. The cursor automatically moves to the right column for you to begin typing. Pressing **[F1]** displays the corresponding help information for that input.

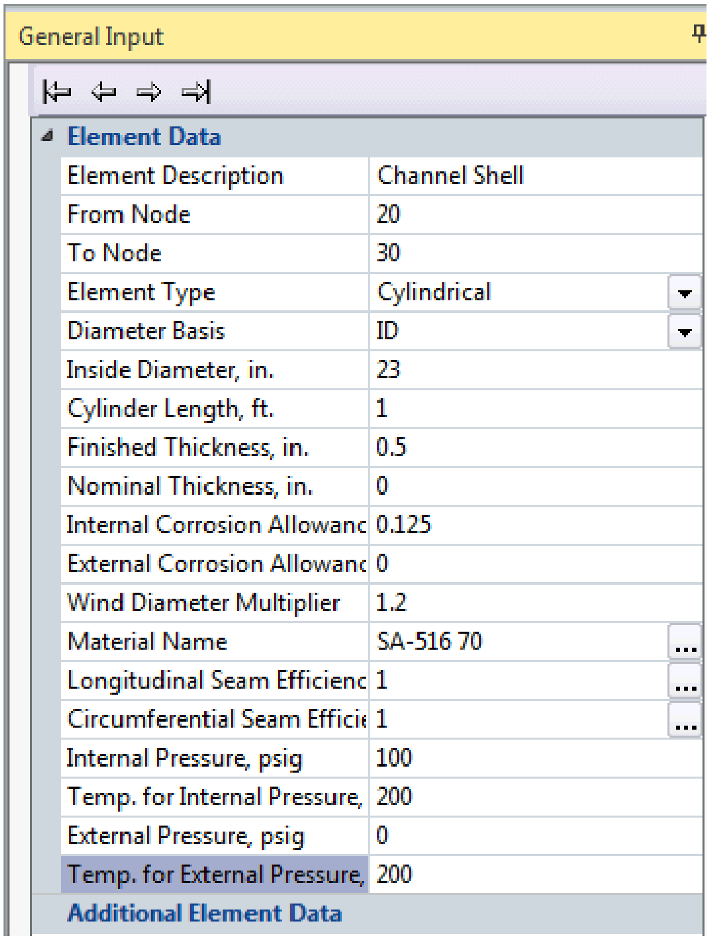
The screen should then look exactly like this when you are finished with your input.



5. When you are finished typing all the data, press **Enter** twice.

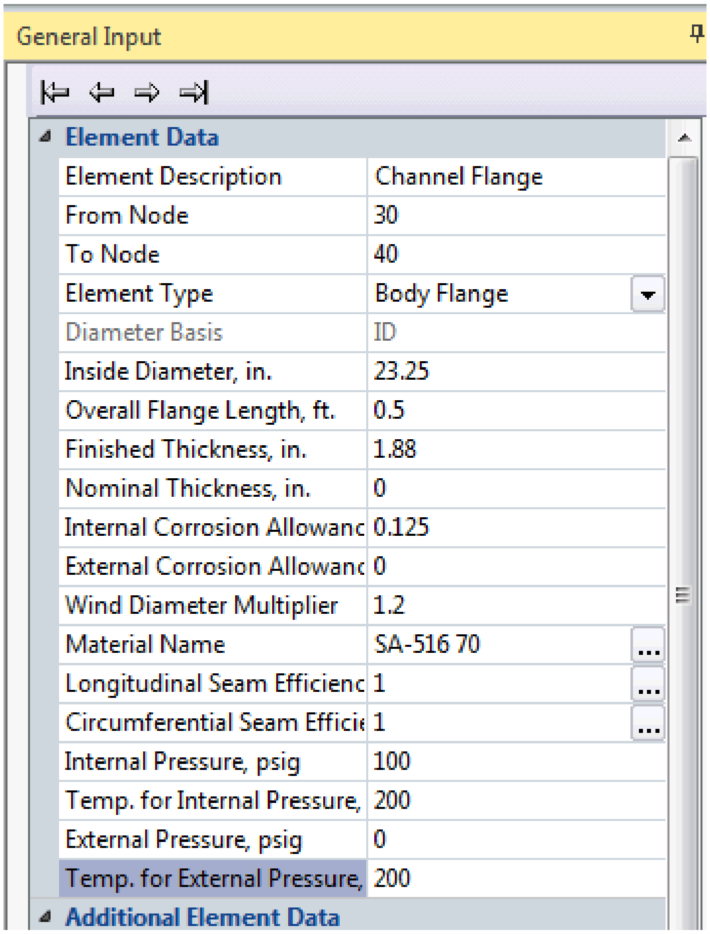
6. Click **Cylinder**  to add a cylinder to the head.

The screen should then look exactly like this when you are finished with your input.

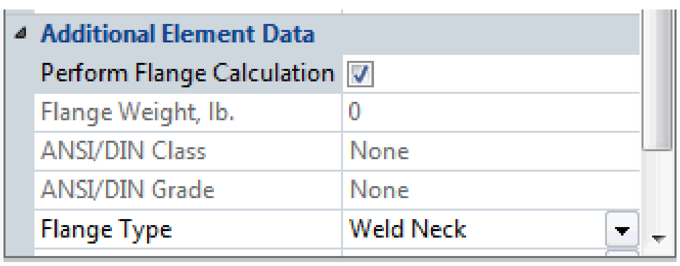


7. Click **ANSI/Bolted Flange** to add a body flange to the right hand end of the channel shell.

8. After adding the flange, enter all the values exactly as shown below.



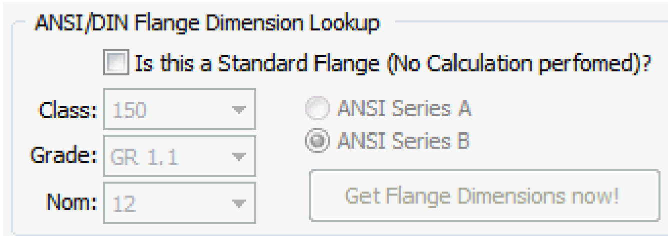
9. Select the **Perform Flange Calculation** check box.



*The **Flange** dialog box displays.*

You must correctly dimension the flange. To do this, change the flange as it appears to a 24 inch Class 150 flange, which will fit into the heat exchanger.

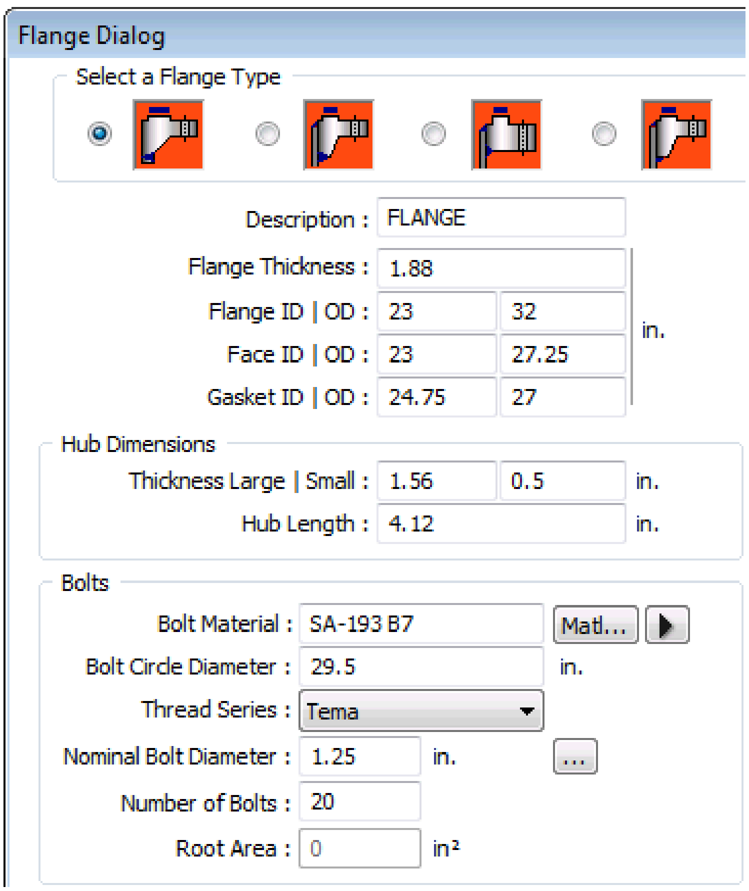
10. At the bottom of the new **Flange** dialog box you see a section that resembles this:



11. Select the **Is this a Standard Flange (No Calculation performed)?** check box.
12. Select 150 as the Class
13. Select 24 for the Nom.
14. Click **Get Flange Dimensions now!**.

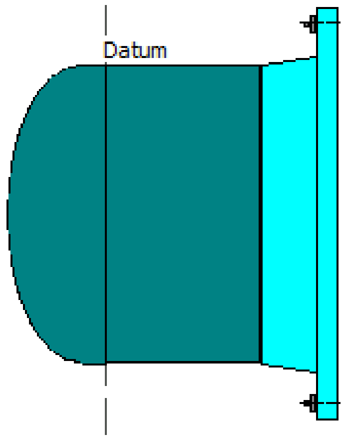
    The Flange screen is now set up for the 24 inch Class 150 dimensions and bolting.

15. Verify your screen looks like the following figure:



16. Click **OK** on the **Flange Dialog**.
17. Click **Save** to save your work. Use **My First Model** or something similar for the file name.

18. Your model should look similar to this.



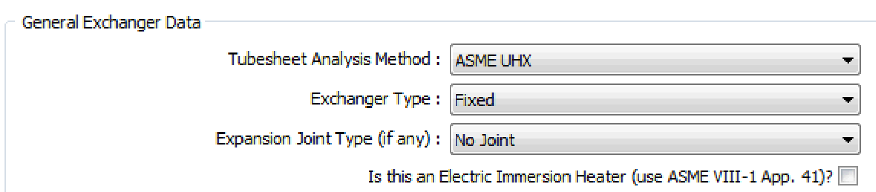
Up to this point, you have been using the normal PV Elite vessel building techniques for non-heat exchanger pressure vessels. You are now ready to start modeling the heat exchanger main elements, the tubesheets, tubes, and main shell that enclose the tube bundle.

1. Click **Tubesheet Analysis** .

   *The **Heat Exchanger Tubesheet Input** dialog displays.*

   You will now construct an ASME Section UHX exchanger, which requires a large amount of input data.

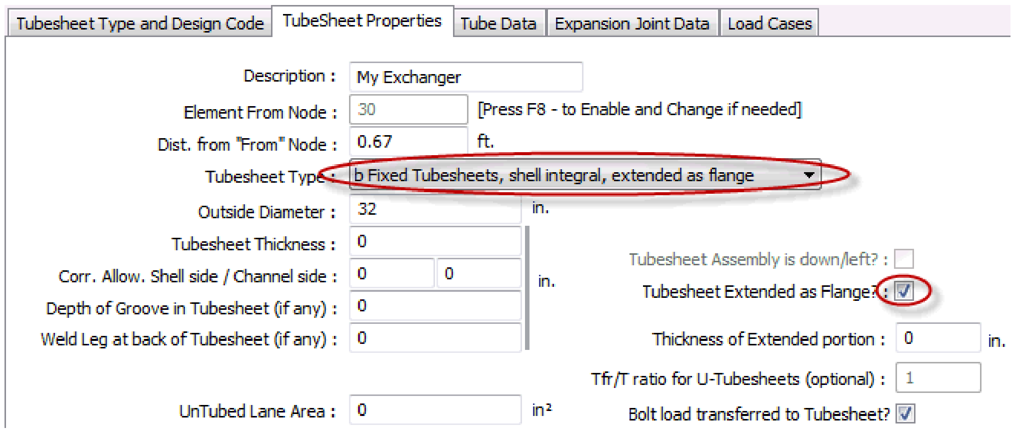
2. Select **Fixed** for **Tubesheet Analysis Method**:



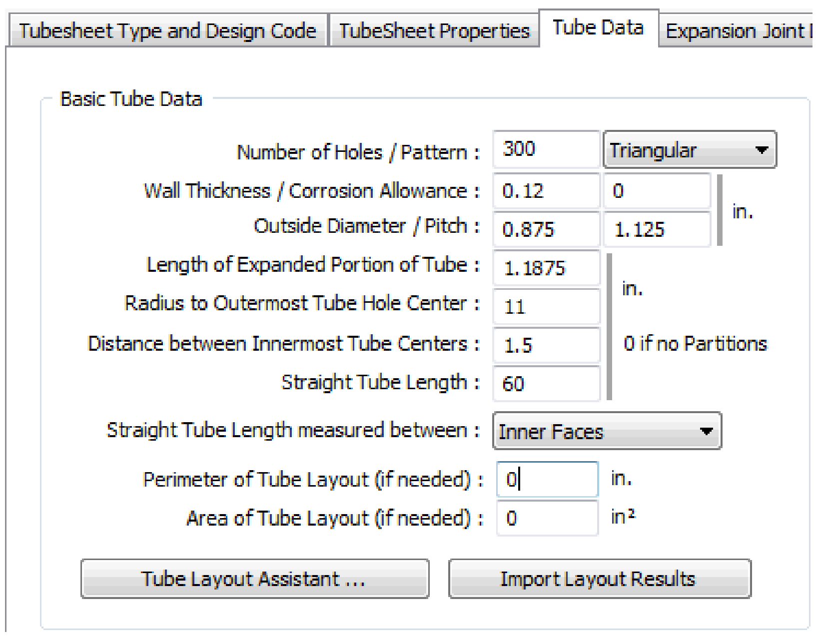
3. Select the **Tubesheet Properties** tab.

4. For **Tubesheet Type**, select **b Fixed Tubesheet, shell integral, extended as flange**.

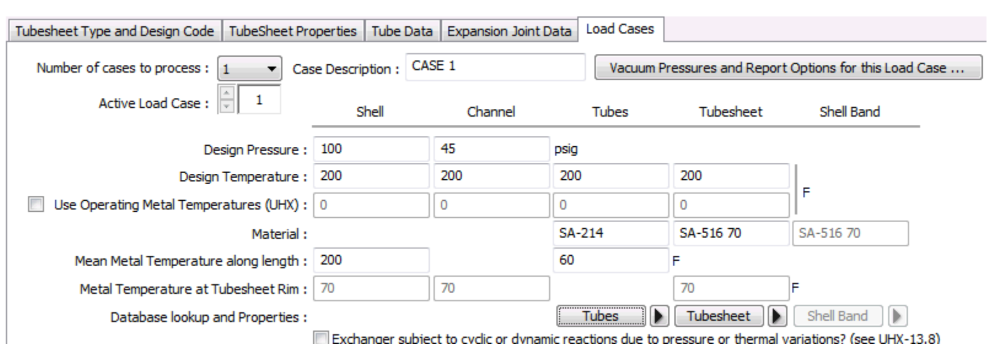
5. Select **Tubesheet Extended as Flange?**



6. Select the **Tube Data** tab and enter the information as shown below.

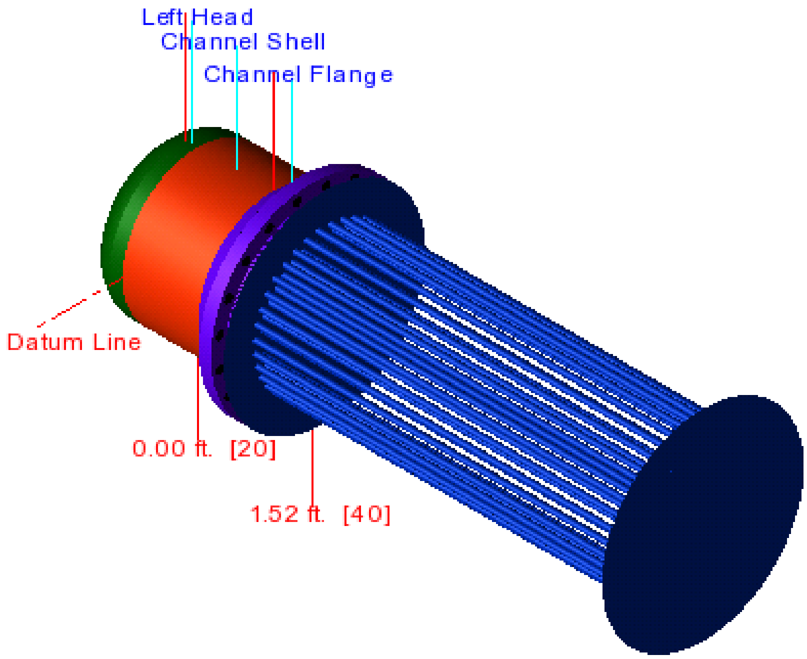


7. Select the **Load Cases** tab and enter the pressures and temperatures for the heat exchanger to complete the tube sheet and tubes data. You can enter multiple combinations of pressures and temperatures for heat exchangers.



8. Click **OK**.
9. Near the bottom-center of the window, select the **3D View** tab.

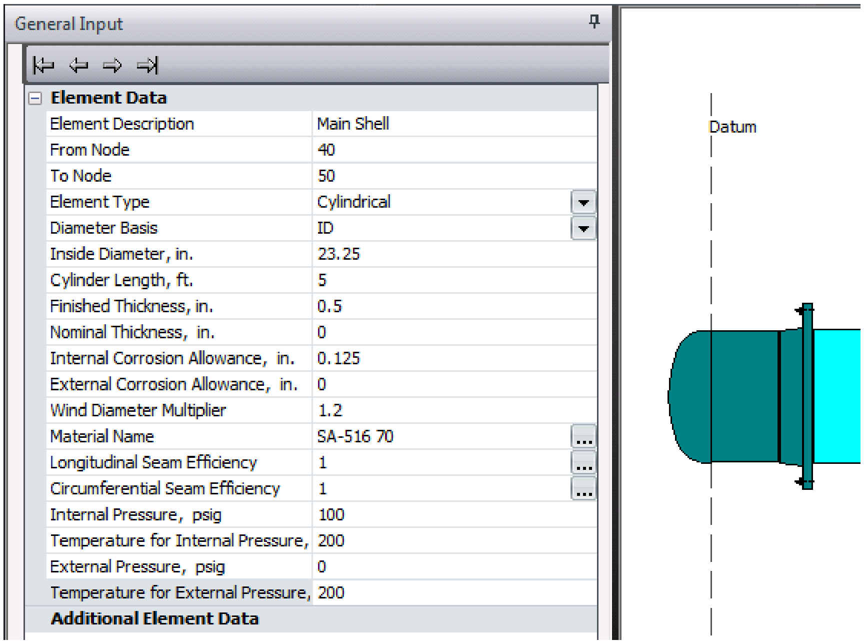
   Look at the 3D model on your screen, and it should resemble this figure.



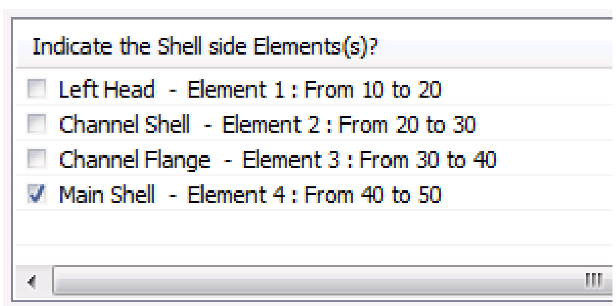
NOTE There are only two rows of tubes displayed. Because of the intensive nature of 3D graphics it is impractical to show hundreds of tubes.

Notice that there is no cylindrical shell between the two tubesheets. PV Elite cannot perform tubesheet analysis unless the shell is present because the thermal load from the shell is needed to complete this analysis. You must add the shell between the tubesheets. Before you do this, recall that that the tubes are 60 inches long in the **Heat Exchanger** dialog. This is the distance between the tubesheets. So for a good match, the outer shell must also be 60 inches long.

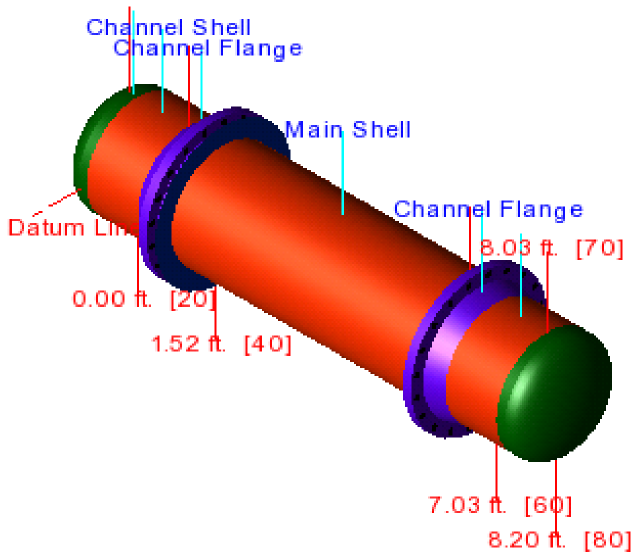
1. Click **Cylindrical** to add a cylindrical shell to your model. This shell is 60 inches or 5 feet long as discussed above. Verify your entries match those below:



2. Click **Tubesheet Analysis** again and go to the **Tubesheet Type and Design Code** tab.
3. Select the **Main Shell** check box, and then click **OK**.



4. All that remains is to add the body flange to the right end of the heat exchanger, then another channel shell and the final right channel head. By doing so, your model should look like this.



5. For the remaining steps, add other details to the elements using the commands on the **Detail** toolbar. You can add details such as saddle supports, nozzles, and loads.

Analysis

PV Elite can be used to confirm a safe design for a proposed or existing vessel. The program also provides direct design capabilities with which the wall thickness of individual elements is increased to meet the code requirements for internal and external pressure and longitudinal stress from a variety of dead and live loads. Whether or not the program changes wall thickness during the analysis is controlled through a **Design/Analysis Constraint** specification under **Design Modification**. For more information see *Design Constraints Tab* (on page 325).

A simple analysis run (no design) occurs when the flags under **Design Modification** are all unchecked. If any of these boxes are checked, the program automatically increases the wall thickness until the constraint is satisfied. Your input in the resulting output report is automatically updated to reflect any changes made during the analysis. In addition to wall thickness, a fourth flag can be set - **Select Stiffener Rings for External Pressure**. In this case, rather than increasing the wall thickness, stiffener rings are located along the vessel to satisfy the external pressure requirements. As with the wall thickness changes, these stiffener rings are added to the model input for this analysis.

PV Elite analyzes each element to determine the required wall thickness for internal and external pressure based on the Section VIII Division 1 rules, Division 2, PD:5500 or EN-13445 rules. The program then calculates the longitudinal stresses in the wall due to four categories of vessel loads: pressure, deadweight, deadweight moments from vessel attachments or applied loads, and moments due to the live loads - wind and earthquake. These four categories are set for three different load conditions: empty, operating, and hydrotest. The sensible combination of these various categories and conditions produce the default set of 19 load cases that are found in the **Design/Analysis Constraints** processor. For each load case, PV Elite will calculate the maximum longitudinal stress around the circumference of the elements and compare these values to the allowable stress for the material, both tensile and compressive. If stresses in the vessel wall exceed the design limits, PV Elite proceeds according to the design modification settings in the input.

After the software finishes a pass through the analysis, a check is made for any program design modifications. If PVElite changed any data, then the program automatically re-runs the complete analysis to review the impact of the changes.

There are several additional analysis controls that should be reviewed here. These controls, however, are more general in nature and are not defined for the individual job. Instead, these seven computational control directives are set for all jobs executed in the Data sub-directory. These controls are viewed and modified on the **Tools** tab, **Set Configuration Parameters**, for more information, see *Configuration* (on page 200).

Performing the Analysis

In the **Analyze** panel on the **Home** tab, are two options: **Error Check Only** and **Analyze**. Use the **Error Check Only** option immediately after any questionable data is entered. **Analyze** automatically performs an error check before the analysis starts. Comments from an error check can be examined using *Review Reports* (on page 196).

NOTE Errors must be corrected before the analysis can proceed.

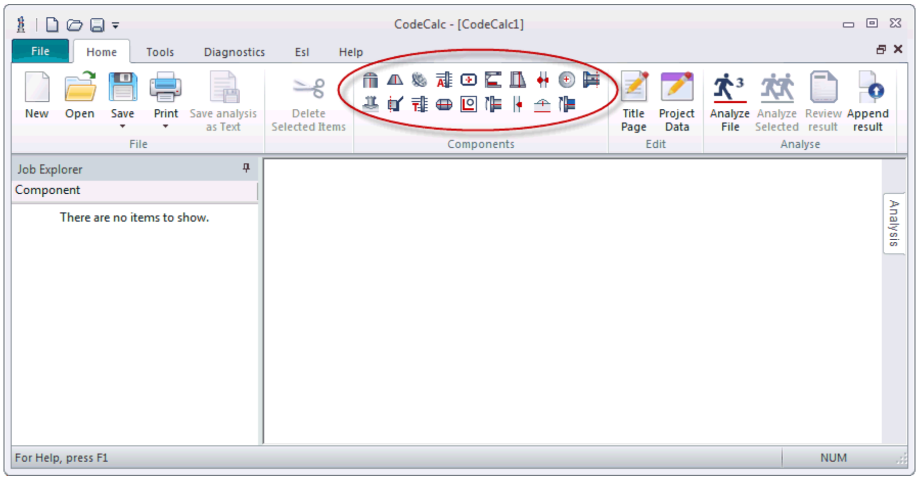
As the analysis proceeds, PV Elite displays the step or component being analyzed. If any *Design Modification (Design Constraints Tab)* (on page 332) were set, PV Elite resets the thickness to the necessary value and exports these increased thicknesses to all output reports and in all other calculations. For example, if the user-entered wall thickness of 1/2 inch is insufficient for the load and the design flag is enabled, the program will calculate the required thickness (for example, 5/8 inch) and replace the user-entered input value (1/2) in the output report with the calculated required thickness (here, 5/8). The program does not change the original model data. PV Elite checks the element wall thicknesses for the various pressure cases (internal, external, and hydrostatic) and then assemble the axial and bending loads to construct each load case defined in the Global Design data. PV Elite also calculates the longitudinal stress on both sides of the vessel (for example, both *windward* and *leeward* for loads with wind) and compare the calculated stresses with the allowable stresses, both tensile and compressive. PV Elite displays the *windward* or *leeward* side stress, which is closest by ratio to the allowable limit, again either tension or compression.

After the analysis is complete, the *Review Reports* (on page 196) processor displays the results of the analysis on the screen.

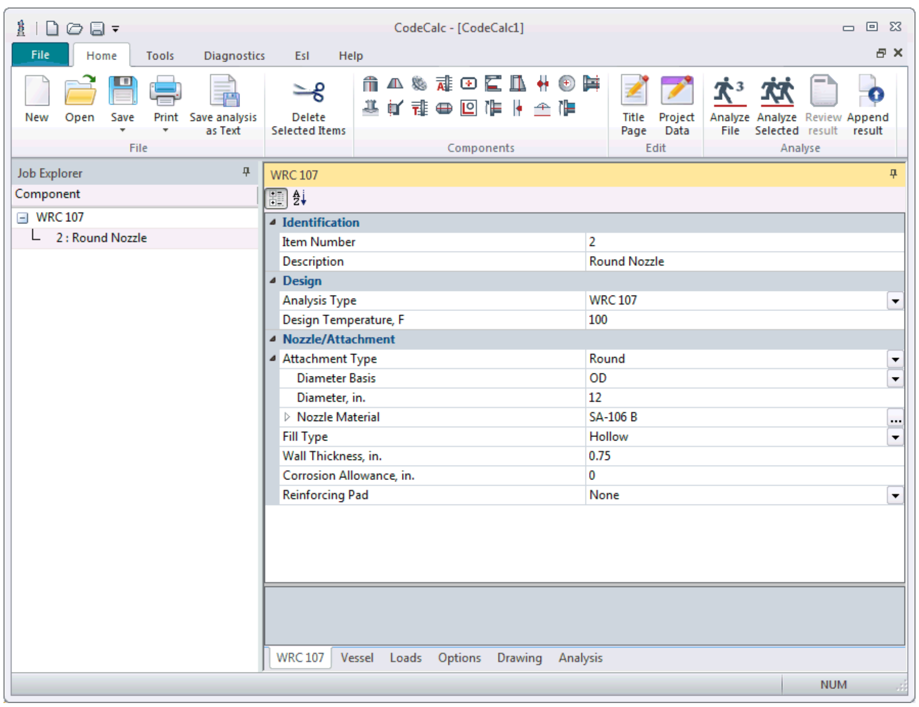
Design and Analysis of Vessel Details

At this point in the analysis the vessel details have been defined only so that their weights could be included in vessel calculations. With the structural analysis of the vessel complete and the wall thickness set, vessel details can be evaluated.

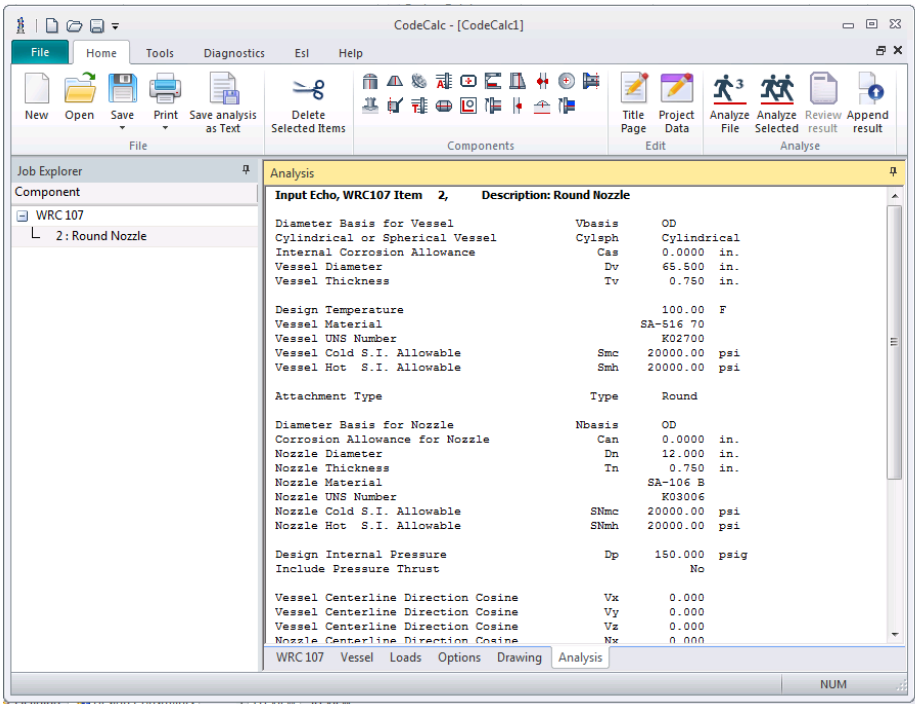
To access the input processor for these vessel details, click **Component Analysis** on the **Home** tab. This activates CodeCalc. The component selection is available on the **Home** tab.



WRC 107/537 Input Screen



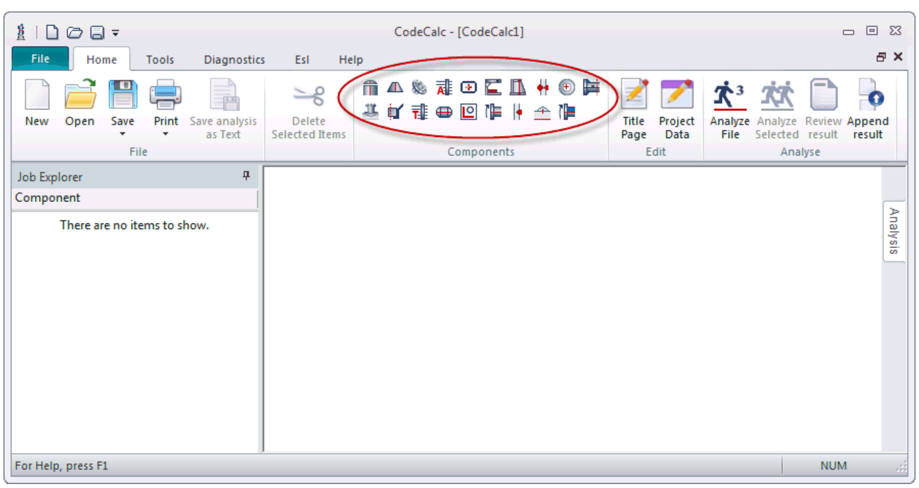
WRC 107/537 Results



Analyzing Individual Vessel Components Details

PV Elite provides for the analysis of a variety of vessel components that are not included in the overall vessel analysis: Appendix Y Flanges, Floating Heads, Lifting Lug, Pipe & Pad, WRC 107/537 and 297, Thin Joints, Thick Joints, ASME Tubesheets, TEMA Tubesheets, Halfpipe Jackets, Large Openings, and Rectangular Vessels.

To access the input processor for these vessel details, click **Component Analysis**  on the **Home** tab. This activates CodeCalc. The component selection is available on the **Home** tab.

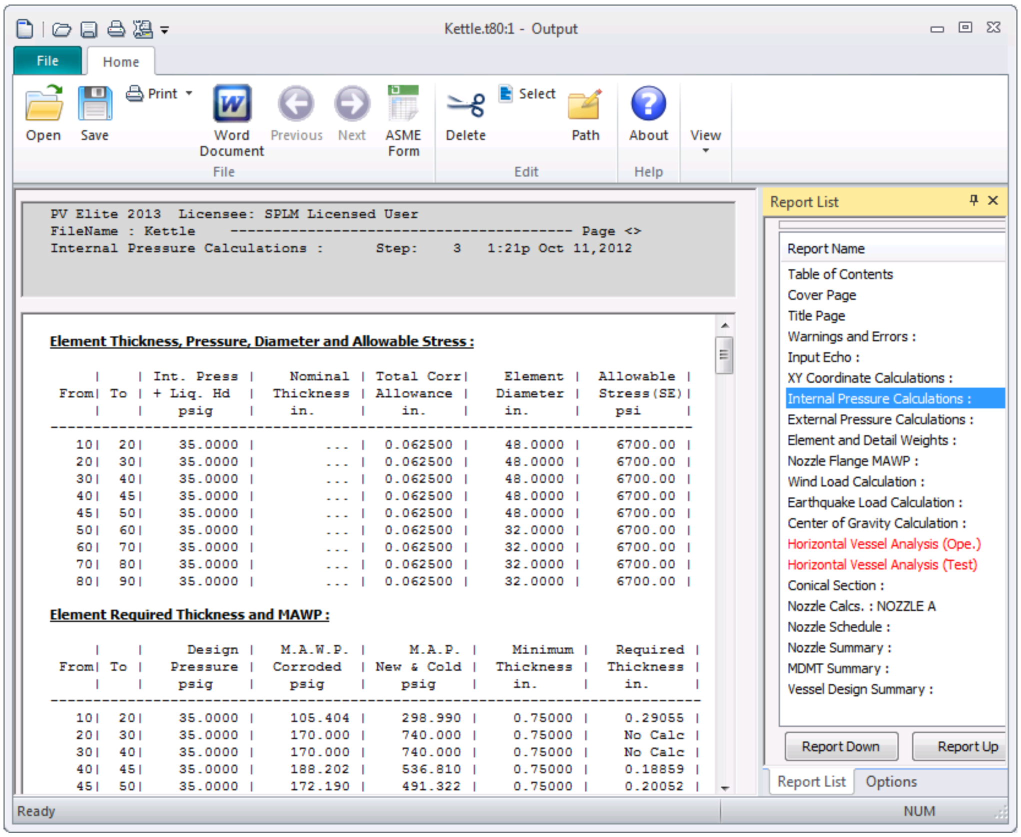


Output Review and Report Generation

Output from PV Elite analysis is stored in a binary data file that has the same name as the input file but with a ".T80" extension. Use *Review Reports* (on page 196)  to review every report contained in the output from input echo through stress reports.

Select the report to view from the **Report List**. Reports can be reviewed on the screen, sent to a printer, or sent to a file for review later.

Each analysis module creates its own report in the output data file. Most of the reports take the form of tables with the rows related to the elements and the columns holding the values such as thickness, MAWP, and stress.

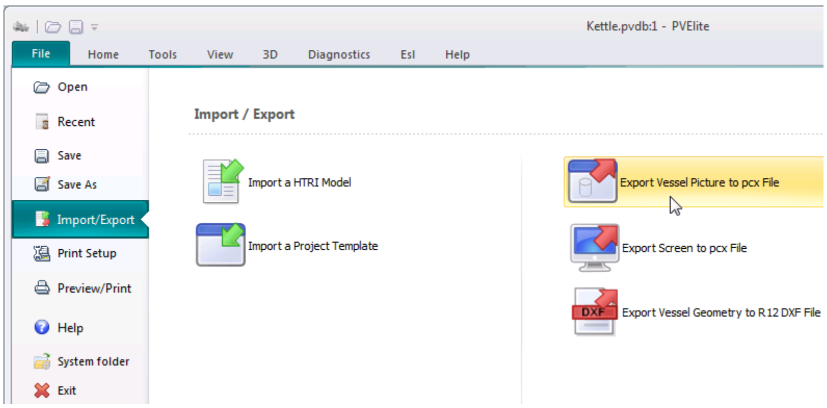


These are some reports available from PV Elite. Depending on the type, position, geometry and configuration settings the list of reports will vary.

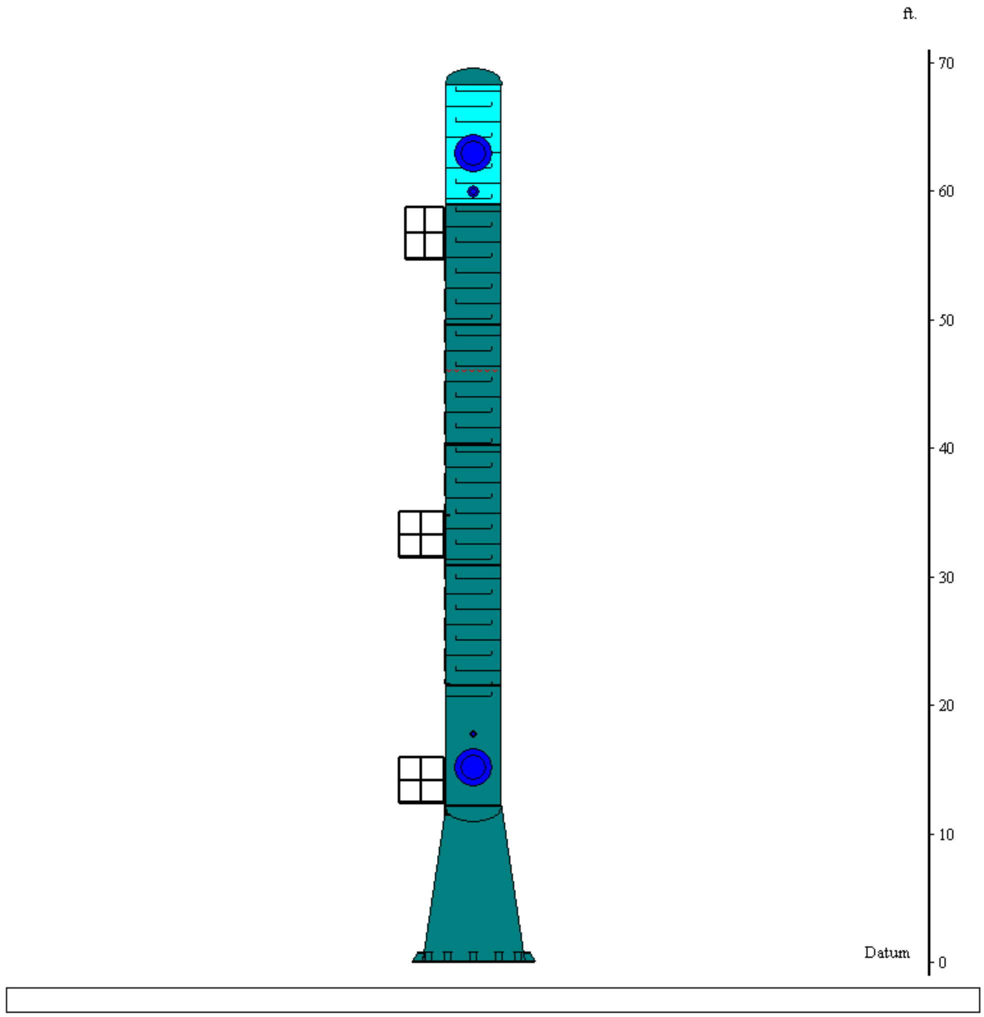
Step 0	Vessel Element Error Checking
Cover	Cover Sheet
Title	Title Page
Step 1	Vessel Input Echo
Step 2	XY Coordinate Calculations
Step 3	Internal Pressure Calculations
Step 4	External Pressure Calculations
Step 5	Weight of Elements & Details
Step 6	ANSI Flange MAWP
Step 7	Natural Frequency Calculations
Step 8	Forces & Moments Applied to Vessel
Step 9	Wind Load Calculation
Step 10	Earthquake Load Calculation
Step 11	Wind and Earthquake Shear, Bending
Step 12	Wind Deflection
Step 13	Longitudinal Stress Constants
Step 14	Longitudinal Allowable Stresses
Step 15	Longitudinal Stresses Due to Load Components
Step 16	Stress Due to Combined Loads
Step 17	Basering Calculations
Step 18	Center of Gravity Calculation
Cone 1-N	Conical Sections
Nozl 1-N	Nozzle Calculations
Step 20	Nozzle Schedule
Step 21	Nozzle Summary
Step 22	Vessel Design Summary

Recording the Model - Plotting the Vessel Image

At any point during the input process, a standard PCX file with the vessel image is available. This file can then be incorporated into reports or printed directly (on all printers) through most Microsoft Windows™ packages with graphics capabilities (such as Microsoft Word™ or Paintbrush™). The 3D graphic can also be plotted. Click on the background of the 3D View tab before pressing the print button.



The vessel graphic may also be sent directly to the printer using **File > Preview/Print** .



DXF File Generation Option

PV Elite can write out Data Interchange Files (3 all together) using **File > Import/Export > Export Vessel Geometry to R12 DXF File**. This DXF file is a text file that contains commands for generating a 2D CAD drawing of the vessel that is on a one to one scale. The border and text are scaled by the diameter conversion constant and the scale factor generated by the program or defined by you. Many popular drawing programs such as AutoCad® and MicroStation® can read and process these files. The DXF files produced by **PV Elite** are release 12 compatible. Any version of AutoCad including release 12 and after should be able to read the DXF file.

Three files are produced: the vessel drawing, the nozzle schedule, and the Bill of Material. The files are written in the folder where the input file for the vessel file is located. These files are written at the end of the program's calculation execution. Nearly every individual has his/her own way of drafting. A conscious effort was made not to be too specific. This approach allows the drafter to take the vessel drawing file and edit it as necessary.

SECTION 3

File Tab

Controls general operations of PV Elite files.

	Open - Opens an existing .pvdb or .pvi file. For more information, see *Open* (on page 43).
	Recent - Displays recently-opened files and folder. Select a file to open.
	Save - Saves the open file. For more information, see *Save* (on page 44).
	Save As - Saves the open file with a new name. For more information, see *Save As* (on page 44).
	Import/Export - Exports the open .pvdb or .pvi file to a .pcx or .dxf file. For more information, see *Export* (see "*Import/Export*" on page 44).
	Print Setup - Selects a printer and printer options. For more information, see *Print Setup* (on page 44).
	Preview/Print - Prints the graphics in the open file. For more information, see *Print* (see "*Preview/Print*" on page 45).
	Help - Displays help, getting started, contact information, and version number for PV Elite.
	System Folder - Opens the System folder in Windows Explorer.
	Exit - Closes the software.

Open

File tab: **Open**

Activates the **Open** dialog box from which you can open a previously saved PV Elite file for editing. You can also open one of the many example files delivered with PV Elite.

Open is also available on the quick access toolbar in the top-left corner of the PV Elite window.

Save

File tab: **Save**

Saves the PV Elite file that you have open. When you save a file for the first time, the **Save As** dialog box appears so that you can name the file and select a folder location.

Save is also available on the quick access toolbar in the top-left corner of the PV Elite window.

NOTE The software automatically saves .pvi files into .pvdb format.

Save As

File tab: **Save As**

Saves the open file with a different name or to a different folder location.

NOTE The software automatically saves .pvi files into .pvdb format.

Import/Export

File tab: **Import/Export**

Select one of the following:

- **Import a HTRI Model** - This command is not available at this time.
- **Import a Project Template** - Imports a PV Elite project template into the current file.
- **Export Vessel Picture to PCX File** - Sends the vessel graphics to a .pcx file. This file can be printed at a later date or added to other documents. The .pcx file is created in the same folder and with the same name as the .pvdb or .pvi file.
- **Export Screen to PCX File** - Sends a snapshot of the entire screen to a .pcx file. The .pcx file is created in the same folder and with the same name as the .pvdb or .pvi file.
- **Export Vessel Geometry to R12 DXF File** - Exports the vessel to a .dxf file. For more information, see *Export to DXF File* (see "*Setting Up the Required Parameters*" on page 194).

Print Setup

File tab: **Print Setup**

Selects a printer and defines printer options in the **Print Setup** dialog box.

Preview/Print

File tab: **Preview/Print**

Prints the model to a printer or to PDF file. Options are available for print range, number of copies, and other printing characteristics.

NOTE You must have access to a printer, either locally or over your network, before you can use this command.

Exit

File tab: **Exit**

Closes the open file and exits the software. If you have changed data since the file was last saved, or if you have not saved a new file, the **Save As** dialog box appears and the software prompts you to save your changes.

SECTION 4

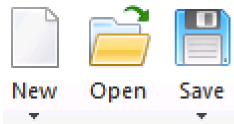
Home Tab

The **Home** tab contains the most common commands that you use in PV Elite.

Panel	**Description**
File	Provides file management commands, such as **Open**, **Save**, and **Print**. For more information, see *File Panel* (on page 48).
Elements	Provides commands for elements used to create a vessel, such as cylinders, spheres, and cones. For more information, see *Elements Panel* (on page 48).
Details	Provides commands to add details to a vessel element, such as stiffeners, nozzles, forces, moments, lining, half-pipe jackets, and tubesheets. For more information, see *Details Panel* (on page 49).
Input/Output	Provides commands to add load and restraint information. For more information, see *Input/Output Panel* (on page 189).
Utilities	Provides miscellaneous element commands, for functions such as insert, delete, update, share, and flip. For more information, see *Utilities Panel* (on page 191).
Auxiliary	Provides miscellaneous model commands for functions such as manipulating of the model, creating drawings, and viewing properties. For more information, see *Auxiliary Panel* (on page 192).
Analyze	Provides commands for analyzing the model. For more information, see *Analyze Panel* (on page 195).
Units/Code	Provides commands for changing units and the design code. For more information, see *Units/Code Panel* (on page 197).

File Panel

The following commands are available on the **File** panel on the **Home** tab.



	New - Creates a new .pvdb file. For more information, see **New** (on page 48).
	Open - Opens an existing .pvdb or .pvi file. The **Open** command is also available on the Quick Access bar at the top. For more information, see **Open** (on page 43).
	Save - Saves the open file. The **Save** command is also available on the Quick Access bar at the top. For more information, see **Save** (on page 44).

New

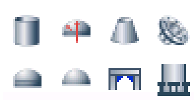
Home tab: **File > New**

Creates a new PV Elite input file (.pvdb). You can also press **Ctrl+N** on the keyboard to create a new input file.

You can select a specification (**ASME Section VIII-Division 1**, **ASME Section VIII-Division 2**, **British Standard PD:5500**, or **EN-13445**) or just create an empty .pvdb file.

Elements Panel

Provides commands for elements used to create a vessel, such as cylinders, spheres, and cones. The **Elements** panel is available on the **Home** tab.



	Cylinder - Cylindrical shell
	Ellipse - Elliptical head
	Torisphere - Torispherical head
	Sphere - Spherical head or shell

	Cone - Conical head or shell segment
	Welded Flat Head - Places a welded flat head.
	ANSI/Bolted Flange - Bolted body flange
	Skirt - Skirt support with base ring

The software does not require the complete construction of a vessel for analysis. Individual elements or groups of elements may be defined and partially analyzed. Only complete vessels with proper supports can be analyzed for dead weight and live loads.

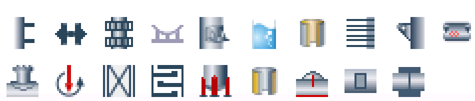
All elements, except **Skirt**, can be used to create either horizontal or vertical vessels.

- Models for vertical vessels are built from bottom to top.
- Models for horizontal vessels are built from left to right.

The vessel orientation is established with the first element. If starting with a skirt, it is a vertical vessel. If starting with a head, the head can be flipped between a bottom head (vertical vessel) and a left head (horizontal vessel) by clicking **Flip Orientation** on the **Utilities** panel. After the second element is added to the model, the orientation is fixed. Skirts are the only vessel supports that are modeled as elements. Other supports, such as legs and lugs for vertical vessels and saddles, are modeled as details on the elements. For more information, see *Details Panel* (on page 49).

Details Panel

Details can only be specified on the currently selected element (see *Elements Panel* (on page 48) for more information on placing elements). Details, such as nozzles, insulation, operating fluid, platforms and the like, define the vessel's weight information for load calculations. These commands are available on the **Details** panel of the **Home** tab.



	Stiffening Ring - Add stiffening rings on the selected cylinder element. For more information, see *Stiffening Ring* (on page 54).
	Nozzle - Add nozzles on the selected cylinder or head element. For more information, see *Nozzle* (on page 58).
	Weight - Add piping and miscellaneous weight added to the selected cylinder or head element. For more information, see *Weight* (on page 88).
	Force and Moment - Add external forces and moments to the selected cylinder or head element. For more information, see *Force and Moment* (on page 91).

	Platform - Add a platform to the selected element. For more information, see *Platform* (on page 92).
	Packing - Add packing to the selected element. For more information, see *Packing* (on page 95).
	Saddle - Add saddles to the selected horizontal cylinder element. For more information, see *Saddle* (on page 97).
	Tray - Add a set of equally spaced trays to the selected vertical cylinder element. For more information, see *Tray* (on page 105).
	Lug - Add support lugs to the selected vertical cylinder element. For more information, see *Lug* (on page 105).
	Legs - Add support legs to the selected vertical cylinder element. For more information, see *Legs* (on page 108).
	Liquid - Add liquid data to the selected element. For more information, see *Liquid* (on page 115).
	Insulation - Add insulation to the selected element. For more information, see *Insulation* (on page 118).
	Lining - Add lining to the selected element. For more information, see *Lining* (on page 119).
	Halfpipe Jacket - Add half pipe jackets to the selected cylinder element. For more information, see *Halfpipe Jacket* (on page 120).
	Tubesheet - Add a tubesheet to the selected element. For more information, see *Tubesheet* (on page 124).
	Generic Clip - Add a clip to the selected element. For more information, see *Generic Clip* (on page 166).
	Lifting Lug Data - Add lugs to the selected element. For more information, see *Lifting Lug Data* (on page 169).
	ASME Appendix 9 Jacket - Add a jacket to the selected element. For more information, see *ASME Appendix 9 Jacket* (on page 173).
	API-579 Flaw/Damage Input/Analysis - Add a flaw to the selected element. For more information, see *API-579 Flaw/Damage Input/Analysis* (on page 182).

Add a Detail

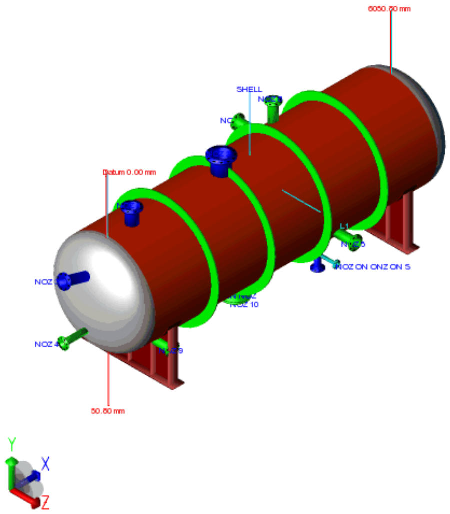
Details are assigned to elements using the commands on the **Details** panel on the **Home** tab.

1. Select an element in the graphics view.
2. Click the command on the **Details** panel on the **Home** tab.

   The dialog box for the command displays.

3. Enter the needed information, and click **OK**.

   The detail is added to the element and displays in the graphics view.



Modify a Detail

To modify one or more existing details:

1. In the graphic view, click the element having the existing detail that you want to modify.
2. Click the corresponding command on the **Details** panel.

   The dialog box for the command displays.

3. If there is only one detail, make the needed changes, and click **OK**.
4. If there is more than one detail, click **Previous** or **Go To Next** to modify the needed detail.

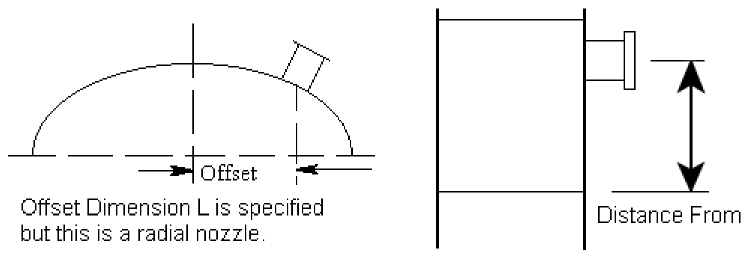
NOTE In the graphic view, you can also right-click a detail on the selected element to open the dialog box for that detail.

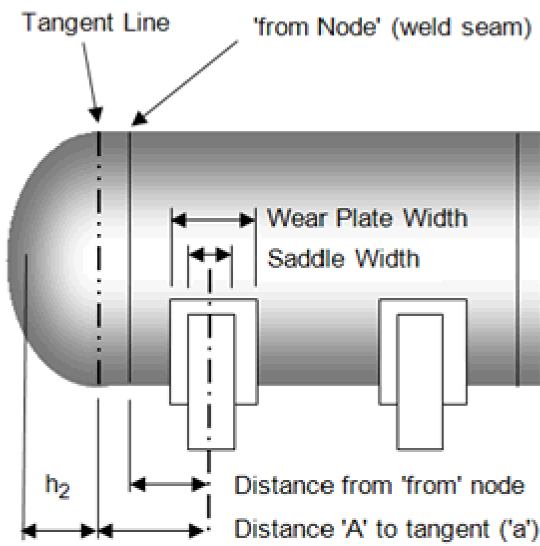
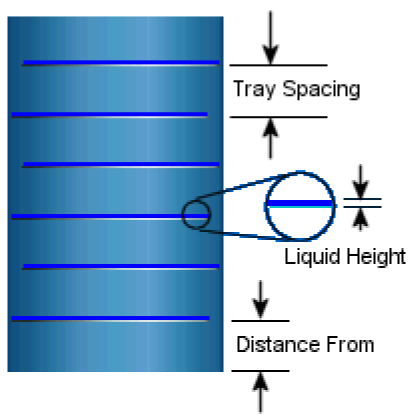
Common Detail Parameters

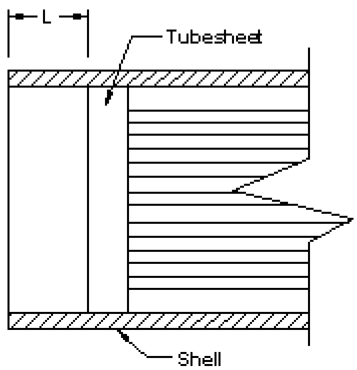
The following fields appear in the dialog box for most details.

From Node - Displays the **From Node** for the selected element. The **From Node** is the software-generated node number describing the starting location of the element. The value cannot be modified.

Distance from "From" Node - Enter the axial or longitudinal distance from the From Node to the detail at the following locations:

Detail	**Location**
Stiffening Ring	Centerline of the first ring.
Nozzle	Centerline of the nozzle. If the nozzle is located in a head, then enter the distance from the head centerline. This dimension should always be entered if it is not zero. 
Weight	Point at which the weight acts.
Force and Moment	Point at which the force or moment acts.
Platform	Bottom of the platform.
Packing	Start of the packed section.

Detail	Location
Saddle	Vertical centerline of the saddle. 
Tray	Bottom of the lowest tray. 
Lug	Centroid of the lug attachment weld.
Legs	Centroid of the leg attachment weld. NOTE The software uses the value of **Overall Length of Legs** when performing AISC unity checks.
Liquid	Start of the liquid section.
Insulation	Start of the insulated section.
Lining	Start of the lined section.

Detail	Location
Halfpipe Jacket	The distance that the jacket starts from the circumferential seam of this element. This value, in conjunction with **Length Along Shell of Jacket Section**, is used to determine the weight of the jacket.
Tubesheet	The distance from the shell to the outer face of the nearer tubesheet. 
Generic Clip	Center of the clip.
Lifting Lug	For a vertical vessel, the centroid of the weld group. For example, the base of the lug + 1/2 the weld distance (*wl*) on the side of the lug. If the vessel is horizontal, enter the distance from the left end or tangent of the vessel to the center of each of the two lugs.
Appendix 9 Jacket	Bottom of the jacket.

Detail Description - Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

Stiffening Ring

Home tab: **Details** > **Stiffening Ring**

Adds one or more stiffening rings to the selected cylinder element.

As stiffening ring data is entered, the software automatically calculates the required inertias, except when the ring is at a cone-to-cylinder junction. For bar rings, the software sizes a new ring based on a default thickness of 0.375 inches or the element value entered for **Bar thickness to use when designing new rings** on the **Equipment Installation and Miscellaneous Options Dialog Box** of the **Load Cases** tab.

NOTE The maximum length between stiffeners is shown on the *Status Bar* (on page 20) as **Slen**. When **>>>** displays, the element does not require a reinforcing ring for external pressure purposes.

Previous Ring - If you created more than one ring on the element, click to go back to the previous ring.

Go To Next Ring - If you created more than one ring on the element, click to go to the next ring.

Add New Ring - Click to add a new ring to the shell or head element.

Delete - Deletes all data for the current ring.

Common Detail Parameters (on page 52)

Ring Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Ring Location - Select the location of the ring. Select **ID** if the ring is on the inside diameter of the cylinder. Select **OD** if the ring is on the outside diameter of the cylinder. **OD** is the most common selection.

Ring Type - Select **Bar Type** for a flat bar stiffener ring. Select **Section Type** for a more complex structural cross-section.

For **Bar Type**, click **Check "Standard" Bars**. The **Bar Selection** dialog box displays, where you select the flat bar size, and enter values for **Axial Thickness** and **Radial Width**. Bar type stiffeners have a simple rectangular cross section and the software calculates the ring properties based on the width and thickness of the ring. Most stiffeners are bar type.

For **Section Type**, select a standard cross-section in **Choose a Section** or click **Section Calculator** to create a custom fabricated section. You need the following properties for the section: **Moment of Inertia, Cross Sectional Area, and Distance to Ring Centroid.** The software provides these values for standard cross-sections. You must enter the values for a custom section, as defined in the AISC Steel Construction Manual.

Inside Diameter - Enter the inside diameter of the stiffening ring. This value is usually equal to the outside diameter of the vessel shell (when **OD** is selected for **Ring Location**), except for the less common case of a stiffening ring on the inside of the vessel (when **ID** is selected for **Ring Location**).This value is used both to calculate the weight of the ring and the stiffness of the ring for external pressure purposes. This entry is only available when **Bar Type** is selected for **Ring Type**.

Thickness - Enter the axial thickness of the stiffening ring. This value is used to calculate the weight of the ring and the stiffness of the ring for external pressure purposes. This entry is only available when **Bar Type** is selected for **Ring Type**.

Outside Diameter - Enter the outside diameter of the stiffening ring. This value is usually greater than the outside diameter of the vessel shell (when **OD** is selected for **Ring Location**), except for the less common case of a stiffening ring on the inside of the vessel (when **ID** is selected for **Ring Location**).This value is used to calculate both the weight of the ring and the stiffness of the ring for external pressure purposes. This entry is only available when **Bar Type** is selected for **Ring Type**.

Structural Database - Select the structural specification database to use for cross-sections. This entry is only available when **Section Type** is selected for **Ring Type**.

Moment of Inertia - Displays the moment of inertia of the stiffening ring about its neutral axis from the specification selected for **Structural Database,** when a standard section is selected for

Choose a Section. If you created a custom section using **Section Calculator**, enter a value according to the selected specification. The software uses this value to determine the adequacy of the ring for external pressure calculations, and for conical calculations according to Appendix 1 of the ASME Code. This entry is only available when **Section Type** is selected for **Ring Type**.

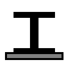
Cross Sectional Area - Displays the cross sectional area of the stiffening ring from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a value according to the selected specification. This entry is only available when **Section Type** is selected for **Ring Type**.

Distance to Ring Centroid - Displays the distance from the surface of the shell to the ring centroid from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a value according to the selected specification. This entry is only available when **Section Type** is selected for **Ring Type**.

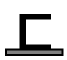
Section Ring Height - Displays the depth of the cross-section from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a value according to the selected specification. This entry is only available when **Section Type** is selected for **Ring Type**.

Section Name - Displays the section name from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a name. This entry is only available when **Section Type** is selected for **Ring Type**.

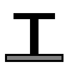
Choose a Section - Select a section type and a section size:



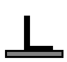
I - Beam
W Section



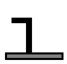
Channel



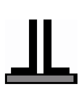
WT Section
MT Section
ST Section
(strong axis out)



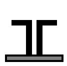
Angle
(weak axis out)



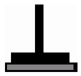
Angle
(strong axis out)



Double Angle
(weak axis out)



Double Angle
(strong axis out)



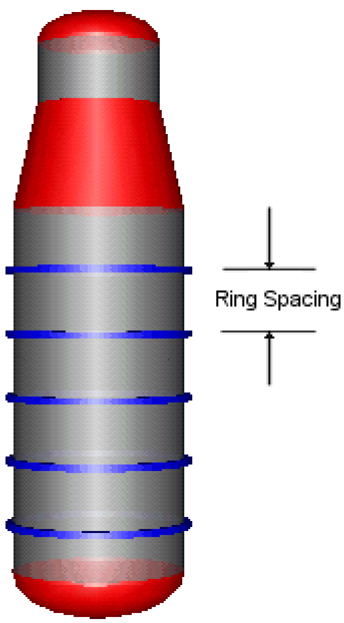
WT Section
MT Section
ST Section
(weak axis out)

Ring Fillet Weld Leg Size - Enter the size of the leg of the ring fillet weld.

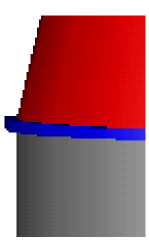
Ring Attachment Style - Select the type of fillet weld. Select **Intermittent**, **Continuous**, or **Both**.

Number of Rings to Add - Enter the number of rings to place on the cylinder element. The software resets this number to 0 after you close the **Stiffening Ring** dialog box, so it is possible to add multiple groups of rings.

Ring Spacing - Enter the ring spacing. The first ring is placed at the distance entered for **Distance from "From" Node**. The rest of the rings are placed after the first ring by the **Ring Spacing** increment. If you are adding one ring, a value is not needed. The software resets this number to 0 after you close the **Stiffening Ring** dialog box, so it is possible to add multiple groups of rings.



Cone to Shell Junction Ring? - Select to attach the ring at the junction of the cone and cylinder elements. In this case, the software does not design the ring for external pressure considerations, but considers it for junction reinforcement according to App. 1-5 and 1-8 (Div. 1) or AD 360.3 (Div. 2).



Nozzle

Home tab: **Details** > **Nozzle**

Adds a nozzle to the selected cylinder or head element. Nozzles add to the total dead weight of the vessel and are used to evaluate the maximum allowable working pressure (MAWP) of the vessel and flange.

Previous Nozzle - If you created more than one nozzle on the element, click to go back to the previous nozzle.

Go To Next Nozzle - If you created more than one nozzle on the element, click to go to the next nozzle.

Add New Nozzle - Click to add a new nozzle to the shell or head element.

Delete - Deletes all data for the current nozzle.

Plot - Click to open the **Nozzle Graphics** dialog box. A cross-section view of the nozzle design is shown.

Topics

Nozzle Main Tab (Nozzle Input/Analysis Dialog Box)

Defines physical and weld parameters for the nozzle.

Common Detail Parameters (on page 52)

Previous Nozzle - If you created more than one nozzle on the element, click to go back to the previous nozzle.

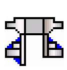
Go To Next Nozzle - If you created more than one nozzle on the element, click to go to the next nozzle.

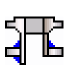
Add New Nozzle - Click to add a new nozzle to the shell or head element.

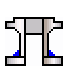
Delete - Deletes all data for the current nozzle.

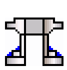
Plot - Click to open the **Nozzle Graphics** dialog box. A cross-section view of the nozzle design is shown.

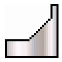
Nozzle Attachment - Select the type of nozzle:

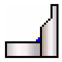
 Inserted nozzle with reinforcing pad

 Inserted nozzle without reinforcing pad

 Abutting nozzle with reinforcing pad

 Abutting nozzle without reinforcing pad

 Heavy Barrel Type, HB

 Type “F” connection

FVC Catalog - Select **FVC Catalogue** to select a nozzle from the Forged Vessel Connections catalog. For more information about FVC, see *http://www.forgedvesselconn.com/* (*http://www.forgedvesselconn.com/*).

- **Nominal Diameter** - Select the nominal diameter that you need.
- **Flange Class** - Select the pressure rating that you need.
- **Connection Type** - Select the connection type that you need.
- **Overall Length Oal** - Enter the length that you need from the surface of the vessel to the flange face.
- **Nozzle does not have a "Nut Relief"** - Select if the nozzle does not have a nut relief.
- **Select Now** - Loads the FVC nozzle data from the catalog into the main **Nozzle Input** dialog box.

Coupling Lookup - Select **Coupling Lookup** to find coupling properties for the nozzle. Select the pressure rating and the diameter, and then click **Select Now**.

Just like - Select to place another nozzle just like an existing nozzle.

Nozzle Description - Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

NOTE When using ASME VIII - 1, a special directive is available for small nozzles. If the text directive **#SN** is placed anywhere in the description, the software calculates the areas and MAWP of this nozzle connection. This directive overrides the global directive set in **Tools** > **Configuration**. It may be necessary to use this directive when required by UG-36. This paragraph in the code defines a small nozzle.

Nozzle Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Schedule - Specify the thickness of the nozzle by selecting the schedule of the nozzle neck pipe. Acceptable schedules are:

SCH 5S	SCH 30
SCH 10	SCH 40
SCH 10S	SCH 40S
SCH 20	SCH 60
SCH 30	SCH 80
SCH 40	SCH 80S
SCH 40S	SCH 100
SCH 60	SCH 120
SCH 80	SCH 140
SCH 5S	SCH 160
SCH 10	SCH STD
SCH 10S	SCH X-STG
SCH 20	SCH XX-STG

NOTES

- DIN schedules are also available.
- All schedules of pipe might not have a corresponding diameter associated. In this case, the software displays an error stating the thickness of the nozzle was not found.

Diameter - Enter the actual diameter of the nozzle when **Actual** is selected for **Thickness Basis**. Enter the nominal diameter, or click ... to select a diameter when **Nominal** or **Minimum** is selected for **Thickness Basis**. Values are given in English units and must be multiplied by the diameter conversion constant so that the software arrives at the correct diameter when it calculates the English value of the diameter. Valid English-unit nominal diameters range from 0.125 to 30.0 inches.

Dia. Basis - Select the type of diameter to use for the element. Select **ID** for the inside diameter. Select **OD** for the outside diameter. **ID** and **OD** are available for ASME Division 1. Only **OD** is available for ASME VIII-1.

NOTES

- The ASME code provides different equations for required thickness based on whether the geometry is specified on **ID** or **OD**. By using the **ID** basis, the software computes a thinner required thickness, Tr, for the nozzle, such as in high-pressure, thick-wall geometries.
- If you are modeling a cylinder with welded flat heads on either end, and the welded flat heads sit just inside the cylinder shell, set **Diameter Basis** to **ID** and specify the **Inside Diameter** value on the welded flat heads to be the same size as the **Inside Diameter** of the cylinder. Once you make these changes, if the flat head element still displays as sitting on the cylinder shell (instead of inside of the shell), select **Flip Orientation** twice. The software refreshes the model display to show the welded flat head inside the cylinder shell.

Thickness Basis - Select the thickness basis:

- **Actual** - The software uses the actual values entered for **Diameter and Actual Thk**. Select this value if the nozzle is fabricated from plate.
- **Nominal** - The software uses **Diameter** as the nominal diameter and looks up the actual diameter. The software also looks up the nominal thickness based on the selection for **Schedule**.
- **Minimum** - The software uses **Diameter** as the nominal diameter and looks up the actual diameter. The software also looks up the nominal thickness based on the selection for **Schedule**. It then multiplies the nominal thickness by a factor of 0.875.

Total CA. - Enter the corrosion allowance. The software adjusts both the actual thickness and the inside diameter for the corrosion allowance. For ASME VIII-1, if the nozzle has an external corrosion allowance, add the external corrosion allowance to the internal corrosion allowance and enter the total value.

Actual Thk - If you selected **Actual** for **Thickness Basis**, enter the minimum actual thickness of the nozzle wall.

Nozzle Orientation (Nozzle Main Tab) (on page 61)

Pad or Hub Properties (Nozzle Main Tab) (on page 70)

Additional Weld Data (Nozzle Main Tab) (on page 72)

Miscellaneous (Nozzle Main Tab) (on page 73)

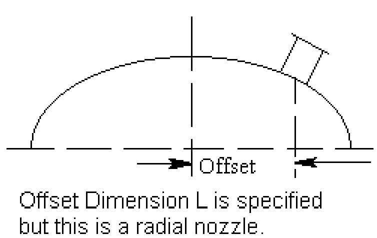
Nozzle Orientation (Nozzle Main Tab)

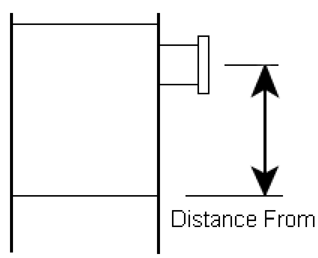
Defines orientation parameters for the nozzle.

Is this nozzle connected to another nozzle? - Select this option if the nozzle is connected to another nozzle.

Parent Nozzle - Select the parent nozzle for this nozzle.

Offset Dimension L - Enter the axial distance from the **From Node** to the centerline of the nozzle. If the nozzle is located in a head, then enter the distance from the head centerline. This dimension should always be entered if it is not zero. This option is similar to **Distance from "From" Node** and is available when the nozzle is on a head element.





Distance from Shell Surface - Enter the axial distance from the **From Node** to the centerline of the nozzle. If the nozzle is located in a head, then enter the distance from the head centerline. This dimension should always be entered if it is not zero. This option is similar to **Distance from "From" Node** and is available when **Is this nozzle connected to another nozzle?** is selected.

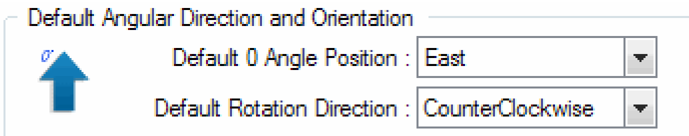
Layout - Click to open the **Nozzle Layout and Placement Dialog Box (on page 67)** and set the nozzle orientation independent of the coordinate system of the model.

Layout Angle

Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the weight at the point where it is attached to the shell.

For a horizontal vessel, the zero degree line is at 12:00 (looking at a clock); 90 degrees is at 3:00. Entering these layout angles is important if the horizontal vessel has a liquid level and the nozzles are being designed using **Design P + Static Head**, selected for **Nozzle/Clip Design Pressure Options** on the **Load Cases** tab. For a vertical vessel, the angle is more arbitrary. For purposes of rendering the graphics, the assumption is that the zero degree line is at 3:00 and 90 degrees is 12:00.

The position of the zero degree reference direction (North, West or East) and the angular rotation (clockwise or counterclockwise) are set in **Tools** tab, **Set Configuration Parameters**, *Set Default Values Tab (Configuration Dialog)* (see "*Set Default Values Tab (Configuration Tab)*" on page 207).



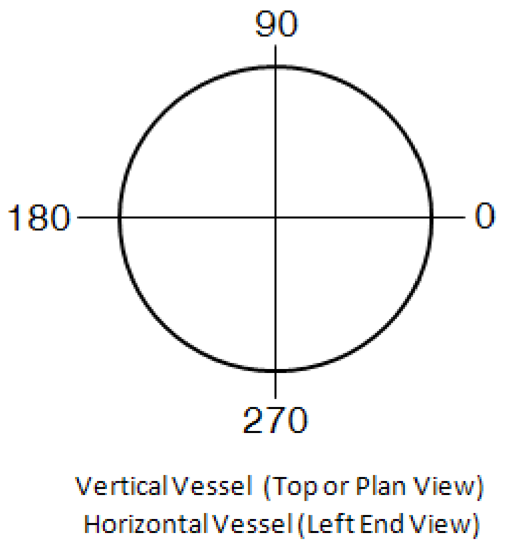
If the new orientation is different from the orientation of the current model, the software asks to update the model with the new settings. If the settings are updated, the weights and areas of platforms must be recalculated:

- Click **List Dialog** (on page 193). On the **Detail Listing** dialog box, click the **Platform** tab so that platform data is visible. Click **Platform Wind Area** and **Platform Weight** to recalculate.
- Click **Platform Input** for each platform. Tab through the fields in the **Platform** dialog box, causing the weight and area to be recalculated

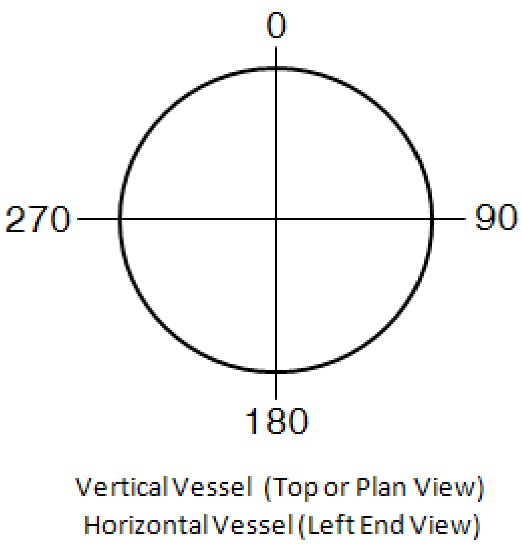
NOTE The angular settings apply to nozzles, clips, legs, lifting lugs, support lugs, base rings, platforms, weights and half-pipe jackets.

Examples

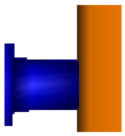
Default Orientation



Angle Position: North
Rotation Direction: Clockwise



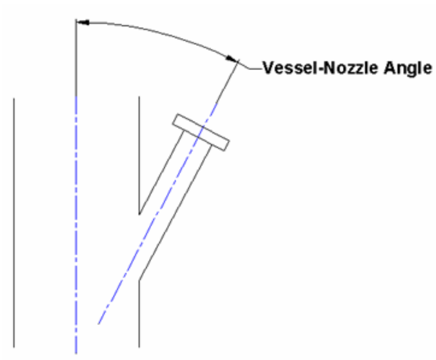
Radial Nozzle - Select to specify a radial nozzle.

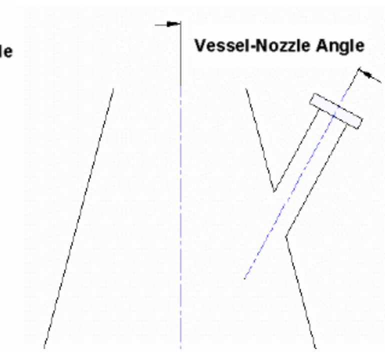


Angle or Lateral Nozzle - Select to specify an angled or lateral nozzle. Also enter a value for **Centerline Tilt Angle**, and — if needed — for **Cyl./Cone Offset Dimension L**.

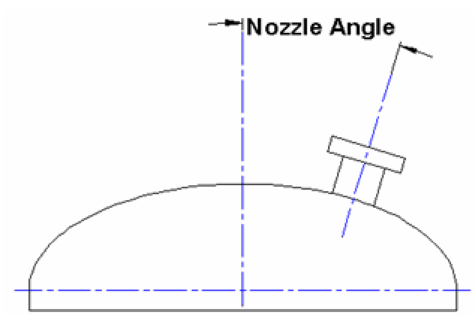
Centerline Tilt Angle

Enter the angle for a non-radial nozzle when **Angle or Lateral Nozzle** is selected.





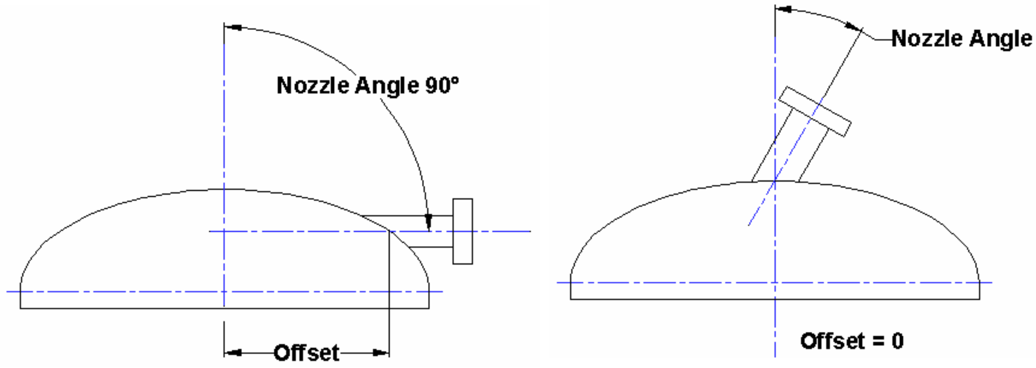
Non-radial nozzles can be specified by entering the angle between the vessel and nozzle centerlines and the offset from vessel centerline. This vessel-nozzle centerline angle can vary from 0 to a limiting value depending upon specific geometry. For nozzles on top heads, this value will generally range between 0 and 90 degrees. On bottom heads, this value would be between 90 and 180 degrees:



NOTE The input specification for non-radial and non-hillside nozzles changed starting with version 2008. The angle is measured between the centerline of the nozzle and the centerline of the vessel. This value can be determined from an electronic drawing of the model.

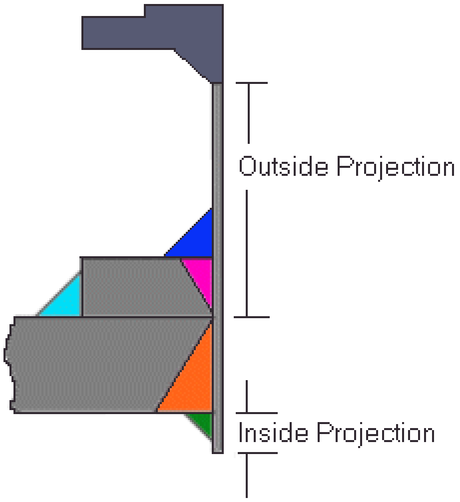
Cyl./Cone Offset Dimension L

Enter an offset dimension from the vessel centerline for hillside nozzles (neither **Radial Nozzle** or **Centerline Tilt Angle** are selected) and angled nozzles (**Centerline Tilt Angle** is selected).



Hillside nozzles and some tilted nozzles are subject to calculations to meet ASME area requirements in both planes of reinforcement. In these cases, the software automatically checks area requirements in both planes using the corresponding lengths of the nozzle opening. For integral construction, the Code F correction factor of 0.5 is automatically applied in the hillside direction. If the connection is pad reinforced, a value of 1.0 is used. The F factor is used to account for the fact that the longitudinal stress is one-half of the hoop stress. The use of the F factor is limited to nozzles located on cylindrical and conical sections under internal pressure.

Projection Outside - Enter the nozzle projection distance outward from the surface of the vessel. The distance is usually to the attached flange or cover. This length is used for weight calculations and for external pressure calculations.

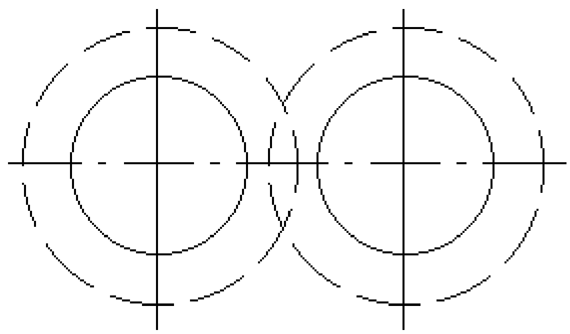


Projection Inside - Enter the nozzle projection distance into the vessel. The software uses the least of **Projection Inside** and **Limits (Thickness)** with no pad to calculate the area available in the inward nozzle. Therefore, you may safely enter a large number such as six or twelve inches if the nozzle continues into the vessel a long distance.

For some nozzle types such as those shown in ASME VIII-1 UW-16.1 sketch K, the inside projection is the distance from the inside surface of the vessel to the inside edge of the nozzle. This distance must be less than the shell thickness.

Limits (Diameter) - Enter the maximum diameter for material contributing to nozzle reinforcement. For example:

- Where two nozzles are close together and the reinforcements overlap.



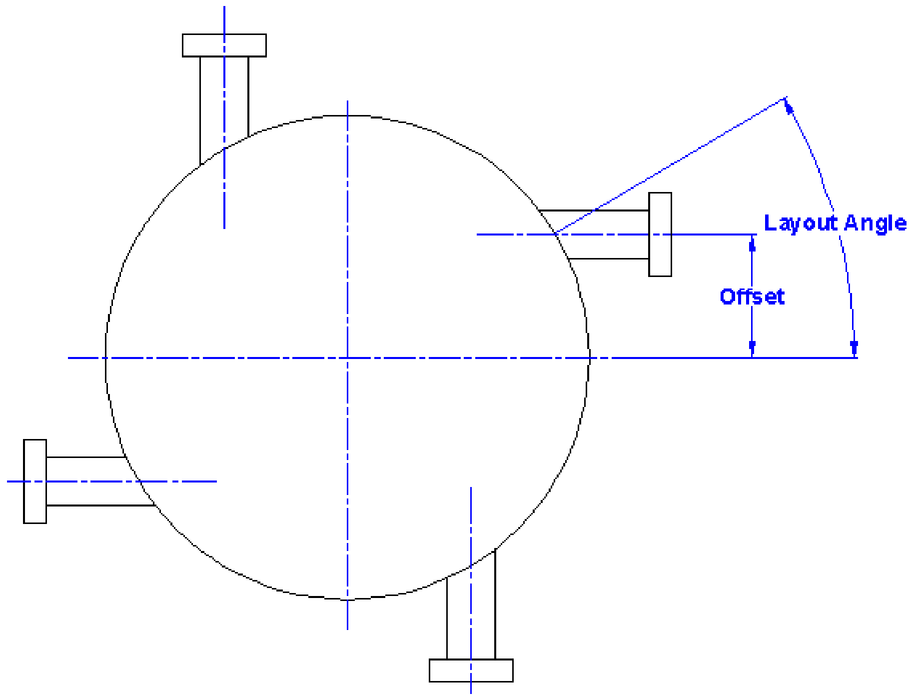
- A vessel seam for which you do not want to take an available area reduction.

Limits (Thickness) - Enter the maximum thickness for material contributing to nozzle reinforcement. For example, where a studding pad or nozzle stub do not extend normal to the vessel wall as far as the thickness limit of the nozzle calculation.

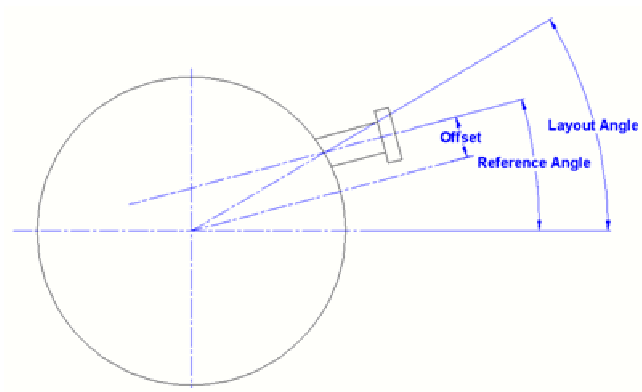
Overriding Weight - Enter a weight if the nozzle is significantly different from a standard weight nozzle. The weight overrides the software-calculated weight that is based on other entered values and internal tables of typical weights.

Nozzle Layout and Placement Dialog Box

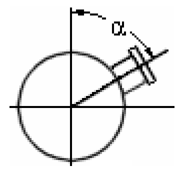
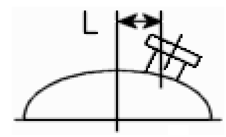
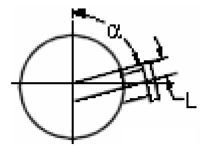
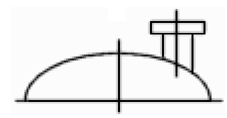
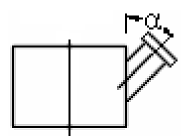
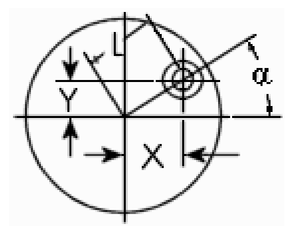
Provides an alternative and more versatile method of nozzle orientation. You are not confined to having nozzles point in the directions of the coordinate system axes of the model. For example, when using the standard parameters (such as **Offset Dimension L** and **Layout Angle**) on the **Nozzle Input/Analysis** dialog box, hillside nozzles may only point in the X, Y and Z directions:



When using the **Layout** command and the **Nozzle Layout and Placement** dialog box, a hillside nozzle can point in any direction by using **Reference Angle alpha** instead of **Layout Angle**:

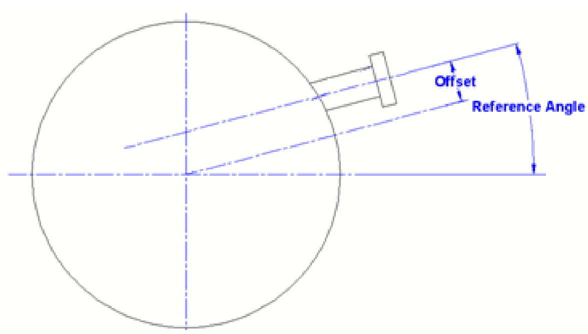


Nozzle Style - Select a nozzle orientation style:

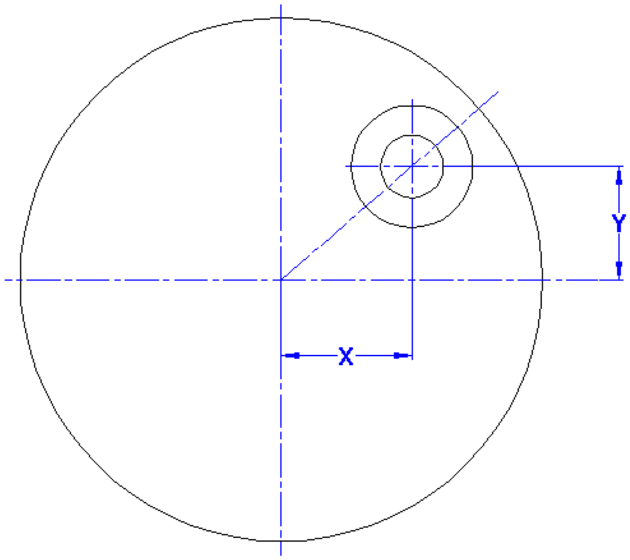
Nozzle Style	Cylinder Element	Head Element
Radial Nozzle		
Hillside Nozzle		
Lateral Nozzle		

Reference Angle alpha - Enter the angle of the line to use as reference for **Nozzle Offset Dimension L**.

Nozzle Offset Dimension L - Enter the distance from the reference line to the centerline of the nozzle. This option is not available for a radial or lateral nozzle.

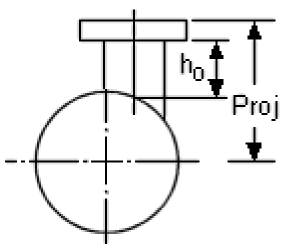


Nozzle "X" Dimension and **Nozzle "Y" Dimension -** For nozzles on heads, enter values for **Nozzle "X" Dimension** and **Nozzle "Y" Dimension** to calculate **Reference Angle alpha** and **Nozzle Offset Dimension L** from the X-Y coordinate location. Nozzle locations on heads are often given in the X-Y coordinate system.

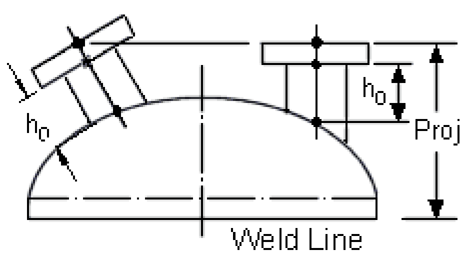


Projection Dimension "Proj" - For a radial or hillside nozzle, enter the projection from the centerline of the cylinder to the end of the nozzle. The software calculates the projection *ho*.

For a nozzle on a cylinder:

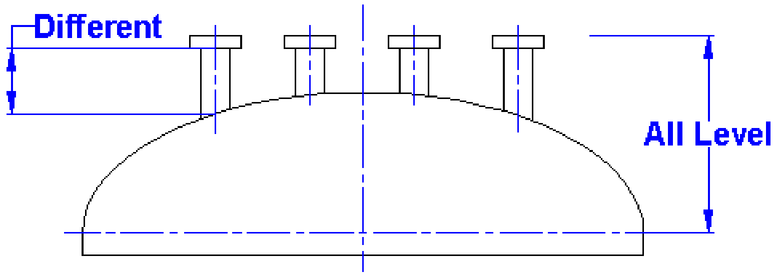


For a nozzle on a head:



Hillside Nozzles

Hillside nozzles in heads are often arranged with flange faces level in the same plane. Due to the curvature of the head, each nozzle has a *different* value for outside projection *ho*:



Because the value of **Projection Dimension "Proj"** is the *same* for each nozzle, this arrangement is simple.

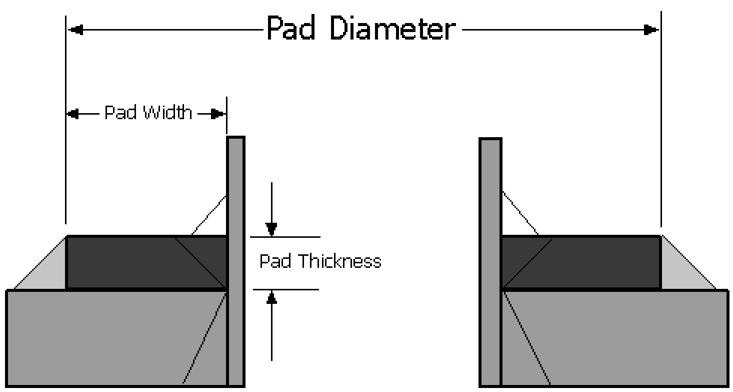
NOTE Click **OK** to save parameter values and return to the **Nozzle Input/Analysis** dialog box with new orientation values displayed.

Pad or Hub Properties (Nozzle Main Tab)

Pad Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

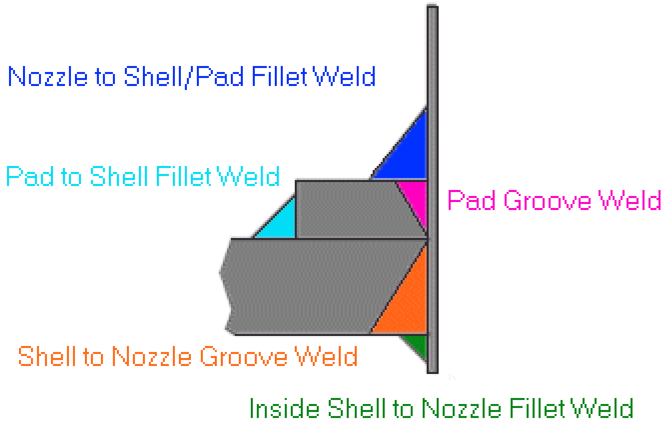
Pad Diameter / Width - Enter the diameter of the pad. The diameter of the pad is the length along the vessel shell, *not* the projected diameter around the nozzle.

Alternatively, you can enter in the width of the pad. The software then calculates the pad diameter.



Pad Thickness - Enter the thickness of the pad. Any allowances for external corrosion should be taken into account for the pad thickness.

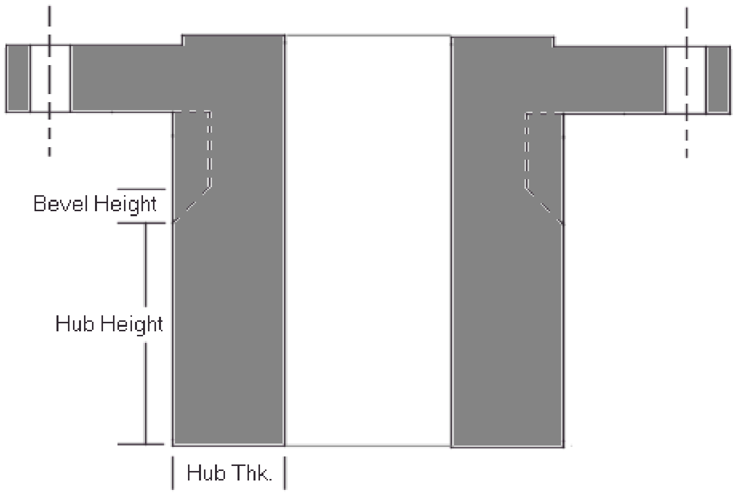
Groove Weld Depth - Enter the total depth of the groove weld between the pad and the nozzle neck. Most groove welds are full penetration welds. The depth of the weld is the same as the thickness of the pad. If the pad is attached with a partial penetration weld, enter the depth of the partial penetration. If the pad is attached with a fillet weld, enter zero.



Weld Leg at Pad OD - Enter the size of one leg of the fillet weld between the pad outside diameter and the shell. If any part of this weld falls outside the diameter limit, only the part of the weld inside the diameter limit is included in the available area.

For VIII-1 split pads, multiply A5 by 0.75 per UG-37(h) - Select to indicate that, with ASME VIII-1 split pads, the software multiplies area A5 by .75 for every UG-37 (h) used. ASME VIII-1 Nozzle F Factor specifies the nozzle F factor in paragraph UG-37 of ASME VIII-1.

Hub Thickness - Enter the thickness of the thicker part of the nozzle at the base. This value is equal to (Nozzle OD at the base - Nozzle ID at the base)/2.



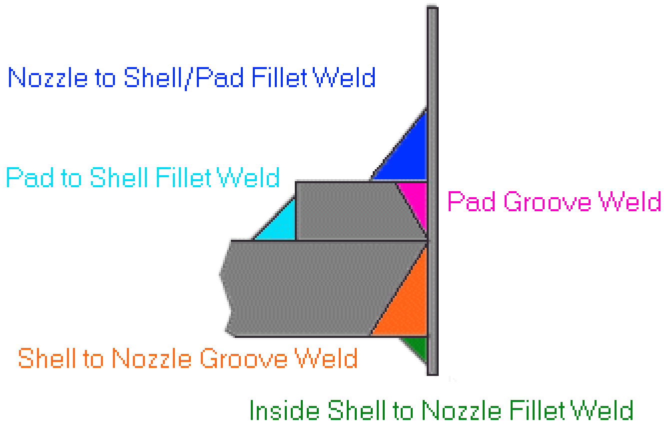
Hub Height - Enter the hub height. This value is equal to (overall length - nut relief height - flange thickness - raised face dimension).

Bevel Height - Enter the bevel height.

Obtain ASME Bevel Angle - Obtain the ASME bevel Angle.

Additional Weld Data (Nozzle Main Tab)

Nozzle to Pad/Shell Outside Fillet Weld Leg - Enter the size of one leg of the fillet weld between the nozzle and the pad or the nozzle and the outside shell.



Nozzle to Shell Inside Fillet Weld Leg - Enter the size of one leg of the fillet weld between the inward nozzle and the inside shell.

Nozzle to Shell Groove Weld Depth - Enter the total depth of the groove weld between the nozzle and the vessel. Most groove welds are full penetration welds. The depth of the weld is the same as the thickness of the pad. If the pad is attached with a partial penetration weld, enter the depth of the partial penetration. If the pad is attached with a fillet weld, enter zero.

ASME VIII-1 Weld Type - Select an ASME VIII Division 1 weld type:

A, **B**, **C**, **D**, **E**, **F-1**, **F-2**, **F-3**, **F-4**, **G**, **X-1**, **Y-1**, **Z-1** - For these weld detail sketches, according to UW-16.1, the software does not perform the weld strength calculation. In these cases, the code does not require weld strength/path calculations for full penetration groove welds for pressure loadings.

I, **J**, **K**, **L**, **X-2**, **Y-2**, **Z-2** - For these weld types, the software performs the additional weld size calculations according to UW-16(d)(1).

None - The software performs the calculation regardless of the type of weld.

NOTES

- Some sketches, such as UW-16.1 (K) or UG-40 (n), show that the nozzle does not completely extend to the inner surface of the shell. In these cases, be sure to enter the correct value for **Projection Inside**.
- This entry only available when **ASME Section VIII-Division 1** is selected for **File** > **New** (on page 48).

Miscellaneous (Nozzle Main Tab)

Flange Class - Select the pressure rating class for the ANSI B16.5 or DIN flange, based on the pressure rating class for the attached nozzle. Select **150**, **300**, **400**, **600**, **900**, **1500**, **2500**, **NP6**, **NP10**, **NP24**, **NP40**, **NP63**, **NP100**, or **None**.

Flange Grade

Select the flange material grade (group). Please note that there are certain advisories on the use of certain material grades. Please review those cautionary notes in the ANSI B16.5 code. ASME B16.5-2003 and ASME B16.5-1996 flange grades are available:

Table 1A List of Material Specifications (ASME B16.5-2003)

Material Group	**Nominal Designation**	**Forgings**	**Castings**	**Plates**
1.1	C-Si C-Mn-Si C-Mn-Si-V 3½ Ni	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1 A 350 Gr. LF3	A 216 Gr. WCB	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si C-Mn-Si-V 2½Ni 3½Ni	A 350 Gr. LF 6 Cl.2	A 316 Gr. WCC A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	A 203 Gr. B A 203 Gr. E
1.3	C-Si C-Mn-Si 2 ½Ni 3 ½Ni C-½Mo		A 352 Gr. LCB A 217 Gr. WC1 A 352 Gr. LC1	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1		A 204 Gr. A A 204 Gr. B
1.7	½C-½Mo Ni-½Cr-½Mo ¾Ni-¾Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1¼Cr-½Mo 1¼Cr-½Mo-Si	A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	2¼Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.11	Cr-½Mo			A 204 Gr. C
1.13	5Cr-½Mo	A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	

Material Group	Nominal Designation	Forgings	Castings	Plates
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
1.17	1Cr-½Mo 5Cr-½Mo	A 182 Gr. F12 Cl.2 A 182 Gr. F5		
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H A 182 Gr. F317	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H		A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	23Cr-12Ni			A 240 Gr. 309H
2.7	25Cr-20Ni	A 182 Gr. F310		A 240 Gr. 310H
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
2.9	23Cr-12Ni 25Cr-20Ni			A 240 Gr. 309S A 240 Gr. 310S
2.10	25Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	
2.11	18Cr-10Ni-Cb		A 351 Gr. CF8C	
2.12	25Cr-20Ni		A 351 Gr. CK20	
3.1	35Ni-35Fe-10Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400

Material Group	Nominal Designation	Forgings	Castings	Plates
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe 64Ni-29.5Mo-2Cr-2Fe-Mn-W	B 462 Gr. N10665 B 462 Gr. N10675		B 333 Gr. N10665 B 333 Gr. N10675
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu 55Ni-21Cr-13.5Mo 55Ni-23Cr-16Mo-1.6Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825 B 462 Gr. N06022 B 462 Gr. N06200		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825 B 575 Gr. N06022 B 575 Gr. N06200
3.9	47Ni-22Cr-9Mo-I8Fe	B 572 Gr. N06002		B 435 Gr. N06002
3.10	25Ni-46Fe-21Cr-5Mo	B 672 Gr. N08700		B 599 Gr. N08700
3.11	44Fe-25Ni-21Cr-Mo	B 649 Gr. N08904		B 625 Gr. N08904
3.12	26Ni-43Fe-22Cr-5Mo 47Ni-22Cr-20Fe-7Mo 46Fe-24Ni-21Cr-6Mo-Cu-N	B 621 Gr. N08320 B 581 Gr. N06985 B 462 Gr. N08367	A 351 Gr. CN3MN	B 620 Gr. N08320 B 582 Gr. N06985 B 688 Gr. N08367
3.13	49Ni-25Cr-18Fe-6Mo Ni-Fe-Cr-Mo-Cu-Low C	B 581 Gr. N06975 B 462 Gr. N08031		B 582 Gr. N06975 B 625 Gr. N08031
3.14	47Ni-22Cr-19Fe-6Mo 40Ni-29Cr-15Fe-5Mo	B 581 Gr. N06007 B 462 Gr. N06030		B 582 Gr. N06007 B 582 Gr. N06030
3.15	33Ni-42Fe-21Cr	B 564 Gr. N08810		B 409 Gr. N08810
3.16	35Ni-19Cr-1¼Si	B 511 Gr. N08330		B 536 Gr. N08330
3.17	29Ni-20.5Cr-3.5Cu-2.5Mo		A 351 Gr. CN7M	

Table 1A List of Material Specifications (ASME B16.5-1996)

Material Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1	A 216 Gr. WCB A 216 Gr. WCC	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si C-Mn-Si-V 21/2Ni 31/2Ni	A 350 Gr. LF 6 Cl.2 A 350 Gr. LF3	A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	A 203 Gr. B A 203 Gr. E

Material Group	Nominal Designation	Forgings	Castings	Plates
1.3	C-Si C-Mn-Si 21/2Ni 31/2Ni		A 352 Gr. LCB	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1	A 217 Gr. WC1 A 352 Gr. LC1	A 204 Gr. A A 204 Gr. B A 204 Gr. C
1.7	C-1/2Mo 1/2Cr-1/2Mo Ni-1/2Cr-1/2Mo 3/4Ni-3/4Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1Cr-1/2Mo 11/4Cr-1/2Mo 11/4Cr-1/2Mo-Si	A 182 Gr. F12 Cl.2 A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	21/4Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.13	5Cr-1/2Mo	A 182 Gr. F5 A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H	A 351 Gr. CF8C	A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	25Cr-12Ni 23Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	A 240 Gr. 309S A 240 Gr. 309H
2.7	25Cr-20Ni	A 182 Gr. F310	A 351 Gr. CK20	A 240 Gr. 310S A 240 Gr. 310H

Material Group	Nominal Designation	Forgings	Castings	Plates
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53 A 182 Gr. F55	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
3.1	35Ni-35Fe-20Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe	B 335 Gr. N10665		B 333 Gr. N10665
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825

Flange Material - Enter the name of the material. This software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. Alternatively, you can click **Matl...** to select a material directly from the **Material Database** dialog box.

Flange Type - Select the flange type: **Weld Neck**, **Long WN**, **Lap Joint**, **Slip On**, **Socket Weld**, **Threaded**, **Studding Outlet**, **FFWn**, **FFSo**, **FFThrd**, **RTJWn**, **Clpg-Thrd**, **Clpg-Sw**, or **None**. This value is not used in the analysis, but is printed in reports.

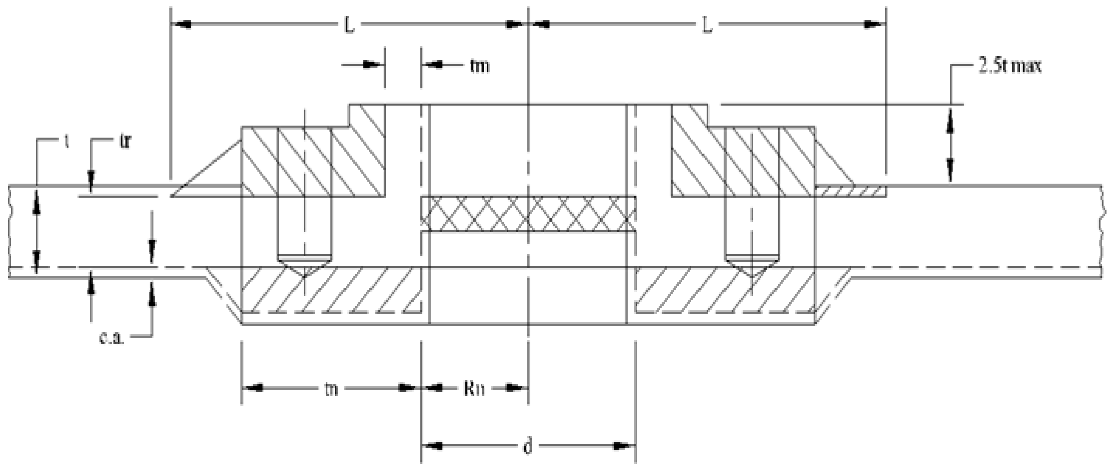
Neglect Areas - Select how the area contributed by the shell or nozzle is handled. Some vessel design specifications mandate that no credit be taken for the area contributed by the shell or nozzle. Select one of the following:

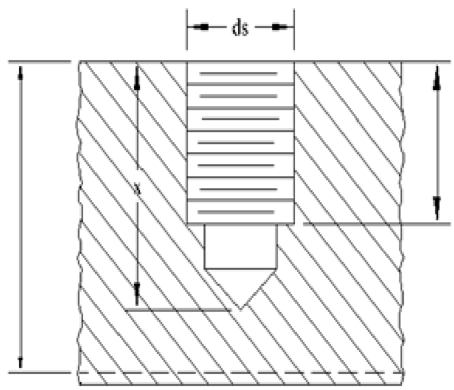
- **A1** - Exclude the available area in the vessel wall.
- **A2** - Exclude available area in the nozzle wall.
- **A1 A2** - Exclude the available areas in both the vessel and nozzle walls.
- **ACWLD -** Exclude the available area in the cover weld.
- **None** - Include all areas.

Tapped Hole Area Loss

Enter the area *S* to exclude when holes are tapped into studding outlets and other similar connection elements. The traditional industry standard is to increase the area required by the tapped hole area loss. Values for tapped area loss are shown in the table below, adapted from the *Pressure Vessel Design Manual:*

ds (in)	**S (in^2)**
5/8	**1.280**
3/4	**1.840**
7/8	**2.500**
1	**3.280**
1 1/8	**4.150**
1 1/4	**5.120**
1 3/8	**6.200**
1 1/2	**7.380**
1 5/8	**8.660**
1 3/4	**10.05**
1 7/8	**11.55**
2	**13.10**
2 1/4	**16.60**





Nozzle Eff. - Enter the seam efficiency of the nozzle groove welds. For nozzle wall thickness calculations, the seam efficiency is always **1.0**. For more information see the definition of E in the ASME Code, paragraph UG-37, and Interpretation VIII-89-171 of the A-90 addenda.

Shell Eff. - Enter the joint efficiency of the shell seam. The seam efficiency is used in "area available" calculations to reduce the area available in the shell. For shell wall thickness calculations, the seam efficiency is always **1.0**. The software accounts for the case where the nozzle passes through a weld by asking for the joint efficiency of the weld.

C Factor - Enter the PD-5500 C factor used in nozzle compensation calculations. This factor accounts for the possibility of external loads. When external loads are negligible, the C factor should not be more than **1.1**. When the nozzle is connected to a piping system, where forces and moments are considered, the C factor should not be greater than **1.0**. For vessels operating in the creep range, C should be less than or equal to **1**.

Local Shell Thk. - If your vessel has insert plates, enter the thickness of the plate. Use this value for vessels having insert plates that are thicker than the surrounding shell. This value is greater than the course thickness of the shell. If the area immediately adjacent to the opening is corroded to a greater degree or locally thinner than the rest of the shell, enter the thinner value. The greater of this value and the element thickness is used in nozzle reinforcement calculations.

User Tr - Enter the minimum required shell thickness *tr*, the actual thickness of the shell or head minus the corrosion allowance. For some vessel designs, the nozzle reinforcement is governed by bending and normal stresses in the local shell area. Under special conditions, project requirements specify that full area replacement for nozzle reinforcement is required. This value replaces the value that the software normally calculates for the required thickness based on internal or external pressure.

NOTE Optionally, for vertical vessels, select **Consider External Loads for Nozzle Tr** in the **Nozzle Design Options** section of the **Load Cases** tab. The software determines the maximum thickness based on the highest stress ratio and uses that value for *tr* if it governs over the required thickness based on internal or external pressure.

Blind Attached? - Select if there is a blind on the nozzle flange. This is used only to determine the weight of the nozzle. There is no structural effect.

Manway/Acs Ope? - Select if this is a manway, access, or inspection opening to bypass the UG-45 minimum nozzle neck thickness requirement. In these cases, paragraph UG-45 states that the minimum thickness requirement according to UG-45 is not required. This option is not used for PD-5500.

Fatigue Calc?

Select to perform a fatigue analysis of the nozzle-to-shell weld. Along with specification of **Weld Class** for PD5500 and EN13445 models, the number and magnitude of fatigue pressure stress cycles must be specified in **Fatigue Analysis** on the **Load Cases** tab.

The table below provides descriptions for ASME VIII-2 2007 and later fatigue curves.

Table Name	**Description**
Table 3.F.1	Coefficients for Carbon, Low Alloy, Series 4xx, High Alloy Steels and High Tensile Strength Steels for Temperatures not Exceeding 700ºF (371ºC) UTS less than or equal to 80ksi (552MPa)
Table 3.F.2	Coefficients for Carbon, Low Alloy, Series 4xx, High Alloy Steels and High Tensile Strength Steels for Temperatures not Exceeding 700ºF (371ºC) UTS (115-130ksi) (793-892MPa)
Table 3.F.3	Coefficients for Series 3xx High Alloy Steels, Nickel Chromium Alloy, and Nickel Copper Alloys for Temperatures not Exceeding 800ºF (427ºC) where Sa > 28.2 ksi (195 MPa)
Table 3.F.4	Coefficients for Series 3xx High Alloy Steels, Nickel Chromium Alloy, and Nickel Copper Alloys for Temperatures not Exceeding 800ºF (427ºC) where Sa less than or equal to 28.2 ksi (195 MPa)
Table 3.F.5	Coefficients for Wrought 70 Copper-Nickel for Temperatures not Exceeding 700ºF (371ºC) Sigmays = 18 ksi (134 MPa)

Table 3.F.6	Coefficients for Wrought 70 Copper-Nickel for Temperatures not Exceeding 700ºF (371ºC) Sigmays = 30 ksi (207 MPa)
Table 3.F.7	Coefficients for Wrought 70 Copper-Nickel for Temperatures not Exceeding 700ºF (371ºC) Sigmays = 45 ksi (310 MPa)
Table 3.F.8	Coefficients for Nickel-Chromium-Molybdenum-Iron, Alloys X,G,C-4, and C-276 for Temperatures not Exceeding 800ºF (427ºC)
Table 3.F.9	Coefficients for High Strength Bolting for Temperatures not Exceeding 700ºF (371ºC)

Weld Class - Select a weld class when **Fatigue Calc?** is selected. The weld classes, descriptions, and illustrations are found in PD:5500 Annex C. Class C is the least severe, while Class W is the most severe. This entry is only available for PD 5500 and EN 13445 models.

Piping Attached - Click to open the Drain Piping Input Dialog Box to set the nozzle piping coordinates.

Quick Results - Click to see a report of nozzle results.

Local Stress Analysis (WRC 107, 297 or Annex G) Tab (Nozzle Input/Analysis Dialog Box)

Defines loading information for nozzle stress analysis. Local loads are entered for stress analysis according to the British code PD:5500. Local or global loads are entered for stress analysis according to the WRC 107/537 and WRC 297 bulletins. The software also checks global loads against ASME VIII-2 allowables. Loads are categorized as sustained, expansion, and occasional.

NOTE In 2010 WRC bulletin 537 was released. The results of the local stress calculation of this bulletin are effectively identical to that of WRC bulletin 107. Bulletin 537 simply provides equations in place of the dimensionless curves found in bulletin 107. Please review the Forward in bulletin 537 for more information.

Calculation Method - Select the needed analysis method. The appropriate force and moment fields are available for the selected method. Select **PD:5500**, **WRC 107**, **WRC 297**, or **No Calc**. When you change the method, the software automatically converts entered loads into the respective coordinate systems used by each method.

Load Convention System - Select **Local** to defined loads locally with respect to the vessel and the nozzle, having the benefit of being independent of the orientation of the vessel. Select **Global** to use the global coordinate system. This entry is only available when **WRC 107/537** is selected for **Calculation Method**.

Quick Results - Click to see a report of local stress analysis quick results.

Nozzle Load Table - Click to open and edit a Microsoft Office Excel workbook with standard nozzle loads that can be imported into the software. You can modify the nozzle loads for Project A and AS1210 projects in this file. These loads are typically listed in the specifications of many engineering companies. After the needed nozzle loads are entered in the workbook, then these loads can be imported and applied to a nozzle depending upon its size and rating. This saves time and reduces chances of error. A default sample workbook is provided.

Loading - Click to select the loading project from the Excel load table workbook. Select **Project A Loadings** or **AS1210 Loadings**.

CAESAR II - Click to import nozzle loads from a CAESAR II .C2 file. This command is only available when **Local** is selected for **Load Convention System** and when CAESAR II is installed. CAESAR II is a separately-purchased Intergraph product.

Node # - Enter the node number of the nozzle used in CAESAR II.

Computed Stress Intensities/Ratios at the Nozzle Edge and Pad Edge - Displays the calculated stress intensities and stress ratios for **Vessel at Nozzle Edge**, **Vessel at Pad Edge**, and **In Nozzle at Vessel**. For example:

	Computed Stress Intensities/Ratios at the nozzle edge and pad edge	Pass/Fail Status
Vessel at Nozzle Edge :	Maximum calculated stress ratio: 0.323	Passed
Vessel at Pad Edge :	Maximum calculated stress ratio: 0.644	Passed

PD:5500 Annex G Analysis

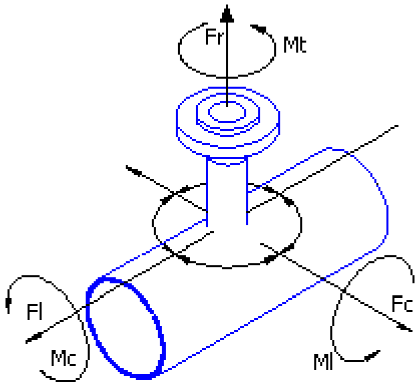
Enter values for the following options when **PD:5500** is selected for **Calculation Method**. The software calculates stresses in cylindrical or spherical geometries with or without reinforcing pads. Only stresses for round hollow nozzle geometries are calculated.

IMPORTANT The example in PD:5500 Annex W does not calculate the membrane stress at attachment edge; you must manually check the membrane stress when selecting the wall thickness. In addition, the membrane stress calculated at the attachment edge, according to Annex G, contains intensified stresses due to the presence of the hole.

Loads

Sustained loads (weight+pressure+forces) are used.

The following force/moment convention is used for PD:5500, Annex G:



Fr - Radial Force
Fc - Shear Force
Fl - Shear Force
Mc - Moment, Fl axis
Ml - Moment, Fc axis
Mt - Torsional moment

Length "L" - Enter in the length of the vessel. For vessels without stiffeners or cones this is the entire vessel length including heads. This value is used along with **Tangent Offset Distance** to calculate the equivalent length for off center loading.

Tangent Offset Distance - Enter in the distance of the centerline of the nozzle from the left tangent line or appropriate line of support. This value is used in conjunction with the **Length "L"** to calculate the equivalent length for off center loading.

Allowable Stress Intensity Factors at Nozzle Edge

Factor for Membrane Stresses - Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane stress intensity. These stresses are in rows 32, 33 and 34 in the printout samples in PD 5500 Annex W. This factor normally has a value of **1.2** or lower at the edge of the reinforcement pad.

NOTES

- This factor is higher than **Factor for Memb Stresses** at pad edge.
- This entry is only available when **Print Membrane Stress at Nozzle Edge** is selected.

Factor for Membrane + Bend Stresses - Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane plus bending stress intensity. These stresses are in rows 27, 28 and 29 in the printout samples in PD 5500 Annex G. This factor normally has a value of **2.25** or lower. At the pad edge, this factor is normally **2.0**.

Print Membrane Stress at Nozzle Edge - Select to calculate membrane stress at the attachment junction. You must also enter a value for **Factor for Membrane Stresses**.

Allowable Stress Intensity Factors at Pad Edge

Factor for Memb Stresses - Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane stress intensity. These stresses are in rows 32, 33 and 34 in the printout samples in PD 5500 Annex W. This entry normally has a value of **1.2** or lower at the edge of the reinforcement pad.

Factor for Memb + Bend Stresses - Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane plus bending stress intensity. These stresses are in rows 27, 28 and 29 in the printout samples in PD 5500 Annex G. At the nozzle edge, this factor normally has a value of **2.25** or lower. At the pad edge, this factor is normally **2.0**.

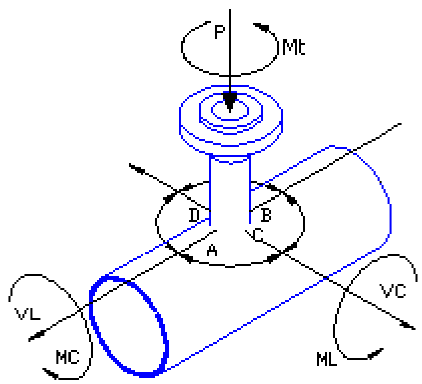
WRC 107/537 Analysis - Local Load Convention

Enter values for the following options when **WRC 107/537** is selected for **Calculation Method** and **Local** is selected for **Load Convention System**. **Local** has the benefit of being independent of the orientation of the vessel. The software calculates stresses for sustained, expansion, and occasional loads and compares stress intensities to allowables.

You can enter values in the following load sets:

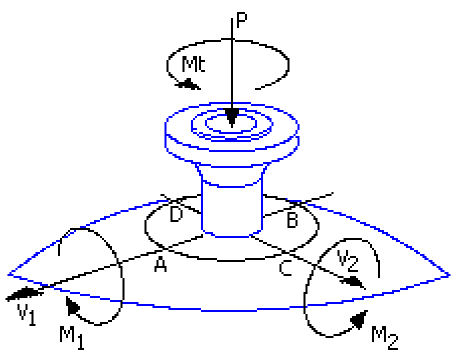
- **Sustained** - (SUS) Primary loads, typically weight + pressure + forces.
- **Expansion** - (EXP) Secondary thermal expansion loads.
- **Occasional** - (OCC) Irregularly occurring loads such as wind loads, seismic loads, and water hammer.

The following WRC 107/537 force/moment convention is used for a **cylindrical** vessel:



P - Radial Force
Vc - Circ. Shear Force
Vl - Long. Shear Force
Mc - Circ. Moment
Ml - Long. Moment
Mt - Torsional Moment

The following WRC 107/537 force/moment convention is used for a **spherical** vessel:



P - Radial Force
V2 - Shear Force, D to C
V1 - Shear Force, B to A
M1 - Moment, B axis
M2 - Moment, C axis
Mt - Torsional moment

Occasional Press Difference - Enter the *difference* between the peak pressure of the system and the system design pressure. The value is always positive or zero. This value is superimposed onto the system design pressure to evaluate the primary membrane stress due to occasional loads. The additional thrust load due to this pressure difference is accounted for in the nozzle radial loading if you also select **Include Pressure Thrust**.

Include Pressure Thrust - Select to include the pressure thrust force (P*A) in the nozzle axial load. For more information on pressure thrust, see the *July 2001 COADE Newsletter http://www.coade.com/newsletters/jul01.pdf*.

Use Division 2 Stress Indices - Select to include the pressure stress indices described in ASME Sec. VIII Div. 2 Table AD-560.7. This value should *only* be used to perform a fatigue analysis. Check ASME VIII Div.2 paragraph AD-160 s to see if the fatigue effect needs to be considered. These factors are used for estimating the peak stress intensity due to internal pressure.

NOTES

- Peak stress intensity due to external loads is included in the analysis by selecting **Use Kn and Kb (to find SCF)**. For normal (elastic) analysis, do not select this option or **Use Kn and Kb (to find SCF).**
- The software does *not* perform the complete fatigue analysis of Section VIII Div.2 Appendix 4 and 5 rules. Instead, the value of peak stress intensity is reported for fatigue effect comparison. For more information, see the *June 2000 COADE newsletter http://www.coade.com/newsletters/jun00.pdf*.

Use WRC 368 - Select to compute pressure stresses in the shell and nozzle according to WRC 368. WRC 368 provides a method for calculating stresses in a cylinder-to-cylinder intersection (such as cylinder-to-nozzle) due to internal pressure and pressure thrust loading.

NOTES

- Using WRC 368 with WRC 107/297 is not accurate for calculating the combined stress from pressure and external loads. So, this option is only available when the attachment type is round and when no external loads are specified.
- For more information on WRC 368 and pressure thrust, see *Modeling of Internal Pressure and Thrust Loads on Nozzles Using WRC-368* in the *July 2001 COADE Newsletter http://www.coade.com/newsletters/jul01.pdf*.

Use Kn and Kb (to find SCF) - Select to include the WRC 107/537 Appendix B stress concentration factors (Kn and Kb) in a fatigue analysis. This value should *only* be used to perform a fatigue analysis. Check ASME VIII Div.2 paragraph AD-160 s to see if the fatigue effect needs to be considered. Also enter the needed value for **Fillet Radius Nozzle** or **Fillet Radius Pad**.

NOTES

- Peak stress intensity due to internal pressure is included in the analysis by selecting **Include Pressure Stress Indices per Div. 2?**.
- For normal (elastic) analysis, do not select this option or **Use Division 2 Stress Indices.**
- The software does *not* perform the complete fatigue analysis of Section VIII Div.2 Appendix 4 and 5 rules. Instead, the value of peak stress intensity is reported for fatigue effect comparison. For more information, see *WRC-107 Elastic Analysis v/s Fatigue Analysis* in the *June 2000 COADE newsletter http://www.coade.com/newsletters/jun00.pdf*.

Fillet Radius Nozzle - Enter the fillet radius between the nozzle and the vessel shell. The software uses this value to calculate the stress concentration factors Kn and Kb according to Appendix B of the WRC 107 bulletin. A value of **0** sets Kn and Kb to 1.0.

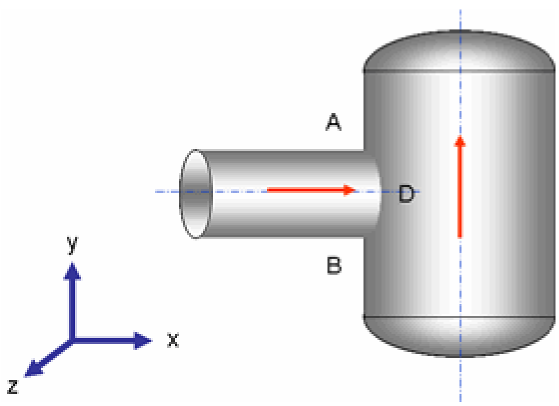
Fillet Radius Pad - Enter the fillet radius between the pad and the vessel shell. The software uses this value to calculate the stress concentration factors Kn and Kb for the vessel/pad intersection, according to Appendix B of the WRC 107 bulletin. A value of **0** sets Kn and Kb to 1.0.

WRC 107/537 Analysis - Global Load Convention

Enter values for the following options when **WRC 107/537** is selected for **Calculation Method** and **Global** is selected for **Load Convention System**. **Global** has the benefit of using the global coordinate system also used by other analyses. When you toggle between the global and local convention systems, the software converts the loads. Options for the global load convention are the same as for **WRC 107/537 Analysis - Local Load Convention** **(on page 84),** except as described below.

Direction Cosines - Displays the direction cosines as described below. You do not usually need to change these values.

The following global convention system is used for a **cylindrical** vessel:



The vessel direction is +Y direction
The nozzle direction is +X direction (towards the vessel)

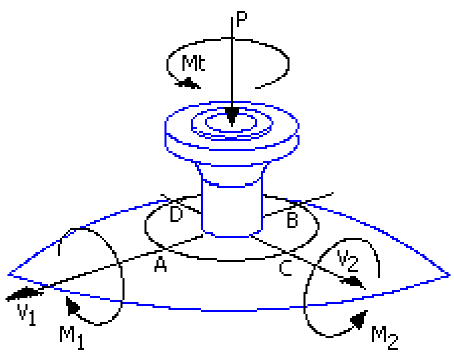
Direction cosines of the vessel are:

- **VX** - 0
- **VY** - 1
- **VZ** - 0

Direction cosines of the nozzle are:

- **NX** - 1
- **NY** - 0
- **NZ** - 0

The following global convention system is used for a **spherical** vessel:



The direction of a spherical vessel is from points **B** to **A**

The software uses these direction vectors to transfer the global forces and moments from the global convention into the traditional WRC107 convention.

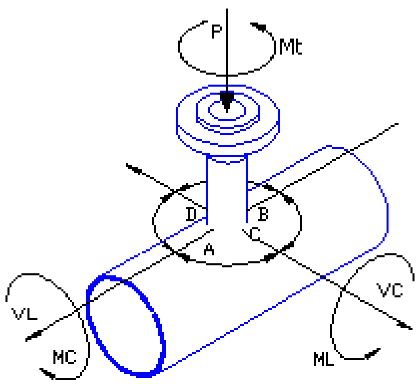
WRC 297 Analysis

Enter values for the following options when **WRC 297** is selected for **Calculation Method**. The software calculates stresses in cylindrical nozzles attached to cylindrical shells.

Loads

Sustained loads (weight+pressure+forces) are used.

The following force/moment convention is used for WRC 297:



P - Radial Force
VC - Shear Force, D to C
VL - Shear Force, B to A
MC - Moment, B axis
ML - Moment, C axis
Mt - Torsional moment

Stress Concentration Factors

Shell Stress Concentration Factor - Enter a value, typically between **1** and **3**, for stress concentration due to weld quality and dimensions in the immediate vicinity of the weld. The stress concentration factor:

- Accounts for *peak stresses* - local stress risers in the immediate vicinity of vessel welds due to factors such as sharp corners and lack of fillet weld radii. Peak stresses are considered in fatigue analysis.
- Applies to the stress calculations in the vessel and the nozzle on both the inside and the outside of the vessel.
- Is used in pressure stress calculations in the vessel on both the inside and outside of the vessel.

Nozzle Stress Concentration Factor - Enter a value, typically between **1** and **3**, for stress concentration due to weld quality and dimensions in the immediate vicinity of the weld. The stress concentration factor:

- Accounts for *peak stresses* - local stress risers in the immediate vicinity of vessel welds due to factors such as sharp corners and lack of fillet weld radii. Peak stresses are considered in fatigue analysis.
- Applies to the stress calculations in the vessel and the nozzle on both the inside and the outside of the vessel.
- Is not used in pressure stress calculations.

Include Pressure Thrust - Select to include the pressure thrust force (P*A) in the nozzle axial load. For more information on pressure thrust, see the *July 2001 COADE Newsletter http://www.coade.com/newsletters/jul01.pdf*.

Use Division 2 Stress Indices - Select to include the pressure stress indices described in ASME Sec. VIII Div. 2 Table AD-560.7. This value should *only* be used to perform a fatigue analysis. Check ASME VIII Div.2 paragraph AD-160 s to see if the fatigue effect needs to be considered. These factors are used for estimating the peak stress intensity due to internal pressure.

NOTES

- Peak stress intensity due to external loads is included in the analysis by selecting **Use Kn and Kb (to find SCF)**. For normal (elastic) analysis, do not select this option or **Use Kn and Kb (to find SCF).**
- The software does *not* perform the complete fatigue analysis of Section VIII Div.2 Appendix 4 and 5 rules. Instead, the value of peak stress intensity is reported for fatigue effect comparison. For more information, see the *June 2000 COADE newsletter http://www.coade.com/newsletters/jun00.pdf*.

Weight

Home tab: **Details** > **Weight**

Adds piping and miscellaneous weight to the selected cylinder or head element. This is weight that cannot be accounted for in other commands. The weight is a static mass, not an applied force, but affects the natural frequency of the vessel and axial stress calculations. Piping weight is modeled here. The area and mass of the piping are considered in the same manner as a weight.

Previous Weight - If you created more than one weight on the element, click to go back to the previous weight.

Go To Next Weight - If you created more than one weight on the element, click to go to the next weight.

Add New Weight - Click to add a new weight to the shell or head element.

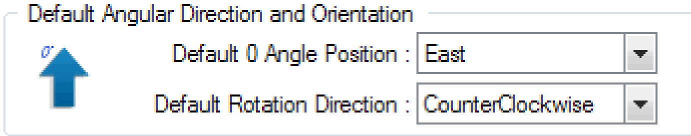
Delete - Deletes all data for the current weight.

Common Detail Parameters (on page 52)

Layout Angle - Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the weight at the point where it is attached to the shell.

For a horizontal vessel, the zero degree line is at 12:00 (looking at a clock); 90 degrees is at 3:00. Entering these layout angles is important if the horizontal vessel has a liquid level and the nozzles are being designed using **Design P + Static Head**, selected for **Nozzle/Clip Design Pressure Options** on the **Load Cases** tab. For a vertical vessel, the angle is more arbitrary. For purposes of rendering the graphics, the assumption is that the zero degree line is at 3:00 and 90 degrees is 12:00.

The position of the zero degree reference direction (North, West or East) and the angular rotation (clockwise or counterclockwise) are set in **Tools** tab, **Set Configuration Parameters**, *Set Default Values Tab (Configuration Dialog)* (see "*Set Default Values Tab (Configuration Tab)*" on page 207).



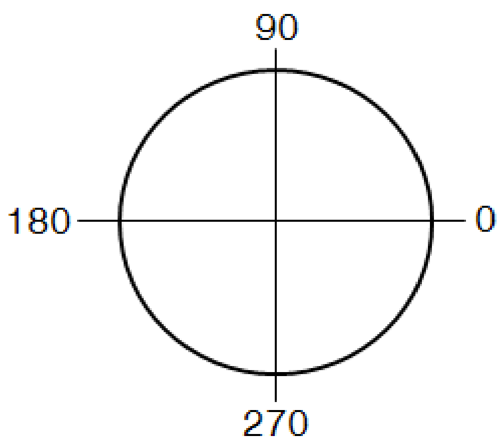
If the new orientation is different from the orientation of the current model, the software asks to update the model with the new settings. If the settings are updated, the weights and areas of platforms must be recalculated:

- Click **List Dialog** (on page 193). On the **Detail Listing** dialog box, click the **Platform** tab so that platform data is visible. Click **Platform Wind Area** and **Platform Weight** to recalculate.
- Click **Platform Input** for each platform. Tab through the fields in the **Platform** dialog box, causing the weight and area to be recalculated

NOTE The angular settings apply to nozzles, clips, legs, lifting lugs, support lugs, base rings, platforms, weights and half-pipe jackets.

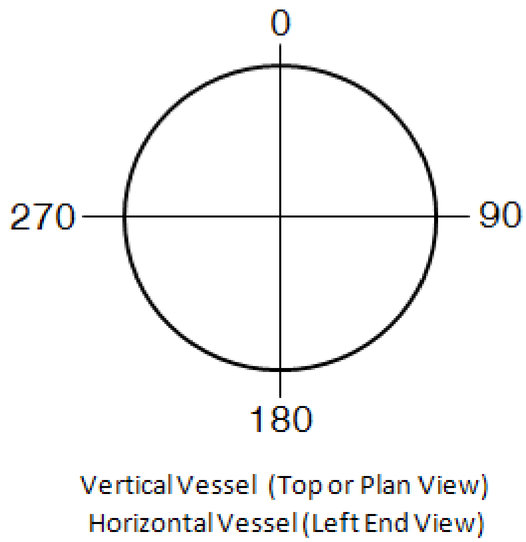
Examples

Default Orientation



Vertical Vessel (Top or Plan View)
Horizontal Vessel (Left End View)

Angle Position: North
Rotation Direction: Clockwise



Offset from Element Centerline - Enter the distance of this weight from the centerline of vessel. The value is multiplied by the weight to obtain a moment that used in stress calculations. For horizontal vessels, the weight is added to the saddle loads and this value is not used.

Miscellaneous Weight - Enter the weight for items such as: an attached motor or other equipment, internal piping, or external structural elements. This value is also used for seismic analysis.

Select the Active Cases for this Weight/Mass - Select the active case for the current weight/mass. You can select any combination of empty, operating, and hydro test cases.

Is this a Welded Internal? - Select if the weight is for an item that is welded to the vessel during shop construction. The weight is then added to the fabricated total weight, other weights that are functions of the fabricated total weight, and the empty total weight.

Area of External Weight/Piping/Equipment - Enter the area to use for the wind load calculation.

Piping Detail - Select **Is this a Piping Detail?** to include the weight and moment of overhead pipelines on vertical pressure vessels. Define the area and weight of the piping with the following options:

- **Pipe Lookup** - Click to open the **Seamless Pipe Selection** dialog box and select values from the piping database for **Pipe Schedule** and **Nominal Pipe Diameter**.
- **Pipe Outer Diameter** - Displays the OD for the pipe selected in **Pipe Lookup**. You can also manually enter a value.
- **Pipe Thickness** - Displays the nominal thickness for the pipe selected in **Pipe Lookup**. You can also manually enter a value.
- **Fluid Specific Gravity** - Enter the specific gravity of the contained fluid. This value is usually **1.0**.

- **Insulation Thickness** - If the pipe is insulated, enter the thickness of the insulation. If there is no insulation, enter **0**.
- **Insulation Density** - Enter the density of the pipe insulation to calculate the weight of the insulation and the moment effect.
- **Compute Weight and Area** - Click to calculate the pipe weight and area based on the entered values.

After weights and offsets are entered, the software calculates the overturning moment due to the eccentricity of the piping. In most designs, the piping is supported by means of braces and clips at specified intervals. The element on which the piping weight is added takes the applied load.

NOTE If piping is specified on a top head, the software attempts to graphically connect the piping to the center-most nozzle.

Force and Moment

Home tab: **Details** > **Force and Moment**

Adds external forces and moments to the selected cylinder or head element. In most cases these are operating loads imposed on the vessel, such as piping loads on nozzles.

Previous For/Mom - If you created more than one force/moment set on the element, click to go back to the previous set.

Go To Next For/Mom - If you created more than one force/moment set on the element, click to go to the next set.

Add New For/Mom - Click to add a new force/moment set to the shell or head element.

Delete - Deletes all data for the current force/moment set.

Common Detail Parameters (on page 52)

Applied Forces - Enter the force in each needed direction. For **X**, positive is from left to right. For **Y**, positive is upward. For **Z**, positive is towards you. Forces perpendicular to the vessel are resolved into a single vector and applied with live load to create the worst load combination. Unlike **Weight**, applied forces are not included in seismic analysis because force does not have mass.

NOTE You should generally enter negative Y forces (downward) because these increase the loads on the saddles and other supports.

Applied Moments - Enter the moment about each needed axis. For **X**, positive is from left to right. For **Y**, positive is upward. For **Z**, positive is towards you.

Compute Stresses Due to Applied Loads - Select one or both of the following:

- **Compute Longitudinal Stresses BW normally added to the Wind Case** - Forces and moments act during wind analysis.
- **Compute Longitudinal Stresses BS normally added to the Seismic Case** - Forces and moments act during seismic analysis.

IMPORTANT You must select at least one option.

Force/Moment Combination Method - Select the method for resolving forces and moments about the support point:

- **SRSS** - All forces and moments act in the same direction. The software takes the absolute value of the entered loads to determine the overall effect on the structure. This method, while not technically correct, yields a conservative result for bending stresses and support calculations.
- **Algebraic** - Forces and moments act in their positive or negative directions. Loads that oppose each other tend to cancel out. If you have an accurate account of the external forces and moments, due to load conditions such as piping reactions, this option provides more accurate and less conservative results. The software sums the forces and moments in both the X and Z planes for vertical vessels. The greater of the two moments is used in calculations of the stresses and moments at the support.

Platform

Home tab: **Details** > **Platform**

Adds a platform to the selected element.

Previous - If you created more than one platform on the element, click to go back to the previous platform.

Next Platform - If you created more than one platform on the element, click to go to the next platform.

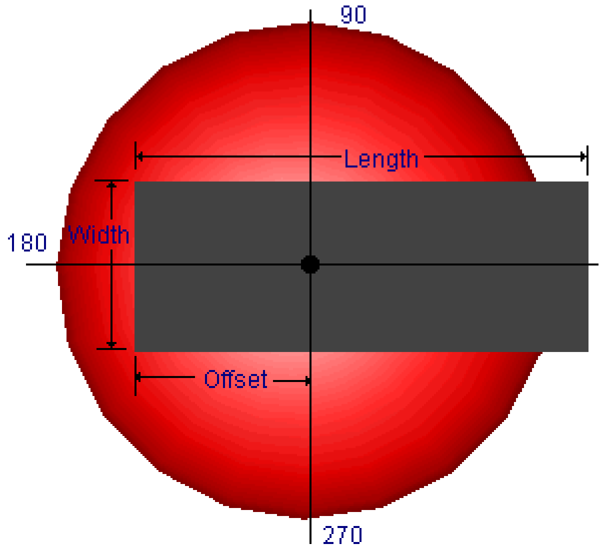
Add Platform - Click to add a new platform to the shell or head element.

Delete - Deletes all data for the current platform.

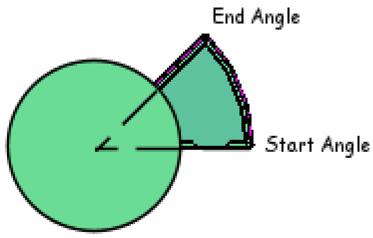
Common Detail Parameters (on page 52)

Non-Circular Platform? - Select if the platform is not circular. This option is selected by default for platforms that cannot be circular, such as on a horizontal cylinder.

Layout Angle - For a non-circular platform, enter the rotation angle through the centerline of the platform. For example, on a non-circular top head platform:



Platform Start Angle - For a circular platform, enter the angle between the designated zero degree line of the vessel and the start angle of the platform.



Platform End Angle - For a circular platform, enter the angle between the designated zero degree line of the vessel and the ending angle of the platform.

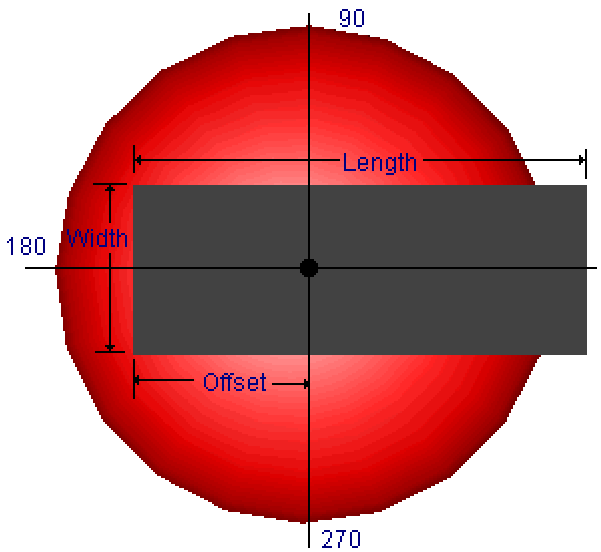
Platform Railing Weight - Enter the weight of the platform railing in units of weight/length. This value is used as part of calculating the weight of the platform.

Platform Grating Weight - Enter the weight of the platform deck grating or plate in units of weight/area. This value is used as part of calculating the weight of the platform. To use standard weights, click ... and select **Open Lattice Grating** or **Checkered Floor Plate**.

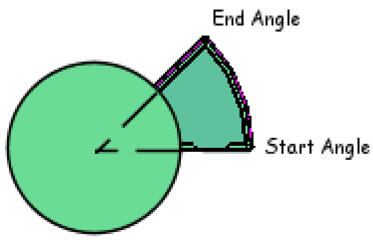
Platform Width - Enter the radial width of the platform. This dimension is used to calculate the weight of the platform in conjunction with **Platform Railing Weight and Platform Grating Weight.**

Platform Height - Enter the height from the bottom of the platform to the top rail. Usually this distance is to be no less than 42 inches. This dimension is used to calculate the wind area of the platform in conjunction with **Platform Width**, **Clearance**, and **Force Coefficient Cf**.

Offset from End - For a non-circular top head platform, enter the offset as shown below. The software uses the offset to calculate the distance to the center of the platform. This is then used as the offset dimension to calculate the eccentric moment for the platform. This value is used in a .DXF drawing of the vessel when **Export** (see "**Import/Export**" on page 44) is used.



Clearance - For circular platforms, enter the distance between the inside edge of the platform and the outside surface of the cylindrical shell. The platform clearance is used to calculate the wind area of the platform.



For a circular top head platform, enter **0**.

Platform Length (Non-Length) - For a non-circular top head platform, enter the long dimension of the platform.

Force Coefficient Cf - Enter the platform force coefficient *Cf*. This coefficient may be specified in a wind load computation standard, such as Table 6-9 of ASCE7-95. The value is typically between **1.2** and **1.8**. This factor is used to calculate the wind area of the platform in conjunction with the **Platform Height, Platform Width**, and **Clearance**.

Platform Wind Area - Enter the tributary wind area of the platform. This value is typically the greatest span of the platform perpendicular to the vessel multiplied by a nominal platform height of between 12 and 36 inches on the hand rails and other equipment on the platform.

To change the platform wind area calculation method, click **Installation | Misc. Options** on the **Load Cases Tab** (see "**Stress Combination Load Cases (Load Cases Tab)**" on page 336) of the main window. The methods are:

1. The height times the width times the force coefficient (conservative).
2. One-half of the floor plate area times the force coefficient.
3. The height times the width times the force coefficient divided by three.
4. The projected area of the platform times the force coefficient divided by three. This option yields the same results as option 3 for platforms that have a sweep angle of greater than 180º.

Control Options - Select one or more of these options:

- **Do not include platform in analysis** - Select to remove platform data from analysis.
- **User computes and enters the platform area** - Select to manually enter a value in **Platform Wind Area** and override software platform area calculations.
- **User computes and enters the platform weight** - Select to manually enter a value in **Platform and Ladder Weight** and override software platform weight calculations.

Ladder Layout Angle - Enter the angle between the designated zero degree line of the vessel and the centerline of the ladder.

Ladder Start Elevation - Enter the start elevation of the ladder. This value and **Ladder Stop Elevation** are used to determine the overall length of the ladder. The software assumes that the top of the ladder is attached to the platform.

Ladder Stop Elevation - Enter the stop elevation of the ladder. This value and **Ladder Start Elevation** are used to determine the overall length of the ladder. The software assumes that the top of the ladder is attached to the platform.

Ladder Unit Weight - Enter the unit weight of the ladder in units of weight/length. For example, in English units this is typically the weight of a one-foot ladder section. The software does not differentiate between a caged and a non-caged ladder, so the total weight of a ladder and cage must be included in this value. The total ladder and platform weight at the platform elevation is used in the calculations.

Is this a Caged Ladder? - Select if the ladder has a cage.

Platform and Ladder Weight - Enter the total weight of the platform, ladders, and associated hardware. This overrides weight calculated automatically by the software.

Packing

Home tab: **Details** > **Packing**

Adds packing data to the selected element.

Previous Packing - If you created more than one set of packing data on the element, click to go back to the previous set.

Go To Next Packing - If you created more than one set of packing data on the element, click to go to the next set.

Add New Packing - Click to add a new set of packing data to the shell or head element.

Delete - Deletes all data for the current set of packing data.

Common Detail Parameters (on page 52)

Height of Packed Section - Enter the height of the packed section along the element. This value is used to calculate the weight of the packed section. For seismic calculations, the weight center of the packed section is taken at half this height. In the rare case of a packed horizontal vessel, the value is the length of the packed section.

Full - If the element is fully filled with packing material, click to calculate the values of **Distance from "From" Node** and **Height of Packed Section.**

Density of Packing - Enter the density of the packing. The following table list typical densities in lbs/ft^3:

Size (in.)	Density (lb/ft^3)	Size (in.)	Density (lb/ft^3)
Ceramic Raschig Ring		**Carbon Raschig Ring**	
1/4	60.0	1/4	46.0
3/8	61.0	1/2	27.0
1/2	55.0	3/4	34.0
5/8	56.0	1	27.0
3/4	50.0	1 1/4	31.0
1	42.0	1 1/2	34.0
1 1/4	46.0	2	27.0
1 1/2	46.0	3	23.0
2	41.0	**Carbon Steel Pall Ring**	
3	37.0	5/8	37.0
4	36.0	1	30.0
Carbon Steel Raschig Ring		1 1/2	26.0
1/4	133.0	2	24.0
3/8	94.0	**Plastic Pall Ring**	
1/2	75.0	5/8	7.25
5/8	62.0	1	5.50
3/4	52.0	1 1/2	4.75
1	39.0	2	4.50
1 1/2	42.0	3	4.50
2	37.0		
3	25.0		

Packing in place during the field hydrotest? - Select this option if the packing will be in place when the field hydrotest is performed.

Percent Volume Hold Up - Enter a percentage value between **0** and **100** for the amount of liquid that the packing retains. Using this value and **Liquid Specific Gravity**, the software calculates the weight of liquid trapped in the packing and adds the weight to the operating weight of the vessel.

Liquid Specific Gravity - Enter the specific gravity of the liquid trapped in the packing. For more information, see **Liquid Density** (see "**Liquid**" on page 115). Using this value and **Percent Volume Hold Up**, the software calculates the weight of liquid trapped in the packing and adds the weight to the operating weight of the vessel.

Saddle

Home tab: **Details** > **Saddle**

Adds a saddle to the selected horizontal cylinder element. The size and location of the saddles are important for Zick calculations of local stresses on horizontal vessels with saddle supports. For proper Zick analysis, only two saddles may be defined; however, they do not have to be symmetrically placed about the center axis of the vessel. If no saddles are defined, dead load and live load calculations are not performed.

Previous Saddle - If you created more than one saddle on the element, click to go back to the previous saddle.

Go To Next Saddle - If you created more than one saddle on the element, click to go to the next saddle.

Add New Saddle - Click to add a new saddle to the shell element.

Delete - Deletes all data for the current saddle.

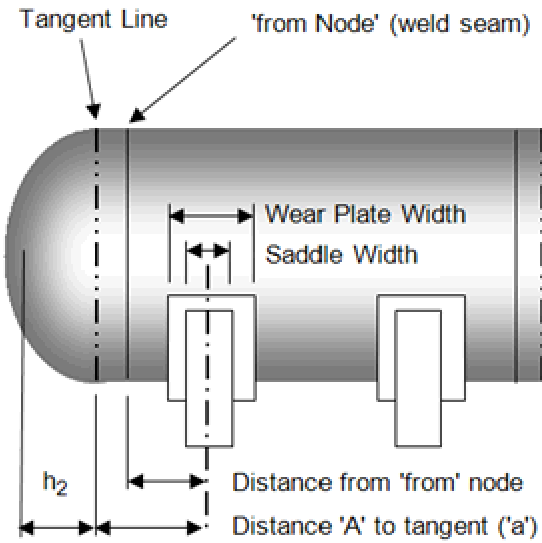
Add Saddle Ring - Click to open the **Stiffening Ring** dialog box to add a ring to the saddle. For more information, see **Stiffening Ring**.

Select Saddle - Click to open the **Saddle Properties** dialog box, and select a standard saddle (from *SaddleData.txt*, found in the PV Elite System folder).

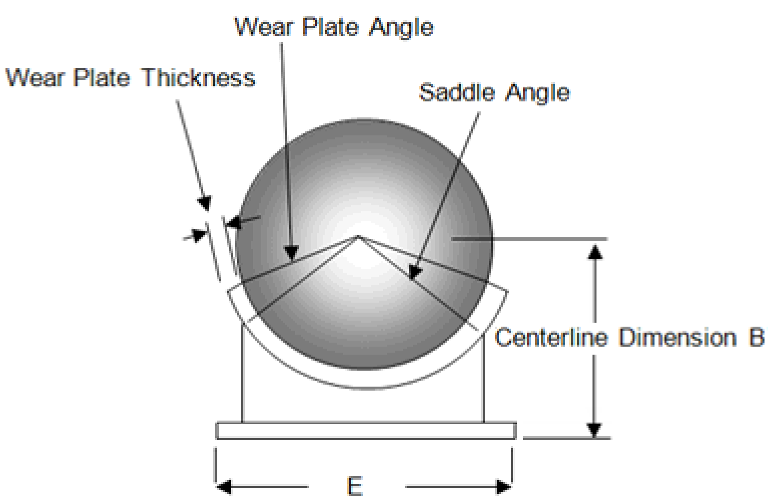
Same as First - Click to copy all data from the first saddle to the current saddle.

Common Detail Parameters (on page 52)

Saddle Width - Enter the width of the saddle support. This value does not include any wear pad on the vessel side, and is used primarily for the Zick analysis of horizontal vessels on saddle supports.



Centerline Dimension B - Enter the distance from the base of the saddle to the centerline of the vessel, referred to as dimension *B* in some pressure vessel texts. This value is used to determine additional saddle loads due to wind or seismic events.



Saddle Contact Angle - Enter the angle between the two contact points ("horns") of the saddle, measured from the axial center of the vessel. This value typically ranges from 120.0º to 150.0º.

Wear Plate Width - Enter the width of the wear plate between the vessel and the saddle support. This value is used primarily for the Zick analysis of horizontal vessels on saddle supports.

Wear Plate Thickness - Enter the thickness of the wear plate between the vessels and the saddle support. This value is used primarily for the Zick analysis of horizontal vessels on saddle supports.

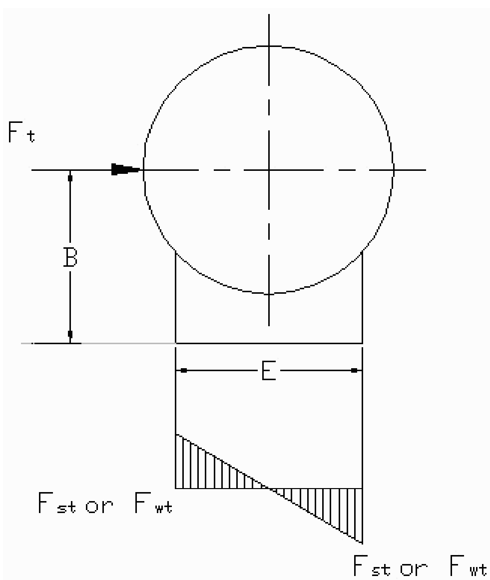
Wear Plate Contact Angle - Enter the angle contained from one edge of the wear plate to the other edge, measured from the axial center of the vessel. Typically this value is approximately 130º.

Height of Section Ring - If a custom fabricated composite (usually T type) stiffener is used over the saddle supports, enter the height from the shell surface to the top of the stiffener. This value will be used to compute the stress at the tip of the stiffener. If a horizontal vessel does not have stiffeners over the saddle supports, enter **0**. If you selected a stiffener from the AISC structural steel database in the **Stiffening Ring** dialog box, enter **0**. In this case, the software gets the ring height from the AISC database.

Friction Coefficient Mu - Enter the coefficient of friction m_u between the base of the saddle and the supporting foundation, piers or structure. A frictionless surface has a m_u value of **0**. Other typical values are in the range of **0.3** to **0.5**. The software uses this value to determine the counteracting force caused by thermal expansion and the dead weight of the vessel on the saddle support. This is essentially a resistive force bending the saddle. The generated force is proportional to m_u times the normal force.

Moment Factor, Ftr - Enter the moment factor for calculating the saddle reaction force due to the wind or earthquake transverse load. The recommended value is **3**.

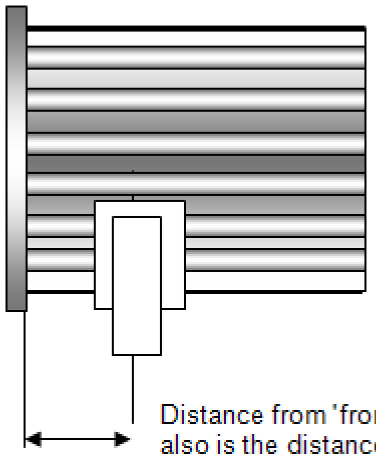
The value of **6** is conservative in that it assumes that the maximum edge load is uniform across the entire base, when realistically it occurs only at the edge. A more accurate method converts this triangular loading into a more realistic uniform load, leading to the value of 3. The following figure shows an end view of a horizontal vessel with a transverse load, simulating wind/seismic loading:



The saddle reaction load Fst (or Fwt for wind) due to the transverse load Ft is:

```
Fst (or Fwt) = (Saddle Moment Factor) * Ft * B / E
```

Saddle Dimension a - Enter the distance between the centerline of the saddle support and the tangent line of the nearest head. This dimension is labeled *A* in most pressure vessel texts.



Dimension E at base - Enter the dimension of the baseplate that is less than the distance in contact with the supporting surface. Dimension E addresses the saddle reaction force due to wind or seismic force when the baseplate distance dimension has a different distance in contact with the supporting surface. This entry is optional.

Circ Eff. over Saddle - Enter the circumferential efficiency in the plane of the saddle.

When you create a shell by welding the ends together, there is a longitudinal weld. If that weld is at the saddle, there are bending moment stress.

Circ Eff. over Midspan - Enter circumferential efficiency at the mid-span.

When you create a shell by welding the ends together, there is a longitudinal weld. If that weld is between saddles, there are bending moment stress.

Wear Plate and Shell Matls are the same? - Select if the wear plate and shell materials are the same.

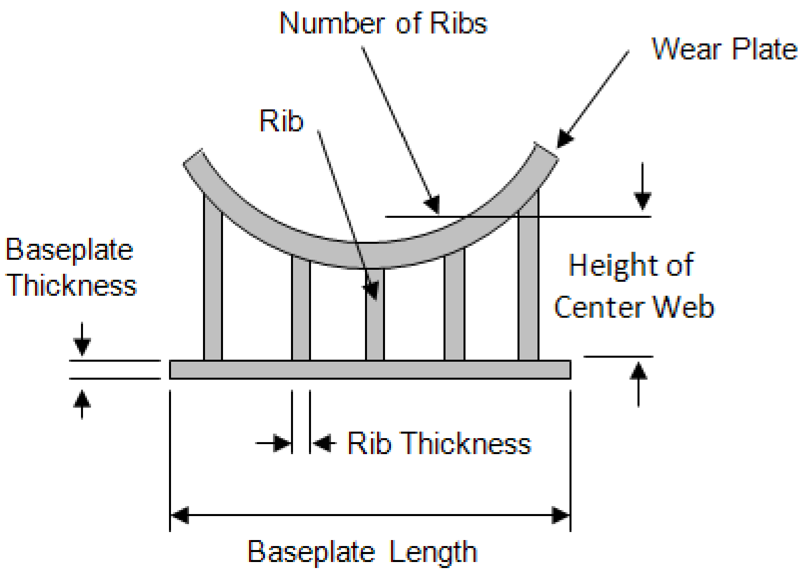
Perform Saddle Check? - Select to add rib, web, and baseplate data, and perform a structural analysis of the saddle.

Saddle Allowable Stress - Enter the saddle allowable stress. Alternatively, click **Matl...** to select a material directly from the **Material Database** (see "**Material Database Dialog Box**" on page 485) dialog box.

Material Yield Stress - Enter the yield stress for the saddles at their design temperature. Alternatively, click **Matl...** to select a material directly from the **Material Database** (see "**Material Database Dialog Box**" on page 485) dialog box.

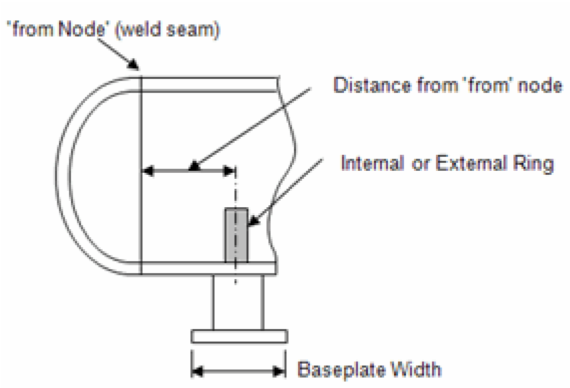
E for Plates - Enter the modulus of elasticity *E* for the saddle material.

Baseplate Length - Enter the long dimension of the baseplate in the direction of the vessel diameter.



Baseplate Thickness - Enter the thickness of the baseplate.

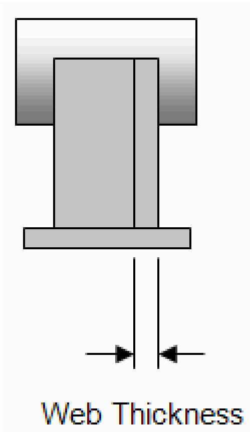
Baseplate Width - Enter the short dimension (the width) of the baseplate.



Number of Ribs - Enter the number of ribs on one saddle, running parallel to the long axis of the vessel.

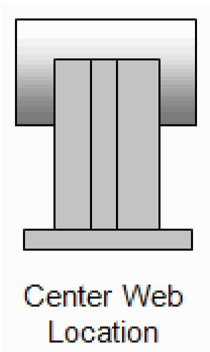
Rib Thickness - Enter the thickness of the rib supports.

Web Thickness - Enter the thickness of the web. The web is the vertical plate between the baseplate and the wear plate, to which the ribs are attached.



Height of Center Web - Enter the distance from the bottom of the center rib to the top plus the thickness of the shell.

Web Location - Select the web location relative to the saddle baseplate and wear plate. Select **Center**, **Side - Left**, or **Side - Right**.

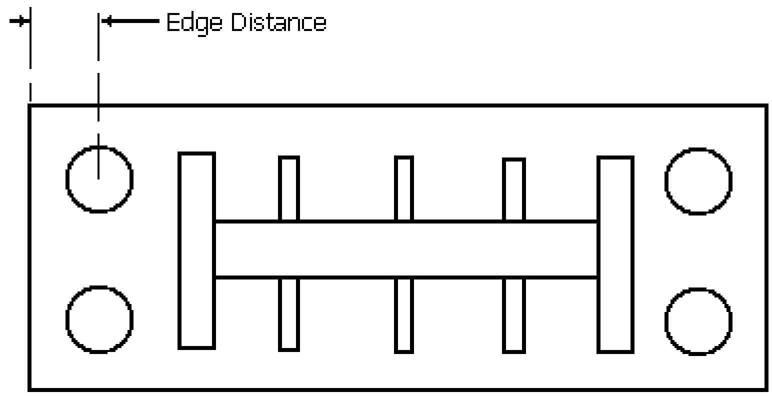


Perform Anchor Bolts Calculations? - Select to add anchor bolt data, and perform anchor bolt and baseplate calculations. Enter values for the options below.

NOTE PV Elite performs anchor bolt calculations for horizontal vessel saddle supports and skirt and leg baseplate supports. This analysis determines the uplift on the saddle supports due to the external loads on the vessel. Loadings include defined forces and moments and wind and seismic loads. If the vessel is in uplift, the required area of the bolts is determined. More information on this analysis is found in the Pressure Vessel Design Manual by Dennis R. Moss, 1997 and later editions. In addition to calculation of the required bolt area, this method also calculates the required thickness of the baseplate due to the applied bolt load. In some cases the bolt load controls the thickness of the saddle baseplate.

Saddle Bolted to Steel Foundation? - Select if the saddles are bolted to a steel substructure.

Number of Bolts - Enter the total number of bolts to be used on the baseplate. The bolts are assumed to be at the edge of the baseplate along the short side.



Num of Bolts in Tension - Enter the number of bolts in tension, generally the total number of bolts divided by two.

Edge Distance - Enter the distance from the edge of the baseplate to the centerline of the bolts.

Bolt Corrosion Allowance - Enter the bolt corrosion allowance (BCA). If the bolt corrosion allowance specified is a "total", then divide it by two; otherwise this calculation will be overly conservative. When dealing with bolt corrosion, the following equation is used to calculate the equivalent bolt OD and corresponding reduced area of the bolt:

$BOD = (BLTAREA * 4.0 / \Pi)^{1/2} - 2.0 * BCA$

Bolt Material - Enter the bolt material. Alternatively, click **Matl...** to select a material directly from the **Material Database** (see "**Material Database Dialog Box**" on page 485) dialog box.

Bolt Allowable Stress - Enter the allowable stress of the bolt. If you select a material directly from the **Material Database** dialog box for **Bolt Material**, the software provides the allowable stress at ambient temperature. If you need the allowable at an elevated temperature (such as for an insulated vessel), then you must enter the allowable stress at that temperature.

Thread Series - Enter the thread series identifier:

- **TEMA -** 8 thread series, adapted from the TEMA Standard
- **TEMA Metric**
- **UNC** - Unified National Course Threads, adapted from Mark's Handbook
- **BS 3643**
- **SABS 1700**
- **User Defined Root Area** - Also enter a value for the root area of a single bolt in **Bolt Root Area**. This information can be obtained from a standard engineering handbook.

Nominal Bolt Diameter - Select the nominal bolt diameter. In general this value ranges from 1/2 inch to 4 inches.

Bolt Root Area - For nonstandard or metric bolts, enter the root cross-sectional area of the bolt. If you have entered a value for **Bolt Corrosion Allowance**, the software modifies the area of the bolt using the equation described above.

Optional Moments for Saddle Analysis - According to definitions in ASME VIII-2 4.15.6 enter values for the following optional moments:

Moment M1 - Net-section maximum, maximum longitudinal bending moment at the saddle support. The moment is negative when it results in a tensile stress at the top of the shell.

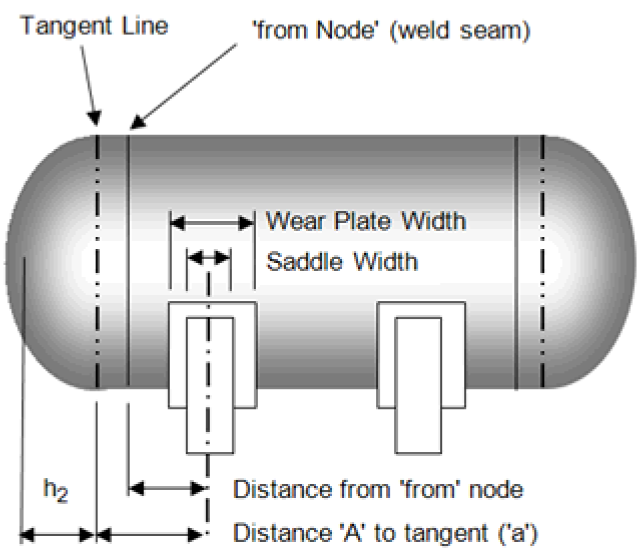
Moment M2 - Net-section maximum, maximum longitudinal bending moment between the saddle support. The moment is negative when it results in a tensile stress at the top of the shell.

The saddle analysis used in PV Elite can be either the method outlined in ASME VIII-2 paragraph 4.15.3 or PD 5500 Annex G. Both of these analyses are based on the original method outlined in the September 1951 Paper by L.P. Zick, "Stresses in Large Horizontal Cylindrical Pressure Vessels on Two Saddle Supports." This paper first appeared in the T*he Welding Journal Research Supplement*. The basis of the analysis is a typical pressure vessel with ellipsoidal, torispherical or hemispherical heads at either end, and cylindrical sections in the middle. The two saddle supports are assumed to be equidistant from the ends. With these assumptions, the bending moments between the saddles and over the saddles can be calculated.

Unfortunately, the geometry of many vessels and exchangers do not fall in line with these assumptions. As a result, the analysis of these vessels may produce moments that are not correct with regard to the original Zick formulas. The software allows the optional input of **Moment M1** and **Moment M2**, as defined in ASME VIII-2 4.15.3, for both the operating and hydrotest conditions. If **Moment M1** and **Moment M2 are zero**, moments are calculated based on the design information provided in the **Saddle** dialog. If **Moment M1** and **Moment M2** are non-zero, these moment values are used, overriding the calculated moments. **Moment M1** and **Moment M2** can be positive or negative.

According to ASME VIII-2, paragraph 4.15.3.2, the moments are calculated for equidistant saddles with $a \leq 0.25L$. If the relationship between *a* and *L* is not maintained, the moments should be calculated using an appropriate engineering beam type analysis method (such as shear and moment diagrams). From ASME VIII-2 4.15.6:

- *a* - Distance from the axis of the saddle support to the tangent line on the curve for a dished head or to the inner face of a flat cover or tubesheet.
- *L* - Length of the cylindrical shell measured from the tangent line for a vessel with dished ends or from the inner face to inner face for vessels with flat covers or tubesheets.



NOTE These overriding moments are only used in Division 2 saddle analysis and PD:5500 saddle analysis.

Tray

Home tab: **Details** > **Tray**

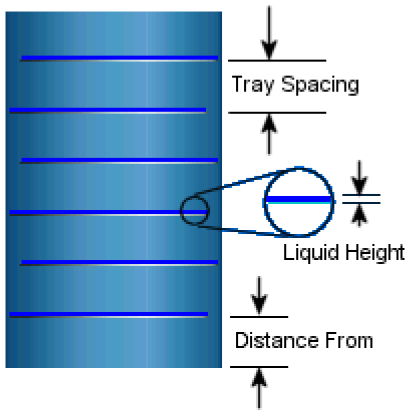
Adds a set of equally spaced trays with a set liquid height to the selected element on a vertical vessel.

Previous Tray Group - If you created more than one tray set on the element, click to go back to the previous set.

Go To Next Tray Set - If you created more than one tray set on the element, click to go to the next set.

Add New Tray Set - Click to add a new tray set to the element.

Delete - Deletes all data for the current tray set.



Common Detail Parameters (on page 52)

Number of Trays - Enter the number of trays on the element.

Tray Spacing - Enter the vertical distance between trays.

Tray Weight Per Unit Area - Enter the unit weight of each tray in the set. Do not enter the total weight, because the software multiplies the unit weight by the cross-sectional area of the element.

Support Ring and Bolting Bar Weight - Enter the support ring and bolting bar weight.

Height of Liquid on Tray - Enter the height of the liquid on each tray.

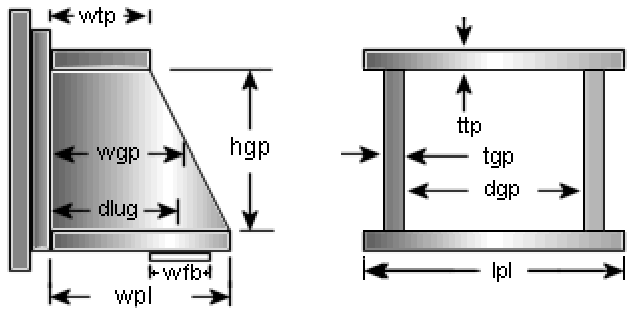
Density of Liquid on Tray - Enter the density of the liquid on each tray. For more information, see **Liquid Density** (see "**Liquid**" on page 115).

Lug

Home tab: **Details** > **Lug**

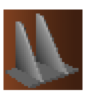
Adds support lugs to the selected element on a vertical vessel. If no skirt or legs are defined for a vertical vessel, the lowest set of lugs are used as the vessel support point for dead load and live load calculations.

Delete - Deletes all data for the lug.

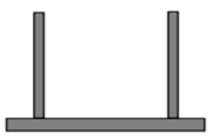


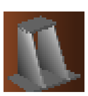
Common Detail Parameters (on page 52)

Lug Type - Select the type of geometry for the support lug:

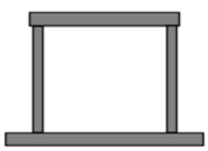


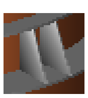
Simple geometry with gussets



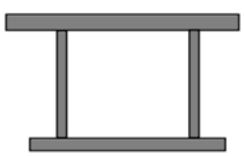


Gusseted geometry with top plate





Gusseted geometry with continuous top encirclement ring



Lug Start Angle - Enter the angle between the designated zero degree line of the vessel and the start angle of the lug.

Lug Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box**

(on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Number of Lugs - Enter the number of support lugs around the periphery of the vessel at this location.

Dist. from OD to Lug MidPt (dlug) - Enter the radial distance from the wall of the vessel to the midpoint where the lug attaches to the structural steel.

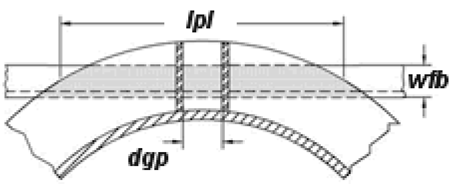
Weight of One Lug - Enter the actual weight of one support lug. The software does not gather enough information to do the detailed calculation of the support lug weight.

Force Bearing Width (wfb) - Enter the width of the structure that is in contact with the bottom lug support plate.

Radial Width of Bottom Ring (wpl) - Enter the distance that the bottom support plate extends from the OD of the vessel. This value must be greater than or equal to **Mean Width of Gussets (wgp)**.

Effective Force Bearing Length (lpl) - For lug types with a bottom plate and no continuous rings (Simple geometry with no gussets and gusseted geometry with top plate in **Lug Type**), enter the distance between gussets plus two times the gusset plate thickness.

For lug types with continuous top and bottom rings (Gusseted geometry with continuous top encirclement ring in **Lug Type**), enter the length of the bottom plate located on a support:



Thickness of Bottom Ring (tpl) - Enter the thickness of the bottom support plate.

Distance Between Gussets (dgp) - Enter the distance between the inside surfaces of the gusset plates.

Mean Width of Gussets (wgp) - Enter the mean gusset width, defined as the gusset width at the top plus the gusset width at the bottom divided by two. The software uses this value to calculate the actual stresses in the gusset plates.

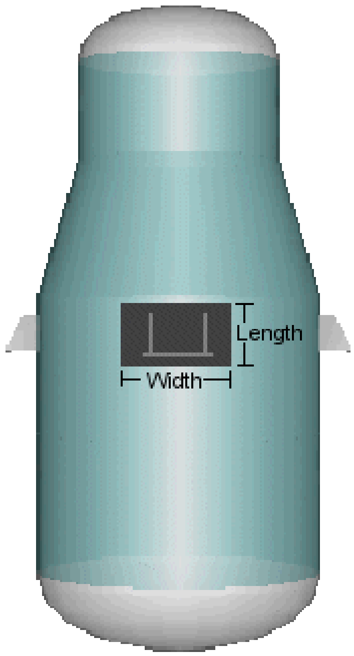
Height of Gussets (tgp) - Enter the height of one gusset.

Thickness of Gussets (tgp) - Enter the thickness of the gusset plate.

Radial Width of Top Ring (wtp) - Enter the radial dimension from the OD of the shell to the edge of the plate. This value should be less than or equal to **Mean Width of Gussets (wgp)**.

Thickness of Top Ring (ttp) - Enter the thickness of the top support plate ring that sits above the gussets.

Perform WRC 107/537 Calculation - Select to perform the WRC 107 local stress analysis on a reinforcing pad for the lug. The software calculates the stresses at the edge of the attachment and the edge of the pad. Enter values for the width, length, and thickness options.



Bolt Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Type of Threads - Enter the thread series identifier:

- **TEMA -** 8 thread series, adapted from the TEMA Standard
- **TEMA Metric**
- **UNC** - Unified National Course Threads, adapted from Mark's Handbook
- **BS 3643**
- **SABS 1700**
- **User Root Area** - Also enter a value for the root area of a single bolt in **Bolt Root Area**. This information can be obtained from a standard engineering handbook.

Nominal Bolt Diameter - Select the nominal bolt diameter. In general this value ranges from 1/2 inch to 4 inches.

Bolt Root Area - When **User Root Area** is selected for **Type of Threads**, enter the root cross-sectional area of the bolt.

Legs

Home tab: **Details** > **Legs**

Adds support legs to the selected element. Legs can be created for a vertical vessel without a skirt element.

Leg Input Parameters Tab (Leg Dialog Box) (on page 109)

Base Plate Parameters Tab (Leg Dialog Box) (on page 112)

Leg Input Parameters Tab (Leg Dialog Box)

Defines parameters for the legs and leg pads.

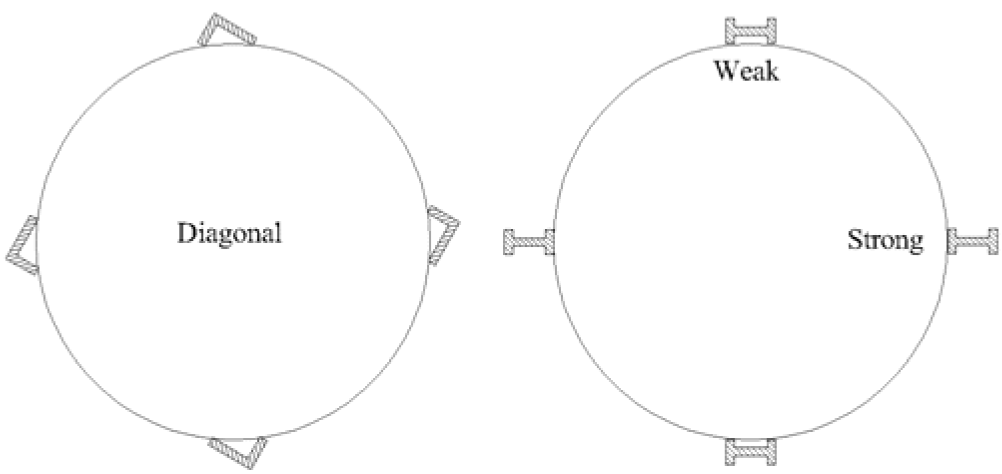
Common Detail Parameters (on page 52)

Delete - Deletes all data for the legs.

Leg Centerline Diameter - Enter the distance between the centerlines of two legs on opposing sides of the vessel. If there are an odd number of legs (therefore, no two legs are opposing), then enter the diameter of a circle drawn through the centerlines of the legs. Alternatively, click **Compute Centerline Diameter** to calculate the value.

Leg Orientation - Select the orientation of the leg cross-section with respect to the centerline. Select:

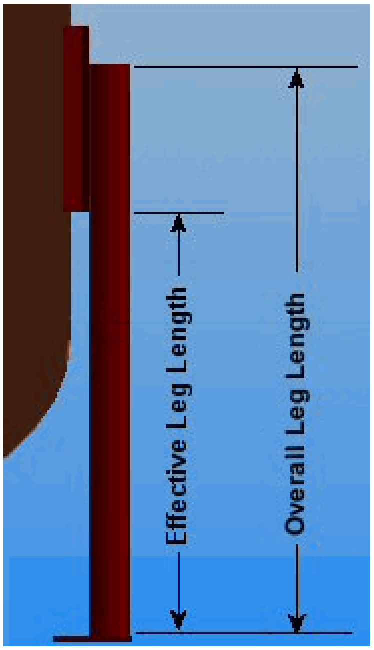
- **Strong Axis** - The strong axis is perpendicular to the vessel.
- **Weak Axis** - The weak axis is perpendicular to the vessel.
- **Diagonal** - The strong axis is diagonal to the vessel.



Number of Legs - Enter the number of legs. You must create at least three legs.

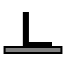
Overall Length of Legs - Enter the distance from the ground to the attachment point of the leg on the vessel.

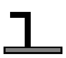
Effective Leg Length "L" - Enter the effective leg length *L*, defined as the free length of the leg that is subject to bending. This is the value that is used in the AISC formula kl/r. This value cannot be explicitly calculated because there are numerous configurations of legs. If the value is zero, the software conservatively uses the overall leg length as the effective leg length in the calculation.

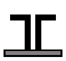


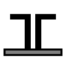
Leg Database - Select the structural specification database to use for leg cross-sections.

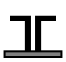
Section Identifier - Enter a section name in the format of the specification selected in **Leg Database**, or click **LookUp** to open the **Select a Leg Shape** dialog box and select a section from the database.

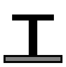
 L - Equal angle

 L - Unequal angle

 B/D - Double angles with large or small sides back to back

 C/HP/M/MC/S - Channels and other miscellaneous channels

 W - Wide Flanges

 ST - Structural Tees

Leg Yield Stress - Enter the yield stress for the legs. Alternatively, click ... to select a material directly from the **Yield Stress Selection** dialog box.

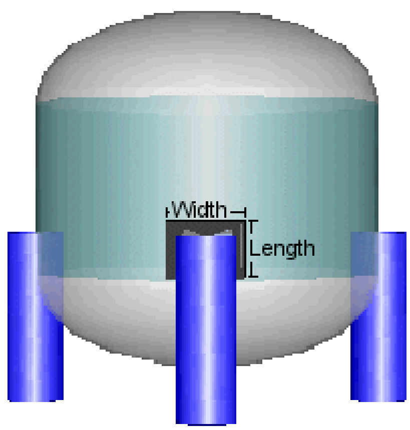
Effective End Condition "K" - Enter the effective end condition *K*. For pressure vessel legs, a value of **1.0** or **1.5** is commonly used.

End Condition	Theoretical K	Recommended K
Fixed - Fixed	0.5	**0.65**
Fixed - Pinned	0.7	**0.80**
Fixed - Trans	1.0	**1.20**
Pinned - Pinned	1.0	**1.00**
Fixed - Rotates	2.0	**2.10**
Pinned - Rotates	2.0	**2.00**

Leg Start Angle - Enter the angle between the designated zero degree line of the vessel and the start angle of the first leg.

Occasional Load Factor - Enter a factor for irregularly occurring loads. The default value is **1.333**.

Perform WRC 107/537 Analysis - Select to perform the WRC 107 local stress analysis on a reinforcing pad between a leg and the vessel. The software calculates the stresses at the edge of the attachment and the edge of the pad.



Pad Width - Enter the pad width as measured along the circumferential direction of the vessel. The pad width must be greater than attachment width.

Pad Length - Enter the length of the pad as measured along the long axis of the vessel.

Pad Thickness - Enter the thickness of the pad.

Compute Centerline Diameter - Click to calculate the value for **Leg Centerline Diameter** based on the element OD, **Number of Legs**, the cross-section selected for **Section Identifier**, and the **Pad Thickness** for **Perform WRC 107 Analysis**.

Are the Legs Cross Braced? - Select if the legs are diagonally braced. Bracing the legs reduces bending and increases the axial load in the legs.

Are these Pipe Legs? - Select to analyze pipe legs. Also enter values for **Pipe Leg Inside Diameter** and **Pipe Leg Outside Diameter**.

Pipe Leg Inside Diameter - Enter the corroded pipe inside diameter. Alternatively, click ... to open the **Seamless Pipe Selection** dialog box, and select a pipe schedule and nominal diameter.

Pipe Leg Outside Diameter - Enter the corroded pipe outside diameter. Alternatively, click ... to open the **Seamless Pipe Selection** dialog box, and select a pipe schedule and nominal diameter.

NOTE This value must be greater than **Pipe Leg Inside Diameter**. The software uses these values to calculate moment of inertia, section modulus and radius of gyration for the legs. These values are used in the AISC unity check and natural frequency calculations.

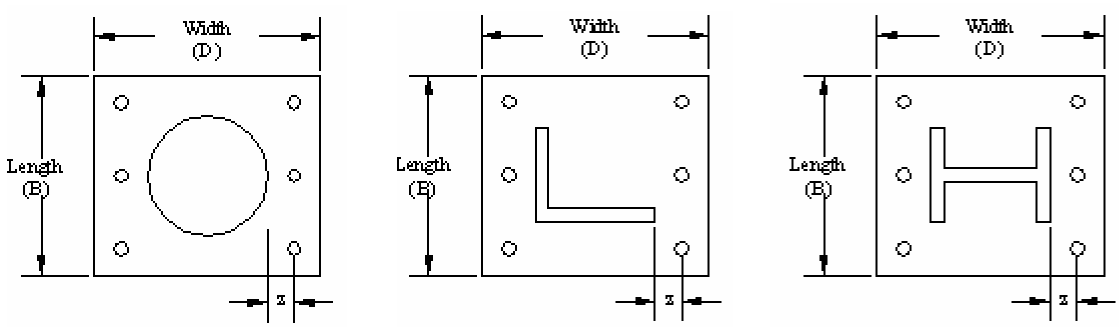
Vessel Translates during Occasional Load? - Select to calculate a more conservative longitudinal moment when **Perform WRC 107 Analysis** is also selected.

Employ Directional Check for W and C Types - Select to indicate you want the software to perform an AISC unity check for the leg angles with W and C types only. When selected, the software calculates additional results for AISC H1-1 and H1-2 on the Leg Check report.

Base Plate Parameters Tab (Leg Dialog Box)

Defines parameters for the leg base plates.

Perform Baseplate Analysis? - Select to place a baseplate on the bottom of each leg and analyze baseplate, bolt, and foundation loads and enter values for the options below. The software assumes the leg is attached symmetrically on the baseplate.



Length, B - Enter the length along the bolt side.

Width, D - Enter the width.

Thickness - Enter the thickness of the baseplate.

Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Thread Series - Enter the thread series identifier:

- **TEMA -** 8 thread series, adapted from the TEMA Standard
- **TEMA Metric**
- **UNC** - Unified National Course Threads, adapted from Mark's Handbook
- **BS 3643**
- **SABS 1700**
- **User Root Area** - Also enter a value for the root area of a single bolt in **Root Area**. This information can be obtained from a standard engineering handbook.

Nominal Diameter - Click **...** and select the nominal bolt diameter from the bolt table selected for **Thread Series**. If you have a bolt that is outside of the bolt table ranges, enter the nominal size and select **User Root Area** in **Thread Series**.

For **TEMA** and **UNC** bolts, the available bolt diameters are:

Bolt Size (inches)	**Bolt Root Area**	
	(cm.^2)	**(in.^2)**
0.500	1.27	0.126
0.625	1.5875	0.202
0.750	1.9050	0.302
0.875	2.22225	0.419
1.000	2.54000	0.551
1.125	2.85750	0.728
1.250	3.17500	0.929
1.375	3.49250	1.155
1.500	3.81000	1.405
1.625	4.12750	1.680
1.750	4.44500	1.980
1.875	4.76250	2.304
2.000	5.08000	2.652
2.250	5.71500	3.423
2.500	6.35000	4.292

Bolt Size	**Bolt Root Area**	
2.750	6.98500	5.259
3.000	7.62000	6.324
3.250	8.25500	7.487
3.500	8.89000	8.749
3.750	9.52500	10.108
4.000	10.1600	11.566

NOTE This information is adapted from Jawad and Farr, *Structural Analysis and Design of Process Equipment*, pg 425.

Corrosion Allowance - Enter the bolt corrosion allowance. The software uses this value to corrode the radius of the root area and calculate a corroded root stress area based on the nominal bolt size and bolt table. This area is then used in the remainder of the bolt load/stress calculations.

The software calculates the required area of the bolt. If the bolt corrosion allowance is greater than zero, the software adds the corrosion allowance and recalculates the diameter based on the new required area:

Corroded Bolt Root Diameter = (4 * New Bolt Area/Pi)$^{1/2}$ - 2 * Bolt Corrosion Allowance

Number per Baseplate - Enter the number of bolts per baseplate. In most cases, this should be an even number, but for angle legs, the value can be **1**.The software assumes that the bolts are located along **Length, B**. The required size of the bolt and the baseplate thickness are looked up from a table in the Pressure Vessel Design Manual and Pressure Vessel Handbook. There are no calculations for one bolt.

Number in Tension - Enter the number of bolts in tension under wind, earthquake, and horizontal loads, defined as the number of bolts along one **Length, B** side (three bolts in the examples above). If these load cases are not needed, no value is required.

Distance from Edge to Bolt - Enter the dimension *z* from the edge of the leg to the centerline of the bolts, measures along the **Width, D** side.

Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Root Area - When **User Root Area** is selected for **Thread Series**, enter the root cross-sectional area of the bolt.

Nominal Compressive Strength - Enter the following for the concrete to which the base is bolted:

- **F'_c** - The nominal ultimate compressive stress of the concrete. This value is F'_c in Jawad and Farr or FPC in Meygesy. A typical entry is 3000 psi.
- **F_c** - The allowable compressive stress of the concrete
- **n** - The steel-to-concrete modulus of elasticity ratio, E_{plates}/E_c.

Average Values of Properties of Concrete Mixes (adapted from Brownell and Young)

Water Content (US Gallons per 94 lb Sack of Cement)	**f'_c 28-day Ultimate Compressive Strength (psi)**	**f_c Allowable Compressive Strength = 0.45*f'c (psi)**	**n Modular Ratio (E_s/E_c)**
7.5	2000	800	15
6.75	2500	1000	12
6	3000	1200	10
5	3750	1400	8

NOTE According to Jawad and Farr, Ec is equal to 57000 multiplied by the square root of f'_c psi. The modulus of elasticity of steel is assumed to be 30×10^6.

Liquid

Home tab: **Details** > **Liquid**

Adds liquid data to the element. Normally, **Liquid** is used on the bottom head of the vessel. Each element is then filled with the appropriate amount of liquid. We recommend that you model the entire vessel before adding liquid data.

Delete - Deletes all data for the liquid.

Full - Click to calculate the values of **Distance from "From" Node** and the value of **Height in this Element**, assuming that the element is completely filled with the defined liquid.

Fill Elements Now - Click to fill the vessel to the level specified by **Height from Datum**. If the height is greater than the vessel height, the vessel is completely filled. The software assigns detail ID values using the **Liquid Density** (see "**Liquid**" on page 115). To use this command effectively, all elements in the vessel model should be created first.

Common Detail Parameters (on page 52)

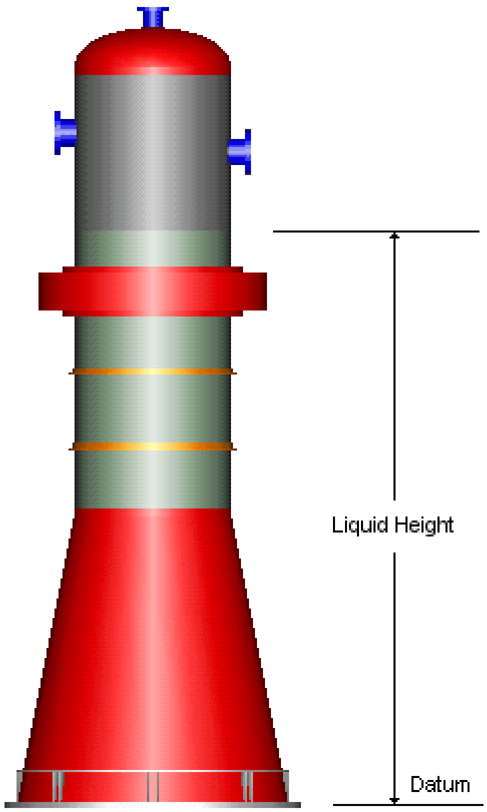
Liquid Density - Enter the density or specific gravity of the liquid. Typical specific gravities and densities are shown below. The densities should be converted if you use another units system.

Name	Specific Gravity	Density (lb/ft3)
Ethane	0.3564	22.23
Propane	0.5077	31.66
N-butane	0.5844	36.44
Iso-butane	0.5631	35.11
N-Pentane	0.6310	39.35
Iso-Pentane	0.6247	38.96
N-hexane	0.6640	41.41
2-methypentane	0.6579	41.03
3-methylpentane	0.6689	41.71
2,2-dimethylbutane	0.6540	40.78
2,3-dimethylbutane	0.6664	41.56
N-heptane	0.6882	42.92
2-methylheptane	0.6830	42.59
3-methylheptane	0.6917	43.13
2,2-dimethylpentane	0.6782	42.29
2,4-dimethylpentane	0.6773	42.24
1,1-dimethylcyclopentane	0.7592	47.34
N-octane	0.7068	44.08
Cyclopentane	0.7504	46.79
Methylcyclopentane	0.7536	46.99
Cyclohexane	0.7834	48.85
Methylcyclohexane	0.7740	48.27
Benzene	0.8844	55.15

Toluene	0.8718	54.37
Alcohol	0.7900	49.26
Ammonia	0.8900	55.50
Benzine	0.6900	43.03
Gasoline	0.7000	43.65
Kerosene	0.8000	49.89
Mineral Oil	0.9200	57.37
Petroleum Oil	0.8200	51.14
Water	1.0	62.4

Height in this Element - Enter the height or length of the liquid on this element. This value is used only to calculate the weight of the liquid section. For seismic calculations, the weight center of the liquid section is taken at half this height. The value is also used to calculate the operating pressure at all points below the liquid.

Height from Datum - Enter the height or length of the liquid from the datum line to the operating or design level. Normally, **Liquid** is used on the bottom head of the vessel. Each element is then filled with the appropriate amount of liquid. It is best to model the entire vessel before adding liquid data. If a value is entered that is greater than the height of the vessel, the software completely fills the vessel.



Insulation

Home tab: **Details** > **Insulation**

Adds insulation to the element.

Delete - Deletes all insulation data.

Full - Click to calculate the values of Distance from "From" Node and the value of Height/Length of Insulation, assuming that the element is fully covered by the defined insulation.

All - Click to cover the entire vessel with insulation. The software assigns detail ID values using the defined insulation values. To use this command effectively, all elements in the vessel model should be created first.

Common Detail Parameters (on page 52)

Height/Length of Insulation - Enter the height or length of the insulation on the element. This value is used only to calculate the weight of the insulation. For seismic calculations, the weight center of the insulated section is taken at half this height. If you have insulation on a horizontal vessel, the value is the length of the insulated section.

NOTE The only software distinction between insulation and lining is that insulation is on the OD of the element, while lining is on the ID of the element. Use **Insulation** to add OD fireproofing, and **Lining** to add ID fireproofing.

Thickness of Insulation - Enter the thickness of the insulation or fireproofing.

Density of Insulation - Enter the density of the insulation, such as the following typical densities:

Material Type	**Density (lbs/ft^3)**
Calcium Silicate	22.5
Foam Glass	16.0
Mineral Wool	14.0
Glass Fiber	11.0
Asbestos	30.0
Careytemp	18.0
Kaylo 10	22.0
Perlite / Celo-temp 1500	23.0
Polyurethane	4.0
Styrofoam	3.0

Type of Insulation - Enter a description for the type of insulation.

Lining

Home tab: **Details** > **Lining**

Adds lining to the element.

Delete - Deletes all lining data.

Full - Click to calculate the values of **Distance from "From" Node** and the value of **Height/Length of Lining**, assuming that the element is fully covered by the defined lining.

Common Detail Parameters (on page 52)

Height/Length of Lining - Enter the height or length of the lining on this element. This value is used only to calculate the weight of the lined section. For seismic calculations the weight center of the lined section will be taken at half this height. If you have a lining in a horizontal vessel, the value is the length of the lined section.

NOTE The only software distinction between insulation and lining is that insulation is on the OD of the element, while lining is on the ID of the element. Use **Insulation** to add OD fireproofing, and **Lining** to add ID fireproofing.

Thickness of Lining - Enter the thickness of the lining or fireproofing.

Density of Lining - Enter the density of the insulation, lining, or packing, such as the following typical lining densities:

Material Type	**Density (lbs/ft^3)**
Alumina Brick	170.0
Fire Clay	130.0
High Alumina	130.0
Kaolin	135.0
Magnesite	180.0
Silica	110.0
Insulating Fire Brick	40.0
Concrete	140.0
Cement	100.0

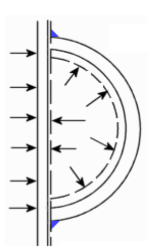
Also see **Density of Insulation** (see "**Insulation**" on page 118).

Halfpipe Jacket

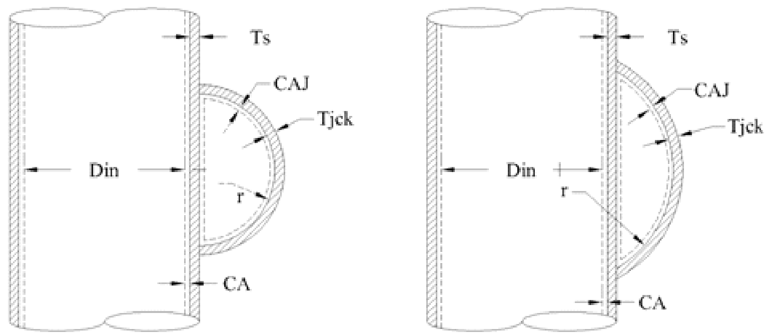
Home tab: **Details** > **Halfpipe Jacket**

Adds half-pipe jackets to the shell on the selected cylinder element, according to ASME Code, Section VIII, Division 1, Appendix EE.

Half pipe jackets are commonly made by rolling and forming flat bar in a specific radius and helical curvature that matches that of the parent shell course. These jackets are used to heat or cool the contents inside of the vessel. For vessels that are under cyclic service, it is advised that the jacket be attached by both a fillet and full penetration groove weld.



PV Elite performs required thickness and maximum allowable working pressure (Mawp) calculations for cylindrical shells with half-pipe jackets attached. The analysis is based on the ASME Boiler and Pressure Vessel Code, Section VIII, Division 1, Paragraph EE-1, Appendix EE. The analysis is only valid for the cylindrical geometries shown in Figure EE-4.



Additionally, only nominal pipe sizes from 2 to 4 can be used. Although there are no charts for sizes 2.5 and 3.5, the software accepts these sizes and performs iterations between the given charts. If the half-pipe is a nonstandard pipe size, or has a formed radius, the actual radius is used in the calculations.

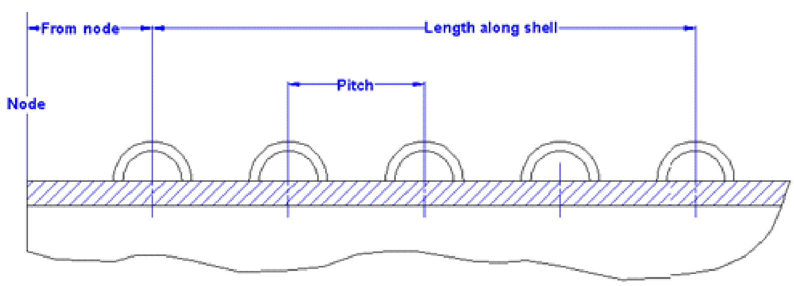
The software takes full account of corrosion allowance. Actual thickness values and corrosion allowances are entered, and the software adjusts thicknesses and diameters when making calculations for the corroded condition.

Delete - Deletes all jacket data.

 **Quick Results** - Click to see a quick report of half pipe jacket analysis results.

Common Detail Parameters (on page 52)

Length Along Shell of Jacket Section - Enter the distance that the jacket extends along the length of the shell section. This value cannot be greater than the specified length of the shell course.



Pitch Spacing - Enter the distance between centers of adjacent half-pipes.

Shell Corrosion Allowance in Jacket - Enter the internal corrosion allowance of the half-pipe jacket.

Start Angle - Enter the start angle.

Jacket Design Temperature - Enter the design temperature of the jacket.

Jacket Design Pressure - Enter the design pressure of the fluid inside the jacket.

Jacket Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Pipe - Click to open the **Seamless Pipe Selection** dialog box and select values from the piping database for **Jacket Corrosion Allowance** and **Minimum Jacket Thickness**.

Jacket Corrosion Allowance - Displays the corrosion allowance for the pipe that you selected in **Pipe**. You can also manually enter a value.

Minimum Jacket Thickness - Displays the thickness for the pipe that you selected in **Pipe**. You can also manually enter a value.

Inside Radius of Formed Jacket - Enter a value for a jacket formed from bar or plate. This value is only used if **Formed Radius Type** is selected for **Nominal Pipe Size**.

Nominal Pipe Size - Select the nominal pipe size of the jacket. Select **2 inch**, **3 inch**, or **4 inch** to use sizes recognized by Appendix EE. Select **2.5 inch** or **3.5 inch** to interpolate from Appendix EE graphs. Select **Formed Radius Type** for a non-standard jacket, and also enter a value for **Inside Radius of Formed Jacket**.

Contents Specific Gravity - Enter the specific gravity of any fluid contained within the jacket.

Enter the density or specific gravity of the liquid. Typical specific gravities and densities are shown below. The densities should be converted if you use another units system.

Name	**Specific Gravity**	**Density (lb/ft3)**
Ethane	0.3564	22.23
Propane	0.5077	31.66
N-butane	0.5844	36.44
Iso-butane	0.5631	35.11
N-Pentane	0.6310	39.35
Iso-Pentane	0.6247	38.96
N-hexane	0.6640	41.41
2-methypentane	0.6579	41.03
3-methylpentane	0.6689	41.71
2,2-dimethylbutane	0.6540	40.78
2,3-dimethylbutane	0.6664	41.56
N-heptane	0.6882	42.92

2-methylheptane	0.6830	42.59
3-methylheptane	0.6917	43.13
2,2-dimethylpentane	0.6782	42.29
2,4-dimethylpentane	0.6773	42.24
1,1-dimethylcyclopentane	0.7592	47.34
N-octane	0.7068	44.08
Cyclopentane	0.7504	46.79
Methylcyclopentane	0.7536	46.99
Cyclohexane	0.7834	48.85
Methylcyclohexane	0.7740	48.27
Benzene	0.8844	55.15
Toluene	0.8718	54.37
Alcohol	0.7900	49.26
Ammonia	0.8900	55.50
Benzine	0.6900	43.03
Gasoline	0.7000	43.65
Kerosene	0.8000	49.89
Mineral Oil	0.9200	57.37
Petroleum Oil	0.8200	51.14
Water	1.0	62.4

Tubesheet

Home tab: **Details** > **Tubesheet**

Adds a heat exchange tubesheet to the element. To build and analyze shell and tube heat exchangers with PV Elite, you must understand the modeling techniques involved:

- First, define and design the pressure envelope or exterior components of the exchanger first, including all covers, flanges and cylinders. Verify that the design pressures and temperatures are correctly specified.
- Tubesheets are frequently paired to cylinders or flanges. Click on the front end flange element just to the left of the tubesheet for proper definition, then click **Tubesheet Analysis** to define the assembly. Enter data on each tab of the **Heat Exchanger Tubesheet Input** dialog box. The software dynamically adds or removes tabs depending on the type of exchanger selected. For example, an exchanger with a floating head requires more input and displays more tabs than a U-tube type exchanger.
- Tubesheets that are either completely integral or integral on the channel side are attached to the channel cylinder. These tubesheets must have a value for **Distance from "From" Node** to define the length of that cylinder because they start at the end of the channel cylinder.
- Verify that the 3D graphic looks like the needed geometry. Use the transparency feature to see inside of the exchanger. The software does not draw all tubes to improve performance.

NOTE Examples are located in the PV Elite installation folder.

Delete - Deletes all tubesheet data.

HTRI In - Select to open a .dbo HTRI output file.

See also

Tubesheet Type and Design Code Tab (Heat Exchanger Tubesheet Input Dialog Box) (on page 125)
Tubesheet Properties Tab (Heat Exchanger Tubesheet Input Dialog Box) (on page 129)
Tube Data Tab (Heat Exchanger Tubesheet Input Dialog Box) (on page 137)
Expansion Joint Data Tab (Heat Exchanger Tubesheet Input Dialog Box) (on page 150)
Load Cases Tab (Heat Exchanger Tubesheet Input Dialog Box) (on page 155)
Floating TubeSheet Tab (Heat Exchanger Tubesheet Input Dialog Box) (on page 159)
Spherical Cover/Backing Ring Tab (Heat Exchanger Tubesheet Input Dialog Box) (on page 162)

Tubesheet Type and Design Code Tab (Heat Exchanger Tubesheet Input Dialog Box)

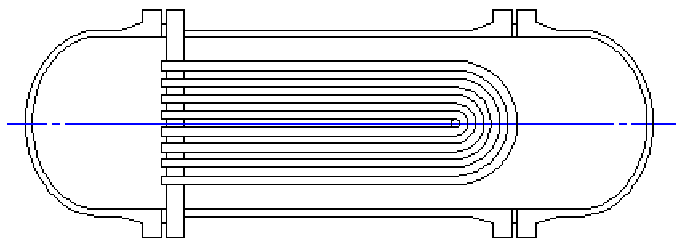
General Exchanger Data

Tubesheet Analysis Method - Displays the standard to use for analysis:

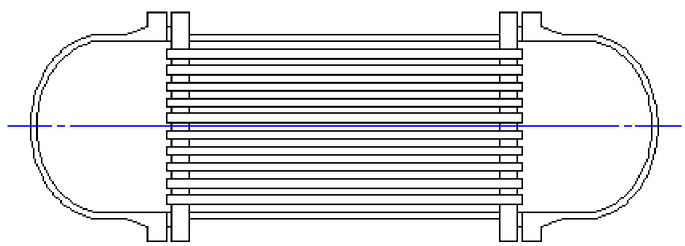
- **TEMA** - Tubular Exchanger Manufacturers Association
- **ASME UHX** - ASME Section VIII, Division 1, Section UHX
- **PD 5500 (British Code)** - British Pressure Vessel Code, Section 3.9, Flat Heat Exchanger Tubesheets
- **EN-13445**

Exchanger Type - Select the type of heat exchanger.

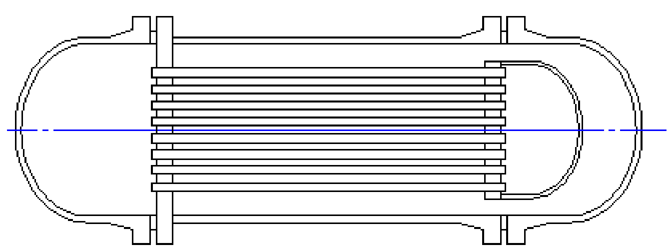
- **U-Tube** has only one tubesheet. The tubes are bent in the form of a "U." The bundle can removed for maintenance, but the inside of the tube is harder to clean because of the bend.



- **Fixed** has two tubesheets that are fixed at each end of the exchanger and are connected to each other via straight tubes. Both the tubesheets are stationary, so differential thermal expansion can develop between the shell and the tubes. An expansion joint is sometimes required to absorb the thermal growth.



- **Floating** has one tubesheet that is fixed (stationary) and one that is free to move. Because one tubesheet "floats," any differential thermal expansion between the shell and tubes is absorbed. This category of exchangers is the most versatile and also the costliest. Tubes can also be cleaned easily compared to U-tube exchangers.



Expansion Joint Type (if any) - Select the expansion joint type.

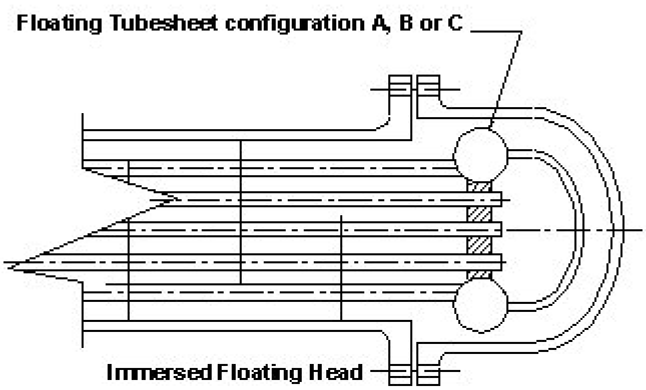
- **No Joint** - The exchanger type has no expansion joint type.
- **Thin Bellows Type** - The joint is comprised of a thin bellows that is very flexible and has low stiffness. Analysis is performed according to ASME Appendix 26 for thin bellows expansion joint type.
- **Thick Joint Type (Flanged and Flued)** - The joint is comprised of a number of shell elements that are added together to form the expansion joint. A thick joint is stiffer than the thin joint and its stiffness must be taken into account. Analysis is performed according to Appendix 5 guidelines, TEMA or Kopp & Sayre Method.

NOTE Click **Vacuum Pressures and Report Options for this Load Case** to open the **Report Print Options** dialog box, then select **Print Intermediate Results for Expansion Joint Calculations**.

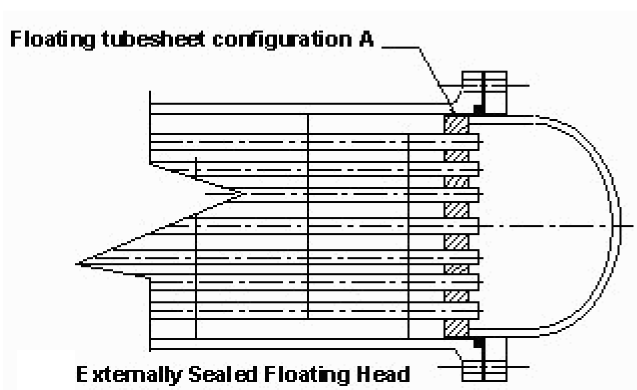
ASME or EN-13445

Floating Exchanger Type - Select the type of floating exchanger, as defined in part UHX of the ASME code:

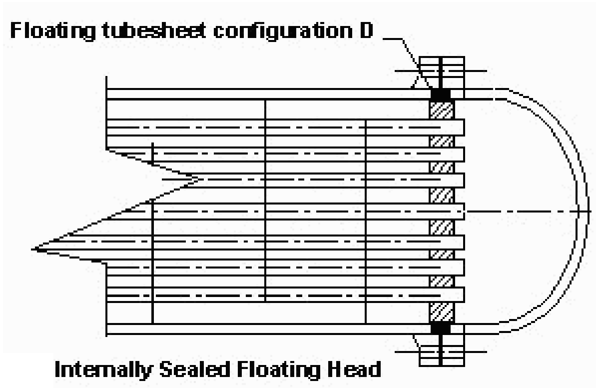
- **Exchanger with an Immersed Floating Head**



- **Exchanger with an Externally Sealed Floating Head**



- **Exchanger with an Internally Sealed Floating Head**

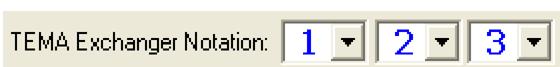


Tubesheet/Shell Junction Stress Reduction Option - Select the option used to reduce the possible over stress at the junction of the tubesheet and the integral cylinder. If the U-tube tubesheet is welded to the shell then the integral cylinder is the shell cylinder and if the tubesheet is welded to the channel then it is the channel cylinder. The U-tube stress reduction options are displayed below:

- **Increase Tubesheet Thickness**
- **Increase Integral Cylinder Thickness** - Increase for the shell, or channel, or both.
- **Increase Cylinder and Tubesheet Thickness** - Increase for both the tubesheet and the integral cylinder.
- **Perform Elastic-Plastic Calculation** - Performing the elastic-plastic calculation at that junction is recommended if all prerequisites are satisfied. For example, the equipment is not operating in the temperature creep range where time-dependent properties affect material allowables.

TEMA

TEMA Exchanger Notation - For a TEMA analysis, select a value for each of the three options. The order of the options is:



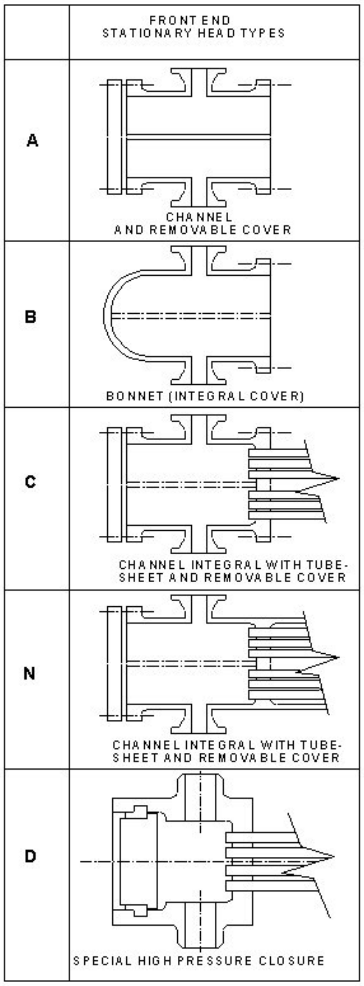
1 - Front End

2 - Shell Type

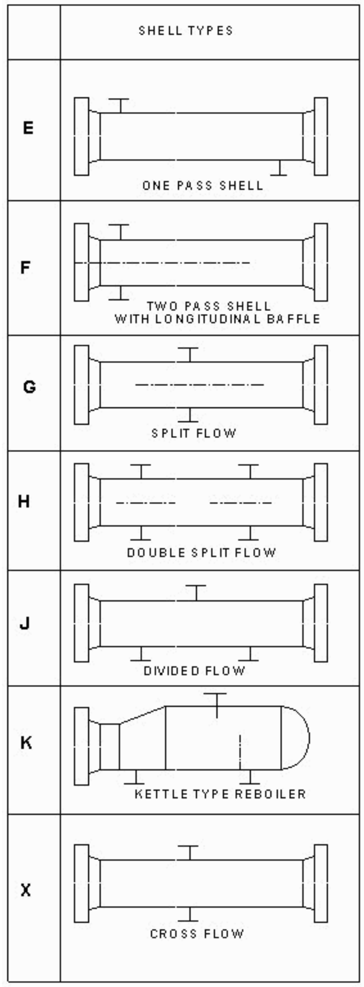
3 - Rear End

The nomenclature for each option is according to the conventions in TEMA page 2, figure N-1.2:

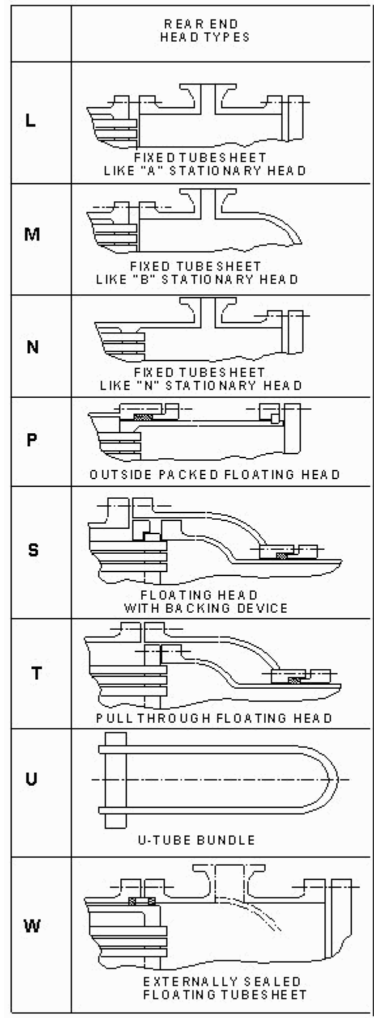
Front End



Shell Type



Rear End



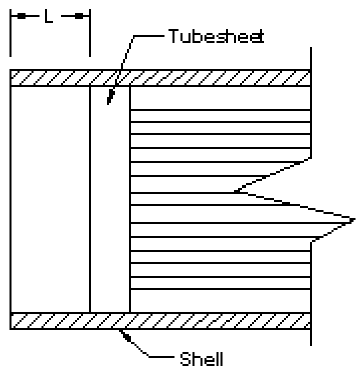
TEMA Exchanger Class - For a TEMA analysis, select one of the classes of exchangers, **R**, **C**, or **B**. The class is determined by the severity of service and is discussed in the TEMA code, paragraph RCB-1.1. The calculation method is the same for all classes, but each class has its own design restrictions. Verify the correct class to use for your application.

Tubesheet Properties Tab (Heat Exchanger Tubesheet Input Dialog Box)

Description - Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

Element From Node - Displays the **From Node** for the selected element.

Dist. from "From" Node - Enter the distance from the shell to the outer face of the nearer tubesheet.



Tubesheet Type - Select the type of tubesheet to analyze, according to ASME classifications. The available selections change depending on the **Exchanger Type** selected on the **Tubesheet Type and Design Code** tab.

- **Fixed** tubesheet exchangers are subject to loads arising from differential thermal expansion between the tubes and the shell. They have stationary tubesheets on both sides. Fixed tubesheet exchangers are classified by ASME UHX:

Configurations	Description
A	Tubesheet integral with both shell and channel
B	Tubesheet integral with shell, gasketed with channel, with tubesheet extended as a flange
C	Tubesheet integral with shell, gasketed with channel, with tubesheet not extended as a flange
D	Tubesheet gasketed with both shell and channel

-

- **U-Tube** exchangers are classified either as integral with the shell, channel, both, or gasketed on both sides, according to ASME UHX

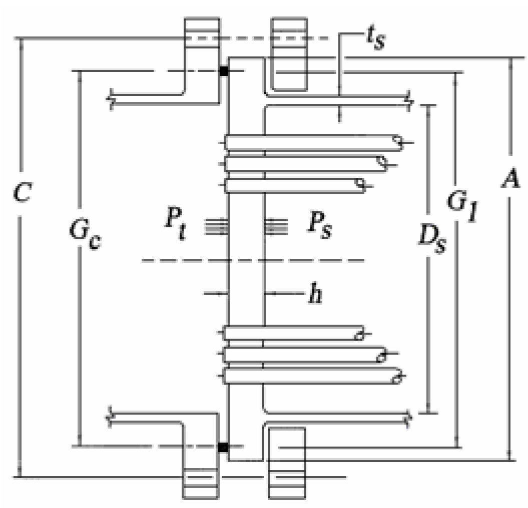
Configurations	Description
A	Tubesheet integral with both shell and channel
B	Tubesheet integral with shell, gasketed with channel, with tubesheet extended as a flange
C	Tubesheet integral with shell, gasketed with channel, with tubesheet not extended as a flange
D	Tubesheet gasketed with both shell and channel
E	Tubesheet integral with channel, gasketed with shell, with tubesheet extended as a flange
F	Tubesheet integral with channel, gasketed with shell, with tubesheet not extended as a flange

- **Floating** tubesheet exchangers have a stationary tubesheet and a floating tubesheet, according to ASME UHX classifications:

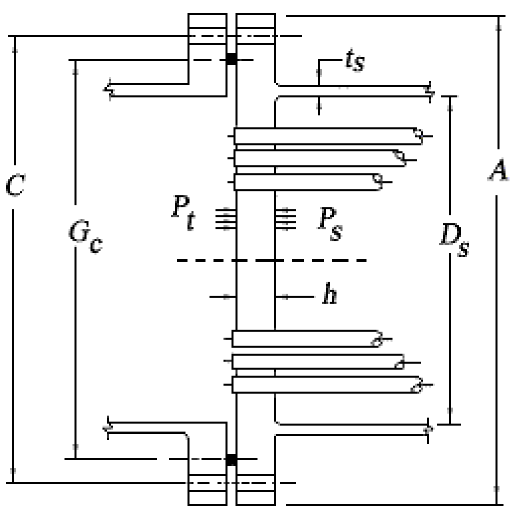
Configurations	Description
A	Tubesheet integral
B	Tubesheet gasketed and extended as a flange
C	Tubesheet gasketed and not extended as a flange
D	Tubesheet internally sealed

Some ASME UHX Tubesheet configurations are shown below:

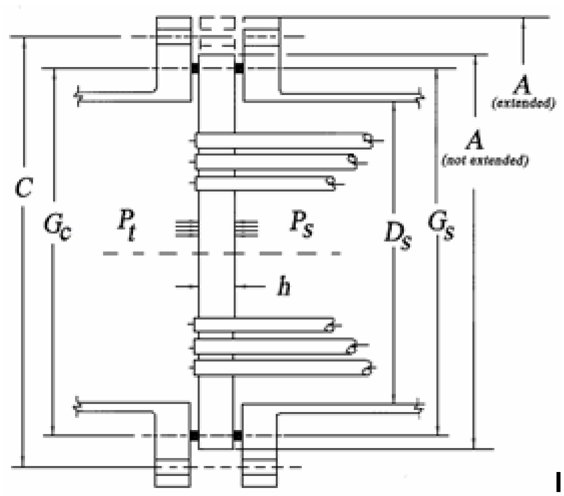
- Tubesheet is integral with the shell and is gasketed on the channel side and is not extending as a flange



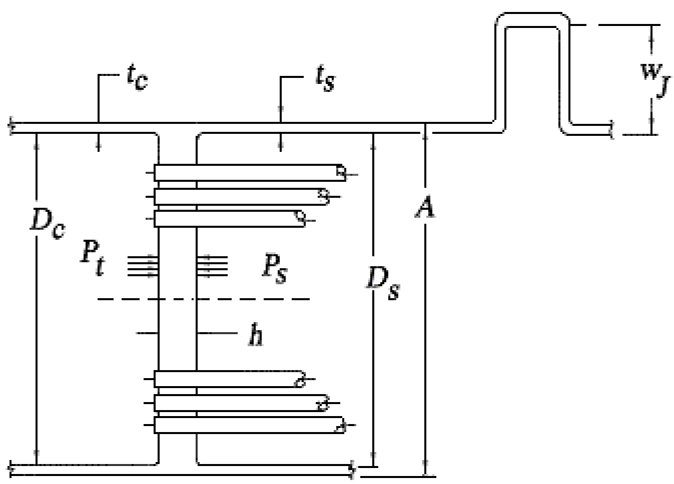
- Tubesheet is integral with the shell and is gasketed on the channel side and is extending as a flange



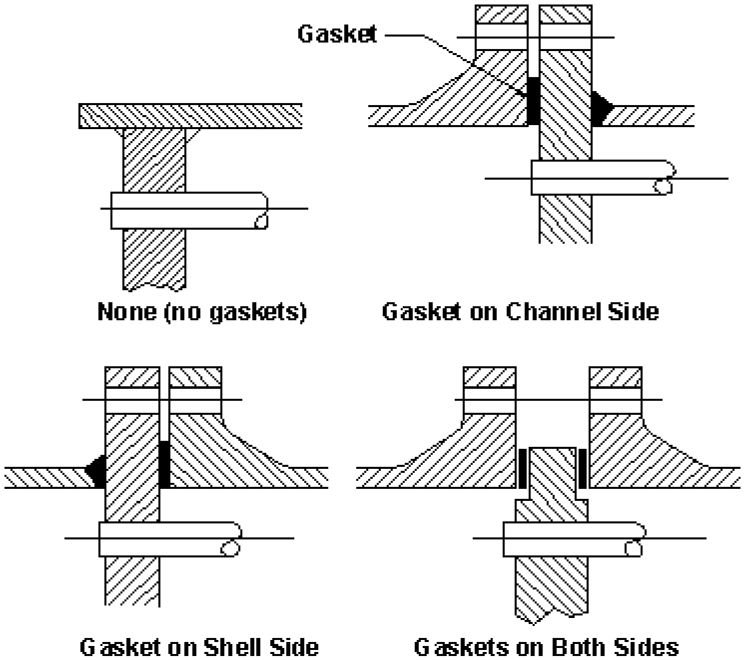
- Tubesheet is gasketed on both the shell and the channel sides and is not extended as a flange. In an alternative arrangement the tubesheet is extending as a flange.



- Tubesheet is integral with both the shell and the channel. This is a fixed tubesheet exchanger, as a flanged and flued expansion joint is used to reduce the differential thermal expansion, between the tubes and the shell.



The following tubesheet attachment types are used:

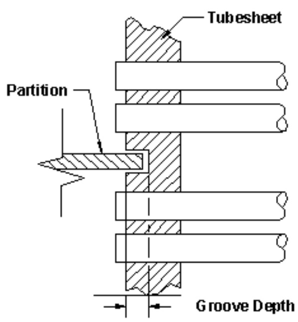


Outside Diameter - Enter the outside diameter of the tubesheet.

Tubesheet Thickness - Enter the tubesheet thickness in an uncorroded condition. If it is a re-rate, then the actual measured thickness is typically used.

Corr. Allow. Shell Side / Channel Side - Enter the corrosion allowance on the shell side (the inner face of the tubesheet), and the corrosion allowance on the channel side (the outer face of the tubesheet facing the channel side).

Depth of Groove in Tubesheet (if any) - Enter the depth of a groove in the tubesheet, used to locate the channel partition plate and its gasket. If there is no groove, such as in a single pass exchanger, this value is **0**.



Weld Leg at Back of Tubesheet (if any) - If the tubesheet is welded to shell and/or channel, then enter the fillet weld length at the back of the tubesheet.

Tubesheet Assembly is Down/Left? - Select if a horizontal U-tube exchanger is modeled with the tube bundle facing left and the tubesheet on the right or for a vertical U-tube exchanger.

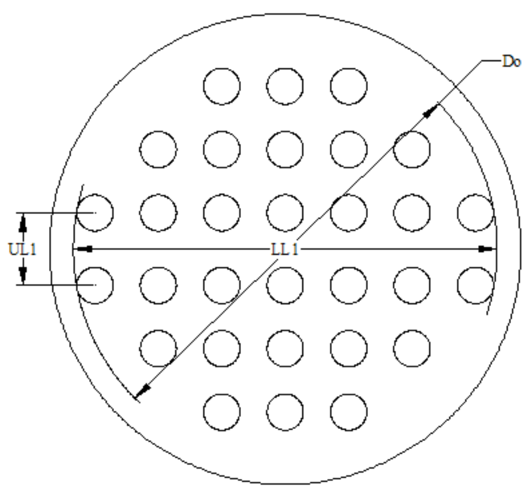
Tubesheet Extended as Flange? - Select if the tubesheet is extended as a flange, so that it is subject to the bolt load from the mating flange.

Thickness of Extended Portion - When **Tubesheet Extended as Flange?** is selected, enter the thickness of the portion of the tubesheet that is extended for bolting.

Tfr/T Ratio for U-Tubesheets (optional) - When **Tubesheet Extended as Flange?** is selected, enter the ratio of the required thickness of the tubesheet flanged extension to the tubesheet, if reducing the required thickness of the flanged extension is required. The ratio should be less than **1.0** and more than **0.2**. The default value is **1.0**. This value is used in TEMA RCB 7.1342 for U-tube tubesheet exchangers. This entry is optional.

Bolt Load Transferred to Tubesheet? - When **Tubesheet Extended as Flange?** is selected, also select this option if the bolt load is transferred to the tubesheet, extended as the flange. Do not select this option if the tubesheet is gasketed with both the shell and channel flanges. Otherwise, the tubesheet can still be extended, but the bolt load is not transferred to the tubesheet extension. Carefully consider all possible cases, such as the hydrotest. When this option is not selected, the required thickness of tubesheet extension is not calculated.

UnTubed Lane Area - Enter the total area of all the untubed lanes on the tubesheet. If there is no pass partition lane then the value is **0**. This value is only needed for ASME code analysis. In a single pass exchanger, this area is UL1 * Do:



The area is UL1 * Do

The maximum limiting value of is 4*Do*p.

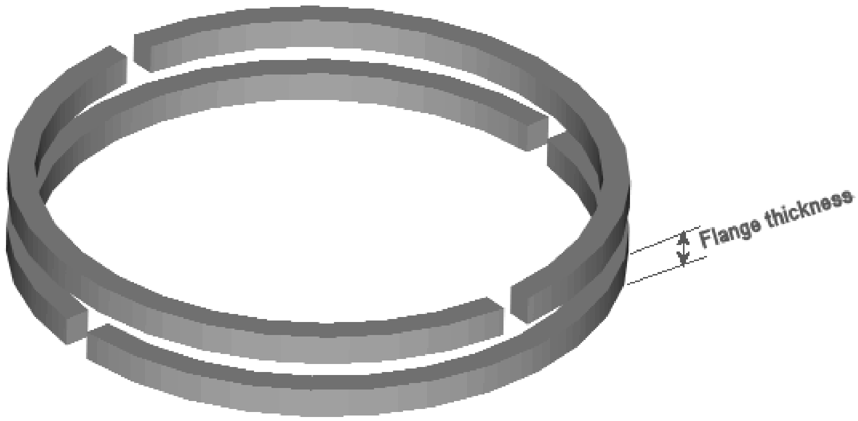
Do - Equivalent diameter of outer limit circle

p - Tube pitch

UL1 - Distance between innermost tube hole centers (width of pass partition lane)

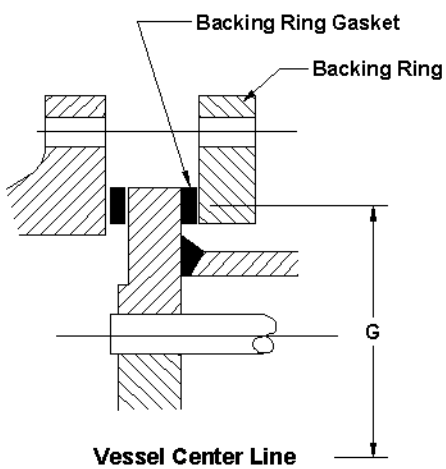
Backing Ring

Backing Ring Thickness - Enter the actual thickness of the backing ring. This value is needed when a tubesheet is clamped and gasketed on one side by a backing ring or device. For doubly-split rings, this is the thickness of each piece:



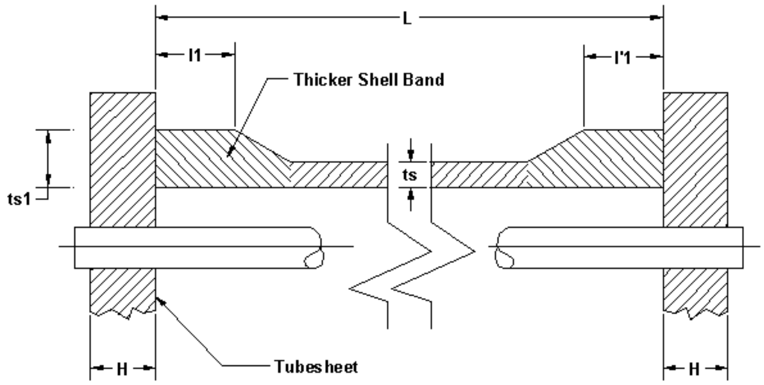
Backing Ring ID / OD - Enter the inside and the outside diameters of the backing ring.

G Dimension for Backing Ring - Enter the effective diameter *G*, defined as the mid-point of the contact between the backing flange and the tubesheet. When a tubesheet is clamped and gasketed on one side by a backing ring or device, the effective diameter of the gasket between the backing ring and the tubesheet is needed.



ASME Part UHX / EN-13445 Shell Band Data

Is There a Shell Band? - Select if there is a shell band. The shell might have thicker courses at either end, called shell bands. Shell bands give added strength to the shell to tubesheet region:



Shell Thickness Adjacent to Tubesheet - Enter the thickness of the shell bands *ts1*.

Shell Band Corrosion Allowance - Enter the corrosion allowance for the shell band.

Shell Band Length Adjacent to Tubesheet, Front End L1 - Enter the front end length *l1* for the shell band.

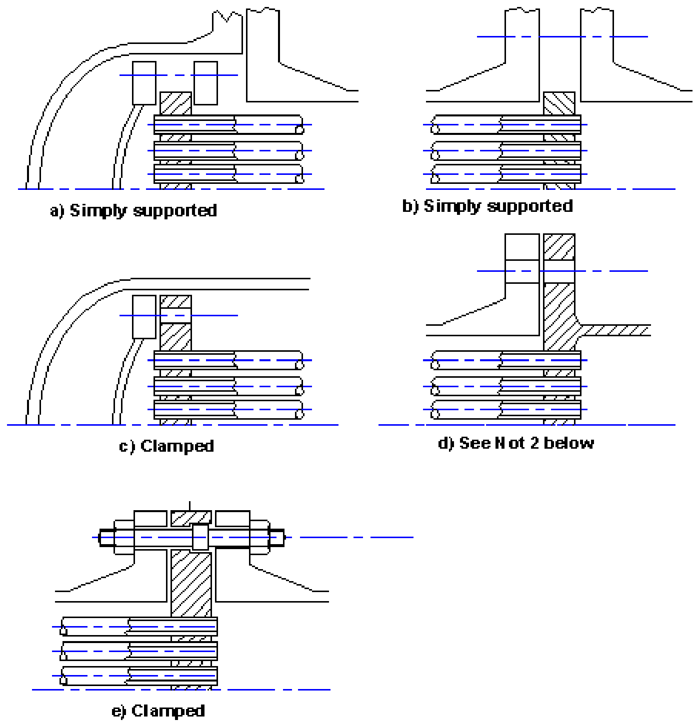
Shell Band Length Adjacent to Tubesheet, Rear End L1' - Enter the rear end length *l1'* for the shell band.

NOTE Also specify the shell band material on the **Load Cases** tab.

PD 5500

How are Tubesheets Clamped - Select the method of clamping for a fixed or a floating tubesheet, as defined by PD 5500, Figure 3.9-6:

Stationary Tubesheet	**Floating Tubesheet**
Simply Supported	Simply Supported
Simply Supported	Clamped
Clamped	Simply Supported
Clamped	Clamped



If either shell or channel is welded to a U-tubesheet, then:

- The U-tubesheet is simply supported if:

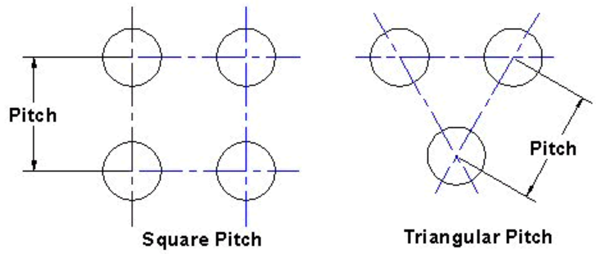
$$\frac{4.3 \cdot D_{star} + 0.65 \cdot D_1 \cdot K_\theta}{1.3 \cdot D_{star} + 0.5 \cdot D_1 \cdot K_\theta} > 2.3$$

- The U-tubesheet is clamped if:

$$\frac{4.3 \cdot D_{star} + 0.65 \cdot D_1 \cdot K_\theta}{1.3 \cdot D_{star} + 0.5 \cdot D_1 \cdot K_\theta} < 2.3$$

Tube Data Tab (Heat Exchanger Tubesheet Input Dialog Box)

Number of Holes / Pattern - Enter the total number of tube holes drilled in one of the tubesheets. Also select the hole pattern: **Square**, or **Triangular**. The code expects the holes to be fairly evenly spaced over the entire area of the tubesheet without large areas that are not drilled.



NOTE For a square rotated pattern, select **Square**. For a triangular rotated pattern, select **Triangular**.

Wall Thickness / Corrosion Allowance - Enter the wall thickness of the exchanger tubes, and the corrosion allowance to which the tubes are subjected. Typical tube thicknesses are below:

Tube O.D. inches (mm)	B.W.G. gage	Thickness inches	Thickness mm
1/4 (6.35)	22	0.028	0.711
	24	0.022	0.559
	26	0.018	0.457
	27	0.016	0.406
3/8 (9.53)	18	0.049	1.245
	20	0.035	0.889
	22	0.028	0.711
	24	0.022	0.559
1/2 (12.7)	16	0.065	1.651
	18	0.049	1.245
	20	0.035	0.889
	22	0.028	0.711
5/8 (15.88)	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651
	17	0.058	1.473
	18	0.049	1.245
	19	0.042	1.067
	20	0.035	0.889
3/4 (19.05)	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769

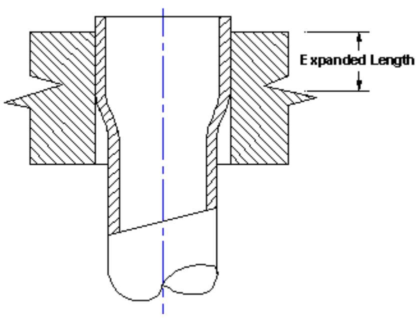
Tube O.D.	B.W.G.	Thickness	Thickness
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651
	17	0.058	1.473
	18	0.049	1.245
	20	0.035	0.889
7/8 (22.23)	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651
	17	0.058	1.473
	18	0.049	1.245
	20	0.035	0.889
1 (25.4)	8	0.165	4.191
	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651
	18	0.049	1.245

Tube O.D.	B.W.G.	Thickness	Thickness
	20	0.035	0.889
1-1/4 (31.75)	7	0.180	4.572
	8	0.165	4.191
	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	16	0.065	1.651
	18	0.049	1.245
	20	0.035	0.889
1-1/2 (38.10)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108
	16	0.065	1.651
2 (50.8)	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
2-1/2 (63.5)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108
3 (76.2)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108

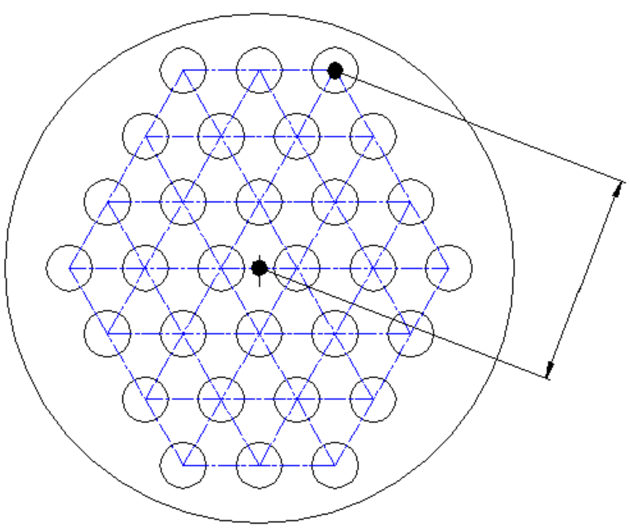
Tube O.D.	B.W.G.	Thickness	Thickness
2-1/2 (63.5)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108
3 (76.2)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108

Outside Diameter / Pitch - Enter the outside diameter and the pitch of the tubes. The tube pitch is the distance between the centers of the adjacent tubes.

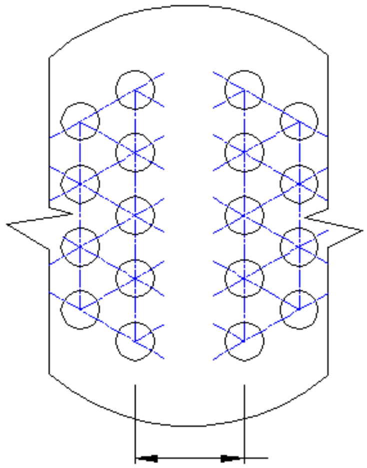
Length of Expanded Portion of Tube - Enter the length of tube that is expanded into the tubesheet hole. This value may not exceed the full thickness of the tubesheet to avoid failure of the tube at the inner tubesheet face and is usually 80% to 90% of the tubesheet thickness.



Radius to Outermost Tube Hole Center - Enter the distance from the center of the tubesheet to the centerline of the tube furthest away.

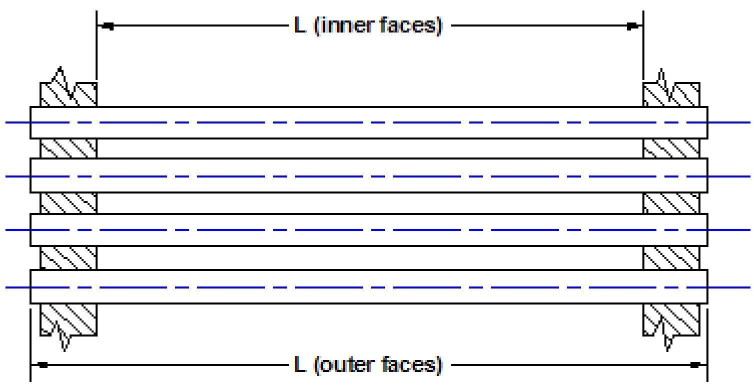


Distance Between Innermost Tube Centers - Enter the maximum distance between the tube innermost centers when a partition plate is installed, because the innermost lanes of tubes may be further apart than the general tube pitch in the remainder of the tubesheet. This is the maximum distance between the tube innermost centers. If there is no partition plate, this value is **0**.

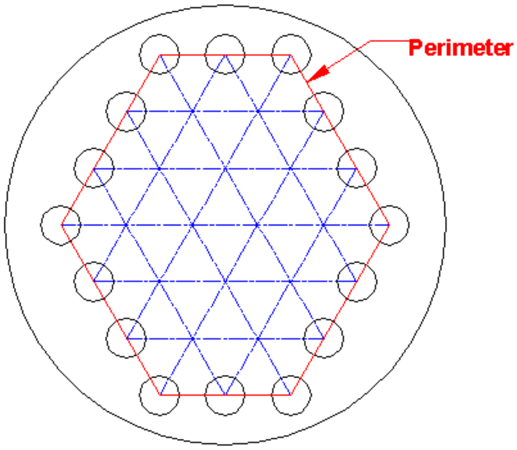


Straight Tube Length - Enter the straight length of the tubes, based on the selection for **Straight Tube Length Measured Between**.

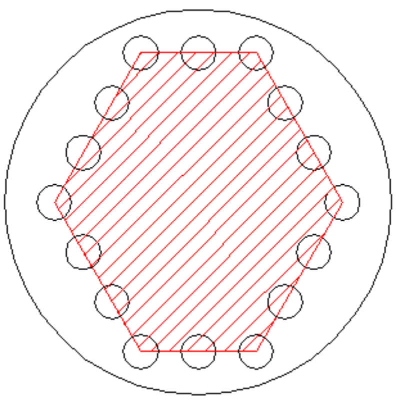
Straight Tube Length Measured Between - Select the method of tube length measurement. Select **Inner Faces** or **Outer Faces**.



Perimeter of Tube Layout (if needed) - Enter the total linear length of the outermost tubes. This value is only required if the software is calculating the punching shear stress. Otherwise, the value is **0**.



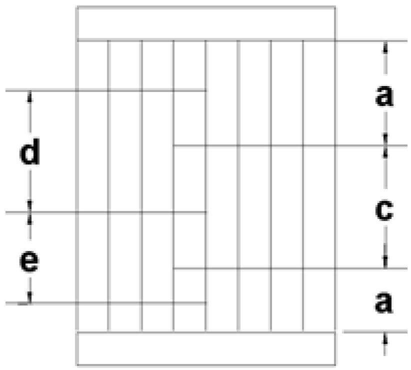
Area of Tube Layout (if needed) - Enter the area defined by the tube layout, including all the tubes in the tube bundle.



Tube Layout Assistant - Opens the **Tube Layout Assistant** (on page 148) utility. This utility helps you determine the number of tubes that will fit in a layout configuration for a heat exchanger tubesheet.

Import Layout Results - Imports results saved in the **Tube Layout Assistant**, and fills in values for basic tube data from the results.

Max. Dist. from Tubesheet to 1st Tube Support - Enter the maximum distance between two tube supports. Supports, often called baffles, are along the heat exchanger, and are in addition to the tube support provided by each of the tubesheets. The software uses the maximum unsupported length to determine the buckling stress in the tubes. Carefully examine the design of the exchanger, and enter the maximum possible unsupported length. For the example below, *a* is the distance between a baffle and a tubesheet, while *c*, *d*, and *e* are the distances between baffles.



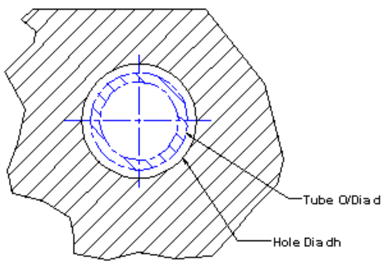
Max. Dist. bet. 2 Tube Supports - Enter the maximum distance from the tubesheet to the first tube support (baffle). This is the maximum of the *a* distances shown above.

End Condition k - Enter the tube end condition value *k*, as defined in the TEMA, ASME, and PD 5500 codes. Each code uses different values, so verify that you use a value for the code defined for your model.

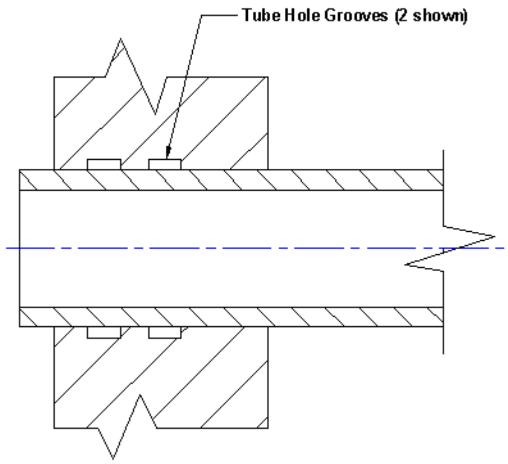
Condition	'k' Value	
	TEMA / ASME	PD 5500
Between two tubesheets	0.60	0.50
Between tubesheet and baffle	0.80	0.707
Between two baffles	1.00	1.00

Max Unsupported Len SL - Enter *l* x *k* as the effective buckling length of the tubes between supports.

Tube Hole Diameter, dh - Enter the diameter of the holes drilled in the tubesheet. The tube hole diameter should be slightly larger than the outside diameter of the tube. This provides a clearance that is closed as the tube is expanded in the hole.

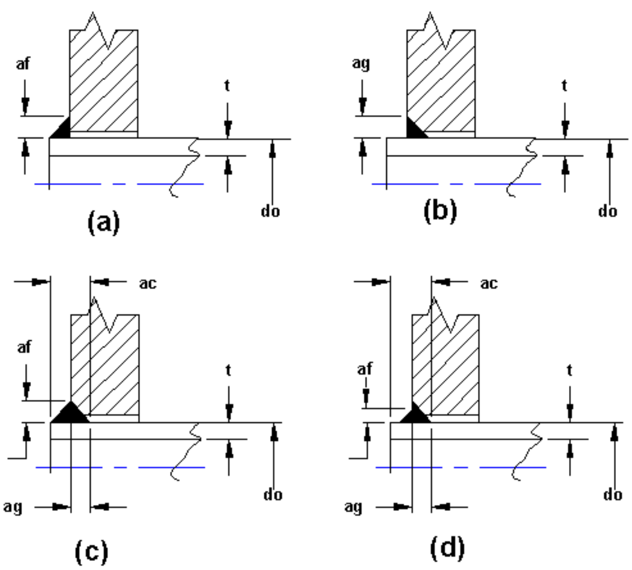


Number of Grooves in Hole - Enter the number of grooves machined into the tube hole.



Fillet Weld Leg Size (if any) - Enter the fillet weld size, if used when the tube is welded to the tubesheet, as defined in ASME VIII Div 1, UW-20. The weld may be fillet-only, groove-only, or both, as shown below.

Groove Weld Leg Size (if any) - Enter the groove weld size, if used when the tube is welded to the tubesheet, as defined in ASME VIII Div 1, UW-20. The weld may be fillet-only, groove-only, or both, as shown below.



Design Strength (not for fixed TS types) - Enter the design strength axial load *Fd*, according to ASME Code paragraph UW-20. This value is used to determine the minimum acceptable fillet/groove weld size that connects the tube to the tubesheet. The design strength should not be greater than the tube strength (*Ft*), defined as $\Pi t(do - t)Sa$.

NOTES

- Because U-tube tubesheet exchangers do not experience differential thermal expansion between the tubes and the shell, the axial load on the tubes cannot be easily calculated. You can specify your own value for the actual load (required design strength) for U-tube exchangers. This value is optional for fixed and floating tubesheet exchangers.

- For partial strength tube-to-tubesheet welds on fixed/floating tubesheet exchangers, the higher of the actual tube-to-tubesheet load and the entered design strength is used to size the welds.
- For full strength tube-to-tubesheet welds on fixed/floating tubesheet exchangers, the tube strength (*Ft*) is used to size the welds.

Tube Weld Joint Type - Select the type of tube/tubesheet weld, as defined in ASME UW-20:

- **Full Strength** - The design strength is equal to or greater than the maximum allowable axial tube strength.
- **Partial Strength -** Design strength is based on the actual tube-tubesheet axial load.
- **Seal/No Weld** - The weld is used to seal and has no strength value, so no calculations are performed.

Tube Joint Type - Select the weld joint type, as defined by TEMA and ASME using ASME Section VIII, Division 1 Table A-2, Efficiencies and Joint Types:

Joint Type	**Description**	**Fr.(test)**	**Fr.(no test)**
a	Welded only, a >= 1.4t	1.00	.80
b	Welded only, t <= a < 1.4 t	.70	.55
b-1	Welded only, a < t	.70	...
c	Brazed, examined	1.00	.80
d	Brazed, not fully examined	0.50	.40
e	Welded, a>=1.4t, expanded	1.00	.80
f	Welded, a<1.4t, expanded, enhanced with 2 or more grooves	.95	.75
g	Welded, a<1.4t, expanded, enhanced with 1 groove	.85	.65
h	welded a 1.4t, expanded, not enhanced (no grooves)	.70	.50
i	Expanded, enhanced with 2 or more grooves	.90	.70
j	Expanded, enhanced with single groove	.80	.65
k	Expanded, not enhanced (no grooves)	.60	.50

For PD 5500, select the weld joint type as defined by Table 3.9-3, Efficiencies and Joint Types:

Joint Type	Description	Fr.(1)
a	Welded with min throat thk. >= tube thk.	.80
b	Welded with min throat thk. < tube thk.	.55
c	Expanded and welded with min throat thk. >= tube thk.	.80
d	Expanded and welded with min throat thk. < tube thk.	.55
e	Expanded only	.50
f	Explosion expanded/welded	.80

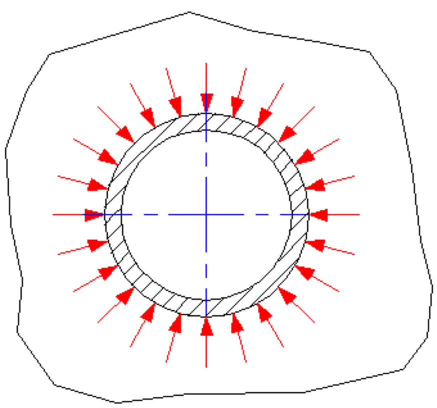
Allowable Joint Load Method - Select the joint allowable load method:

- **ASME APP. A** – Applicable for fixed and floating tubesheet heat exchangers, covering many types of tube-tubesheet joints, such as welded, brazed and expanded. The British code PD 5500 method for determining the tube-tubesheet joint allowable is similar to this method.
- **ASME UW-20** – Applicable for full strength and partial strength tube-tubesheet welds.
- **None**

Is Tube-Tubesheet Jt. Tested? - Select if the tube/tubesheet joint is tested and qualified for ASME. Tested joints get an increase in their strength value.

ASME Tube Jt. Reliability Factor - Displays the reliability factor value after a value for **Tube Joint Type** is selected.

Tube Expansion, Po - Enter the pressure (*Po*) exerted on the outside of the tube after it has expanded. This value is only required for a **Tube Joint Type** of **i**, **j**, and **k**.



Differential Thermal Expansion, Pt - Enter the interface pressure (*Pt*) between the tube and the tubesheet due to differential thermal growth. This value is only required for a **Tube Joint Type** of **i**, **j**, and **k**.

IMPORTANT The ASME code provides neither formulae to calculate *Po* and *Pt* pressures, nor indicates analytical or experimental methods to establish them. If you do not have good method to establish values of interface pressures then enter **0**. This tells the software to ignore the effects of interface pressures on the joint allowable. If you enter **1** as an approximation for both of these pressures, then the joint allowable doubles.

Is This a Welded Tube (Not Seamless)? - Select if the tubes are of welded construction.

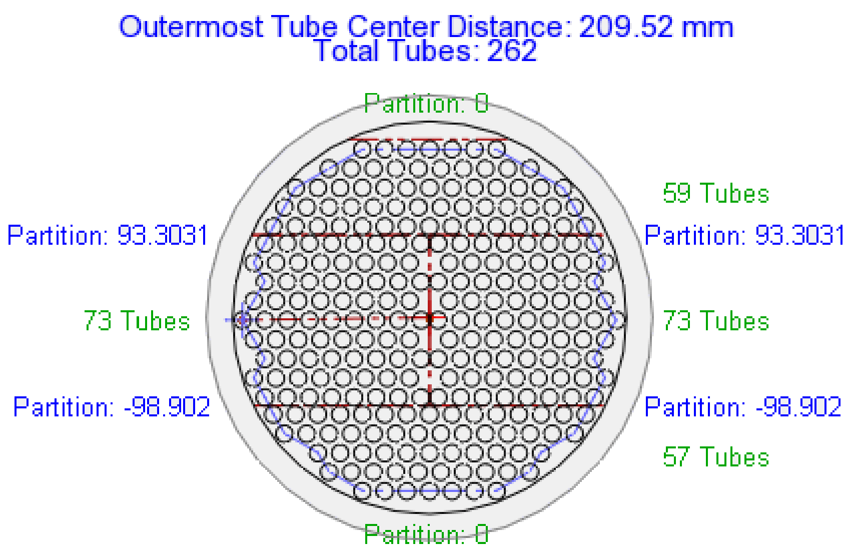
Specific Gravity of the Operating Liquid in the Tubes - Enter the specific gravity of the fluid in the tube channel side.

Tube Layout Assistant

The **Tube Layout Assistant** utility helps you determine the number of tubes that will fit in a layout configuration for a heat exchanger tubesheet. The utility is available from:

- The **Tube Data** tab of the **Heat Exchanger Tubesheet Input** dialog box, opened from **Tubesheet Analysis** in PV Elite.
- **TSLayout.exe**, found in the *[Product Folder]*\Intergraph CAS\PV Elite*[Version Number]*\ folder.

The distance from the center of the tubesheet to the outermost tube center and the total number of tubes is calculated. As you change options on the **Layout Pattern** tab, the layout graphics dynamically update in the right panel. For example:



NOTES

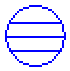
- The outermost tube center distance is used in tubesheet stress analysis.
- Tie rods that are located in tube positions are not considered.
- Layout files are saved with the .tsd extension. You can import a layout file into PV Elite.

Layout Pattern Tab

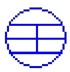
Units - Select **Imperial (in)** or **Metric (mm)**. You can switch units at any time.

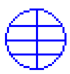
Partitions

Partition Layout - Select the partition plate layout within the shell:


- Horizontal partitions only. With this layout, **Number of Horizontal Partitions** can have values from **0** to **8**.

NOTE For a tubesheet with no partitions, select this option and set **Number of Horizontal Partitions** to **0**.


- Horizontal partitions with an interior vertical partition. With this layout, **Number of Horizontal Partitions** can have values from **2** to **8**.


- Horizontal partitions with a full vertical partition. With this layout, **Number of Horizontal Partitions** can have values from **2** to **8**.

Number of Horizontal Partitions - Type or select the number of horizontal partitions.

Partition Thickness - Type the thickness of the partition plates.

Partition Clearance - Type the clearance distance between tubes and the partition plates.

Symmetrical (Mirror) Layout - Select for a symmetrical tube layout.

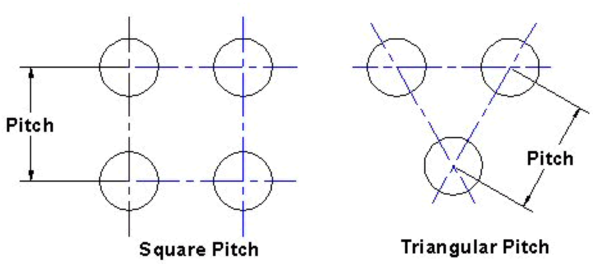
Tubes

Tube Pattern - Select the needed tube pattern. Select **Square**, **Square Rotated**, **Triangular**, or **Triangular Rotated**.

Tube Boundary (OTL) - Enter the outer tube limit (OTL) diameter of the shell. Tubes cannot be placed beyond the OTL.

Tube Outer Diameter - Enter the un-corroded outside diameter of each tube.

Tube Pitch - Enter the distance between tube centers.



Nozzle

Nozzle Clearance (Top, Bottom, Left, Right) - Enter the dimension that a nozzle projects inside the surface of the exchanger shell. Note that all dimensions can be specified. This allows for side and vertical entry nozzles.

Nozzle Clearance (Top) - Enter the distance that a nozzle projects vertically and from the top of the surface of the exchanger shell.

Nozzle Clearance (Bottom) - Enter the distance that a nozzle projects vertically and from the bottom of the surface of the exchanger shell.

Nozzle Clearance (Left) - Enter the distance that a nozzle projects horizontally and from the left of the surface of the exchanger shell.

Nozzle Clearance (Right) - Enter the distance that a nozzle projects horizontally and from the right of the surface of the exchanger shell.

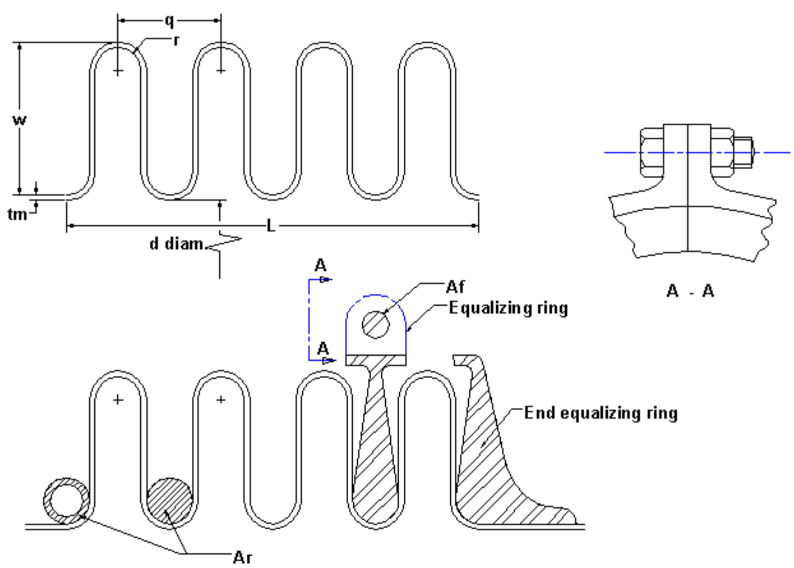
Shell

Shell Inside Diameter - Enter the inside diameter of the shell.

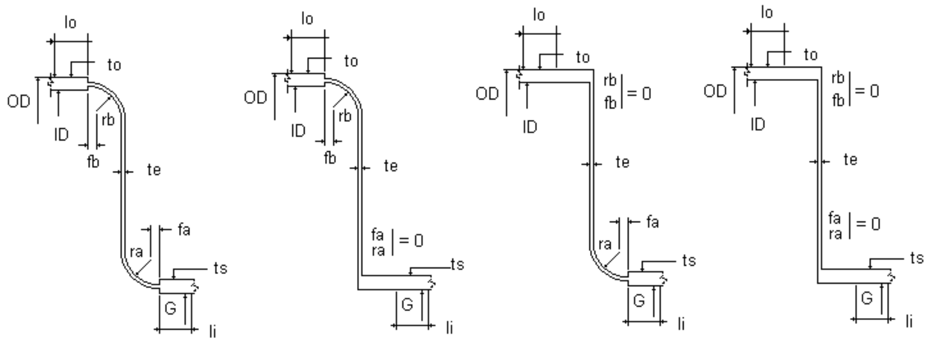
Expansion Joint Data Tab (Heat Exchanger Tubesheet Input Dialog Box)

Expansion joints are selected in **Expansion Joint Type (if any)** on the **Tubesheet Type and Design Code** tab. Two types are available:

Thin Joint is comprised of a thin bellows that is very flexible and has a low stiffness. Below are examples designed according to ASME Appendix 26:

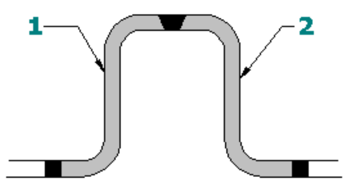


Thick Joint or *Flanged/Flued Expansion Joint* is comprised of a number of shell elements that are added together to form the joint. A thick joint is stiffer than the thin joint, and its stiffness must be taken into account. Below are typical combinations of flexible shell elements for a thick expansion joint, showing one-half of a convolution and using TEMA nomenclature:

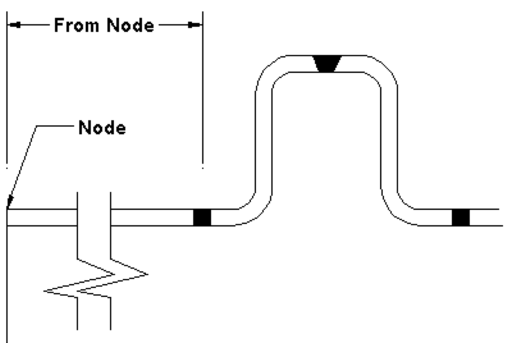


Perform App. 26 Stress and Life Cycle Calculations for the Bellows? - Select if **Thin Bellows Type** is selected for **Expansion Joint Type (if any)** on the **Tubesheet Type and Design Code** tab. Then click **>>** to define the thin joint according to ASME Appendix 26 analysis. For more information, see **Thin Joint Options** (on page 153).

Number of Flexible Shell Elements - Enter the number of flexible shell elements for a thick joint. One convolution has two flexible shell elements:



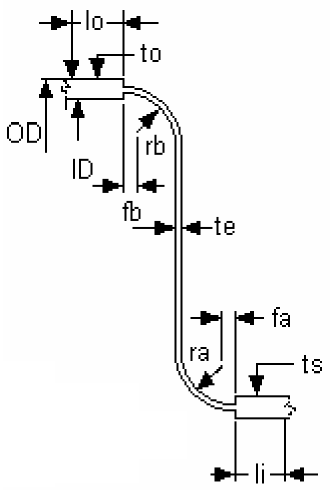
Dist. from "From" Node - Enter the axial or longitudinal distance from the **From Node** to the start of the expansion joint:



Design Option - Select **Existing** to enter the stiffness characteristics of the expansion joint. Use this option if you do not want to analyze the expansion joint itself but are just specifying its spring rate to be used in the tubesheet calculations. For example, you are purchasing the expansion joint from the manufacturer, who has already analyzed the joint. Select **Analyze** to calculate the stiffness characteristics (spring rate) and the stresses in the joint.

Set Defaults - Click to set the dimension starting points for the expansion joint design.

Enter values for the following options to define thick joint characteristics:



Expansion Joint ID

Expansion Joint OD

Wall thickness (te)

Corrosion Allowance

Knuckle Offset Dimension Inside (fa)

Knuckle Offset Dimension Outside (fb)

Knuckle Radius Inside (ra)

Knuckle Radius Outside (rb)

Shell Cylinder Length (Li)

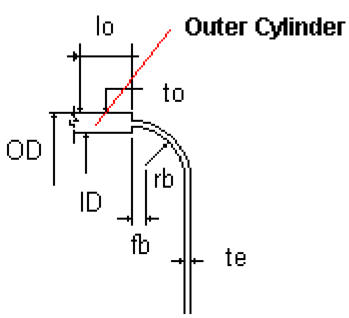
Thick Expansion Joint Calculation Method - Select **TEMA** or **Kopp and Sayre**.

Desired Cycle Life - Enter the number of cycles needed for the life of the expansion joint.

User Input Spring Rate Corroded - Enter a value for thin joints and for thick joints when **Existing** is selected for **Design Option**.

User Input Spring Rate Uncorroded - Enter a value for thin joints and thick joints when **Existing** is selected for **Design Option**.

Enter values for the following options to define outer cylinder characteristics:



Is there an Outer Cylinder?

Outer Cylinder Material

Outer Cylinder Thickness (to)

Outer Cylinder Corrosion Allowance

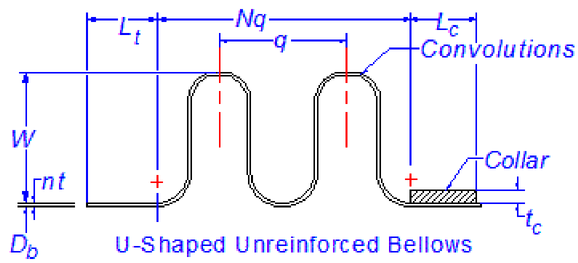
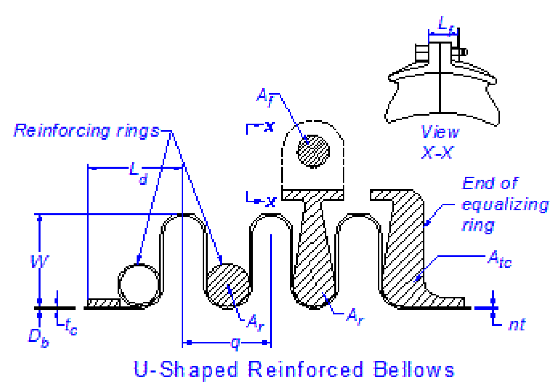
Outer Cylinder Length (lo)

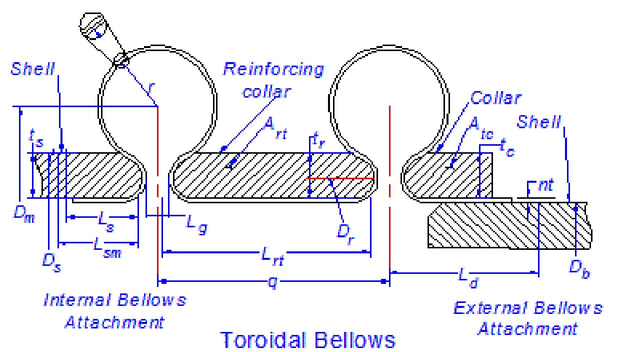
Outer Cylinder Design Temperature

Thin Joint Options

Thin expansion joints are analyzed according to ASME Section VIII, Division 1, Appendix 26 of the Boiler and Pressure Vessel Code. Life cycle and stress analysis is performed.

Enter values for the dimensions corresponding to your joint shape:

Dimension	Description	Sketch
A_f	Cross sectional metal area of one reinforcing fastener	
D_b	Inside diameter of bellows convolution	
D_m	Mean diameter of bellows convolution	
D_r	Cross sectional diameter of the reinforcing ring	
K_f	Forming method factor. Use 1.0 for expanding mandrel or roll forming. Use 0.6 for hydraulic, elastomeric, or pneumatic tube forming.	
L_c	Bellows collar length. For the toroidal bellows, *Lc* is determined by dividing the collar cross section area with the collar thickness.	
L_d	Length from attachment weld to the center of the first convolution for externally attached bellows	
L_f	Effective length of one reinforcing fastener	
L_g	Maximum distance across the inside opening of a toroidal convolution, considering all movements	
L_r	Effective reinforcing collar length, calculated by	

	$\frac{1}{3}\sqrt{D_r t_r}$	
L_{rt}	Overall length of reinforcing collar	
L_s	Effective shell strength, calculated by $\frac{1}{3}\sqrt{(D_s + t_s)t_s}$	
L_{sm}	Minimum required shell length having thickness, *t*	
L_t	End tangent length	
Lw	Distance between toroidal bellows attachment welds	
q	Convolution pitch	
r	Mean radius of toroidal bellows convolution	
t	Bellow nominal thickness of one ply	
tc	Collar thickness	
w	Convolution depth	

delta Q - Enter the differential thermal expansion between the tubes and shell divided by the number of convolutions. If this value is **0**, it is calculated during the analysis. If you are evaluating this joint by clicking **Quick Results**, you need to calculate delta Q and enter that value. In this case, resetting the value to **0** before exiting the dialog box is recommended. A non-zero positive value will be used by the software regardless of what was actually computed for **delta Q**. This entry is optional.

Load Cases Tab (Heat Exchanger Tubesheet Input Dialog Box)

Specifying load cases in addition to the design case enables PV Elite to perform calculations for different combinations of pressures and temperatures, ensuring that all the possible conditions are considered. Examples of additional cases are: shut-down, start-up, and upset. For each load case, values can be entered as needed for:

- Shell - The shell between the tubesheets.
- Channel - The channel at the ends of the exchanger.
- Tubes - The exchanger tubes.
- Tubesheet - The tubesheet. For fixed and floating tubesheets, it is one of the pair.
- Shell Band - The thicker shell courses at either end of the main shell next to tubesheet.

NOTE Enter **0** for any options that are not needed.

Number of cases to process - Select the number of load cases to run. You can specify up to eight different load cases.

Active Load Case - Click the up and down arrows to cycle through the load cases.

For each active load case, enter values as needed for the following:

Case Description - Enter a description.

Vacuum Pressures and Report Options for this Load Case - Click to open the **Report Print Options Dialog Box** (on page 157).

Design Pressure - Enter the design pressures for the shell side and for the channel side.

Design Temperature - Enter the design temperatures for the shell, channel, tubes, and tubesheet.

Specify Operating Temperatures (UHX) - For ASME UHX, select, and then enter the operating temperatures for the shell, channel, tubes, and tubesheet. UHX-13.4(b) Elastic moduli, yield strengths, and allowable stresses are taken at design temperatures. For cases involving thermal loading (loading cases 4, 5, 6, and 7), you can use the operating temperatures instead of the design temperatures (see UG-20).

Material - Enter the name of the material for tubes, tubesheet and, shell bands. This software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. Alternatively, use **Database Lookup and Properties.**

Mean Metal Temperature Along Length - Enter the actual metal temperature for the shell and tubes.

Metal Temperature at Tubesheet Rim - Enter the actual temperature of metal at the outer rim of the tubesheet, where the tubes are in the outer rows, for the shell, channel, and tubesheet.

Database Lookup and Properties - Click **Tubes**, **Tubesheet**, and **Shell Band** to select a material for each directly from the **Material Database Dialog Box** (on page 485). The selection is then entered into **Material**. To modify the material properties, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties for this analysis. It does not modify the database.

Exchanger subject to cyclic or dynamic reactions due to pressure or thermal variations? (see UHX-13.8) - Select to calculate for the effect of radial thermal expansion adjacent to the tubesheet, according to UHX 13.8 (fixed tubesheet heat exchangers) or UHX 14.6 (floating tubesheet heat exchangers).

Modulus of Elasticity

User-Defined Values - Select to enter your own modulus of elasticity values, overriding the material properties from the design code selected for analyzing the exchanger.

Modulus at Temperature - Enter the modulus of elasticity at the design temperature.

Modulus at Mean Metal Temp Along Length - Enter the modulus of elasticity at the actual metal temperature.

Modulus at Mean Metal Temperature - Enter the modulus of elasticity for the actual metal temperature at the channel.

Modulus at Ambient Temperature - Enter the modulus of elasticity at ambient temperature.

Coefficient of Thermal Expansion (alpha values)

User-Defined Values - Select to enter your own coefficient of thermal expansion values, overriding the material properties from the design code selected for analyzing the exchanger.

Alpha at Mean Metal Temp Along Length - Enter the actual metal temperature for the shell, tubes, and shell band.

Alpha at Metal Temp at Tubesheet Rim - Enter the actual temperature of metal at the outer rim of the tubesheet where the tubes are in the outer rows.

Differential Pressure Design? - Select to perform the analysis only for the difference between the shell side pressure and the tube side pressure.

Differential Design Pressure - Enter the differential pressure when **Differential Pressure Design?** is selected. If you do not enter a value, the software uses the effective pressures dictated by the code. this entry is optional.

Is the Exchanger Operating in the Creep Range (skip EP, use 3S for Sps)? - Select if the exchanger is operating in high-temperature creep range. This option only applies if the method used to analyze the heat exchanger is ASME.

Expansion Joint Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▸ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Report Print Options Dialog Box

Sets options for analysis result reports of each load case.

Select the following as needed:

- **For ASME fixed/stat. tubesheets, print intermediate results in tabular form**
- **Print intermediate results for expansion joint calculations**
- **For TEMA and PD:5500 run multiple load cases** - Runs the following load cases for fixed tubesheets:

Load Case #	Load Case Description	
	Corroded	**Uncorroded**
1	Fvs + Pt - Th + Ca	Fvs + Pt - Th - Ca
2	Ps + Fvt - Th + Ca	Ps + Fvt - Th - Ca
3	Ps + Pt - Th + Ca	Ps + Pt - Th - Ca
4	Fvs + Fvt + Th + Ca	Fvs + Fvt + Th - Ca
5	Fvs + Pt + Th + Ca	Fvs + Pt + Th - Ca
6	Ps + Fvt + Th + Ca	Ps + Fvt + Th - Ca
7	Ps + Pt + Th + Ca	Ps + Pt + Th - Ca
8	Fvs + Fvt - Th + Ca	Fvs + Fvt - Th - Ca

Shell-side / Channel-side Vacuum Pressures - Enter positive values. For example, for full atmospheric vacuum condition enter a value of 15.0 psig. If no value is entered, then 0 psi is used.

Set detailed printout options for the currently selected load case - When you want to see detailed equations and intermediate calculations in the results, select the corroded and uncorroded load cases. For all other load cases, the software generates summarized, tabular results.

For ASME tubesheets, the following load cases are run for fixed and floating tubesheet exchangers:

Load Case #	Load Case Description	
	Corroded	**Uncorroded**
1	Fvs + Pt - Th + Ca	Fvs + Pt - Th - Ca
2	Ps + Fvt - Th + Ca	Ps + Fvt - Th - Ca
3	Ps + Pt - Th + Ca	Ps + Pt - Th - Ca

4	Fvs + Fvt + Th + Ca	Fvs + Fvt + Th - Ca
5	Fvs + Pt + Th + Ca	Fvs + Pt + Th - Ca
6	Ps + Fvt + Th + Ca	Ps + Fvt + Th - Ca
7	Ps + Pt + Th + Ca	Ps + Pt + Th - Ca

For ASME stationary tubesheet configuration "d" and ASME floating tubesheet configurations "B", "C" and "D", the design is based only on load cases 1, 2 and 3.

The following load cases are performed for ASME U-tube tubesheet exchangers:

Load Case #	**Load Case Description**	
	Corroded	**Uncorroded**
1	Fvs + Pt - Th + Ca	Fvs + Pt - Th - Ca
2	Ps + Fvt - Th + Ca	Ps + Fvt - Th - Ca
3	Ps + Pt - Th + Ca	Ps + Pt - Th - Ca

For all ASME exchangers, if vacuum pressures are specified, then an additional load case is run:

Load Case #	**Load Case Description**	
	Corroded	**Uncorroded**
8	Fvs + Fvt - Th + Ca	Fvs + Fvt - Th - Ca

Additionally, if **Differential Pressure Design?** is selected on the **Load Cases** tab, then only certain load cases are run.

NOTES

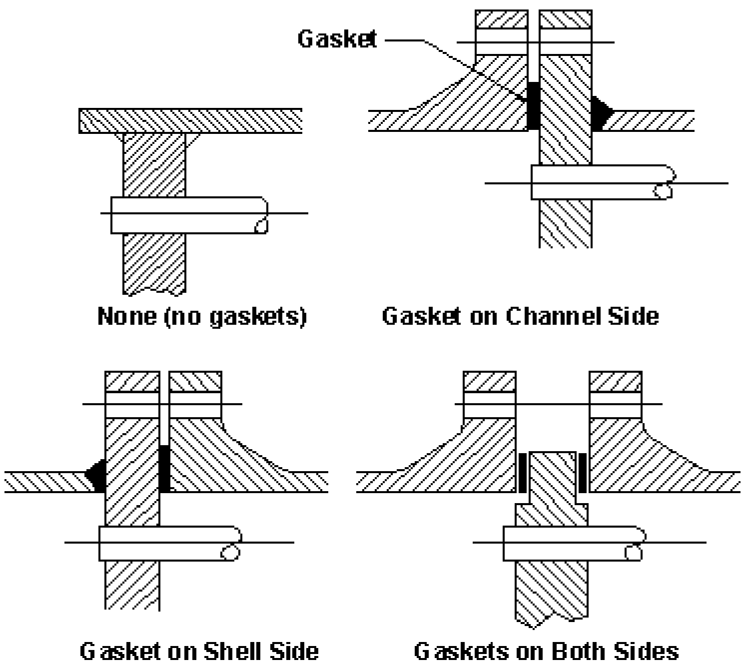
- Fvt, Fvs - User-defined Shell-side and Tube-side vacuum pressures or 0.0.
- Ps, Pt - Shell-side and Tube-side Design Pressures.
- Th - With or Without Thermal Expansion.
- Ca - With or Without Corrosion Allowance.

Floating TubeSheet Tab (Heat Exchanger Tubesheet Input Dialog Box)

Description - Enter the description for the floating tubesheet. The description appears on the final report.

Floating Tubesheet Type - Select the type of floating tubesheet to analyze, according to ASME UHX:

- **(a) Floating tubesheet, integral**
- **(b) Floating tubesheet, gasketed, extended as flange**
- **(c) Floating tubesheet, gasketed, no extended, w/backing device**
- **(d) Floating tubesheet, internal sealed**

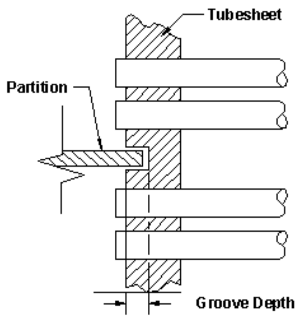


Outside Diameter - Enter the outside diameter of the tubesheet.

Tubesheet Thickness - Enter the tubesheet thickness in uncorroded condition. If it is a re-rate, then the actual measured thickness is typically used.

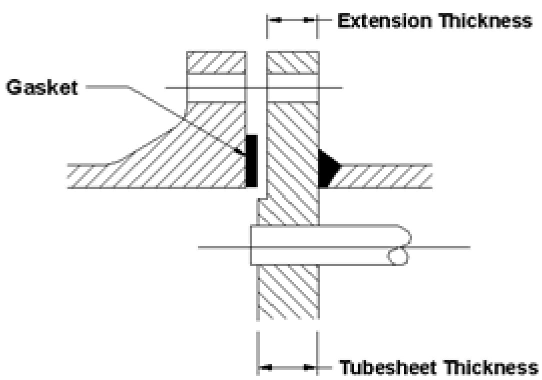
Corr. Allow. Shell Side / Channel Side - Enter the corrosion allowance on the shell side (the inner face of the tubesheet), and the corrosion allowance on the channel side (the outer face of the tubesheet facing the channel side).

Depth of Groove in Tubesheet (if any) - Enter the depth of a groove in the tubesheet, used to locate the channel partition plate and its gasket. If there is no groove for example in a single pass exchanger, this value is **0**.



Tubesheet Extended as Flange? - Select if the tubesheet is extended as a flange, so that it is subject to the bolt load from the mating flange.

Thickness of Extended Portion - When **Tubesheet Extended as Flange?** is selected, enter the thickness of the portion of the tubesheet that is extended for bolting.



Bolt Load Transferred to Tubesheet? - When **Tubesheet Extended as Flange?** is selected, also select this option if the bolt load is transferred to the tubesheet, extended as the flange. Do not select if the tubesheet is gasketed with both the shell and channel flanges. Otherwise, the tubesheet can still be extended, but the bolt load is not transferred to the tubesheet extension. Carefully consider all possible cases, such the hydrotest. When this option is not selected, the required thickness of tubesheet extension is not calculated.

Integral Channel Properties for ASME Floating Configuration A

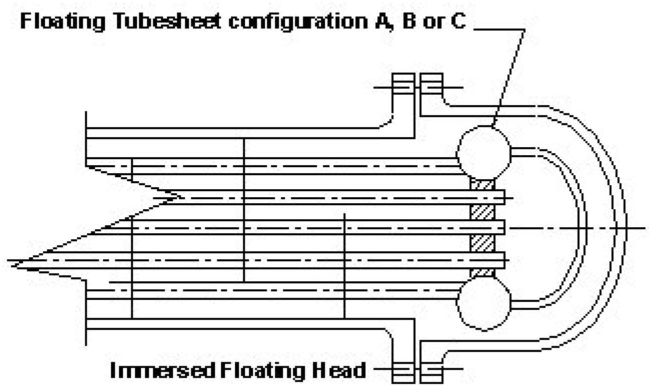
These properties are needed for floating heat exchangers with the floating tubesheet welded to head.

- **Channel Thickness tc**
- **Channel Design Temperature**

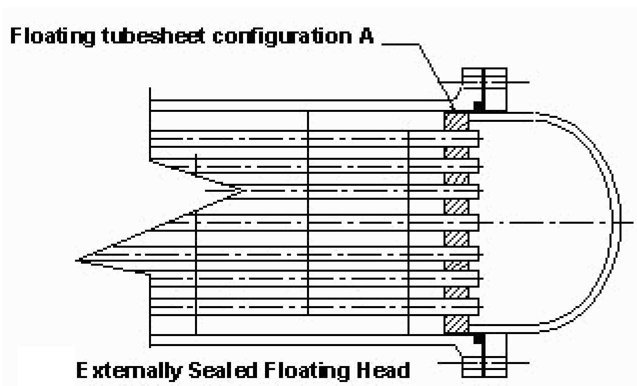
- **Channel Material** - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

In ASME UHX nomenclature this is Configuration A:

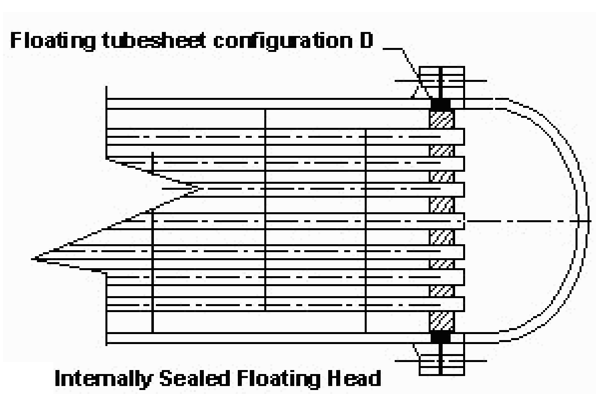
Exchanger with an Immersed Floating Head:



Exchanger with an Externally Sealed Floating Head:



Exchanger with an Internally Sealed Floating Head:

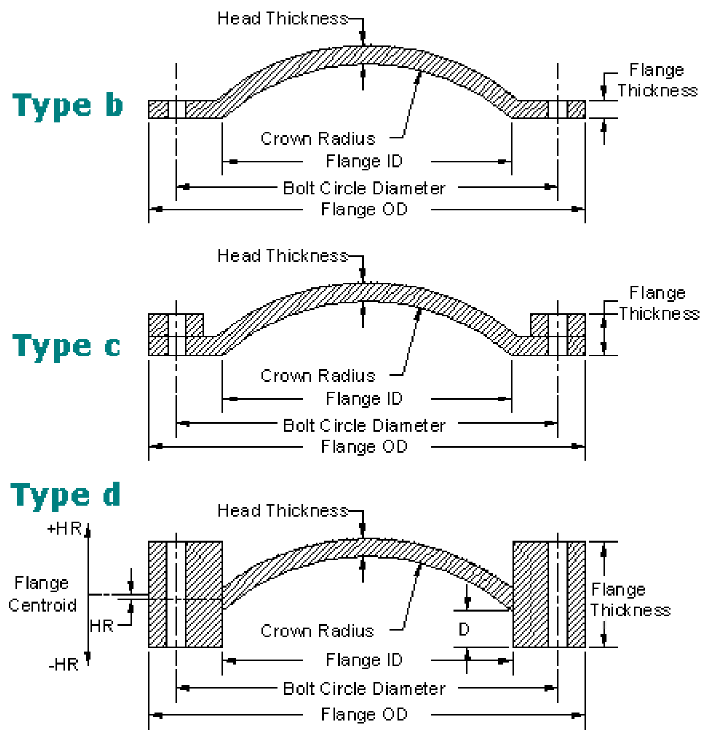


Spherical Cover/Backing Ring Tab (Heat Exchanger Tubesheet Input Dialog Box)

Description - Enter the description for the spherical cover. The description appears on your final report.

Design Temperature - Enter the design temperature for each head. This temperature is used to interpolate the material allowable tables and external pressure curves.

Type of Floating Head (ASME App. 1-6) - Select the type of floating head or spherically dished cover, corresponding to Figure 1-6 of ASME, Section VIII, Division 1, Appendix 1:



b - Solid thick head, spherically dished.

c - Thin dished head, continuous across flange face.

d - Spherical cap welded to flange ID.

None

NOTE **d** is the most common type of head used for heat exchanger floating heads.

NOTE Click ... to open the **Flange and Gasket Information** dialog box. Enter flange, gasket, and bolt data for the select floating head type.

Flange Thickness (tf) - Enter the through thickness of the flange. For type **c** spherical caps, this includes the thickness of the head.

Inside Crown Radius (L) - Enter the inside crown radius, usually roughly equal to the flange ID. This value may be any dimension greater than the inside radius of the flange.

Head Thickness (t) - Enter the minimum thickness of the actual plate used to build the floating head or spherical cap, or the minimum thickness measured for an existing floating head or spherical cap.

Head Internal Corrosion Allowance - Enter the corrosion allowance on the concave side of the head. The software adjusts the thickness and the diameter for the evaluation of allowable pressure. The allowance is also added to the required thicknesses. Some common corrosion allowances are:

- 0.0625 - 1/16"
- 0.1250 - 1/8"
- 0.2500 - 1/4"

Head External Corrosion Allowance - The software adjusts the thickness and the diameter for the evaluation of allowable pressure. The allowance is also added to the required thicknesses.

Slotted Flange? - Select if the flange has slotted bolt holes for quick opening. A slotted flange has bolt holes extending radially to the outer edge of the flange. The software automatically adjusts for this condition; you do not have to change the flange outside diameter.

Full Face Gasket Option - Select **Program Selects**, **Full Face Gasket**, or **Not a Full Face**. A full face gasket extends from the ID to the OD of the flange, enclosing the bolt holes. These gaskets are usually soft materials, such as rubber or an elastomer, so that the bolt stresses do not go too high during gasket seating. The software adjusts the flange analysis and the design formulas to account for the full face gasket.

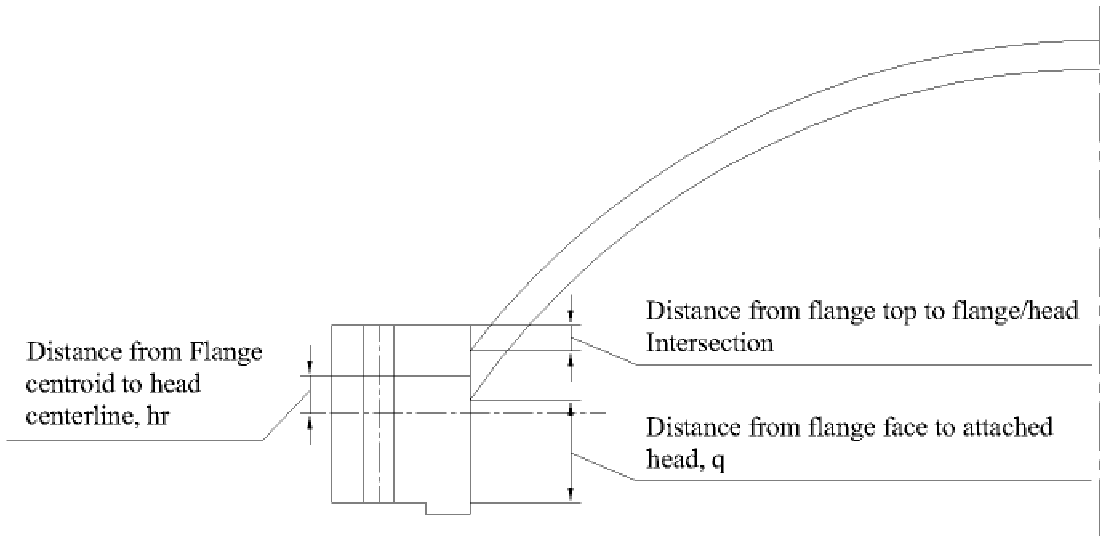
Head Material Name - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Flange Material Name - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Floating Head Factors (Spherical Cover/Backing Ring Tab)

Compute "F" even if the pressure is 0 - Select to calculate the factor *F* in the floating head even when the internal pressure is zero. *F* is a direct function of the internal pressure. If the internal pressure is 0, then *F* is equal to 0. However, a conservative interpretation of the code always calculates *F* regardless of the case being analyzed. Typically, the flange-bolt-up case is in question because there is no internal pressure when bolting up the unit, so this option is not selected. Select this option only for a conservative calculation.

Distance from Flange Centroid to Head Centerline (hr) - Enter the distance *hr* from the flange centroid to the intersection of the head centerline and the flange. Enter the value in the corroded condition. The value is positive if it is above the flange centroid, and negative if it is below the flange centroid. This distance is used in the code calculation but not when **Perform Soehren's Calc** is selected (where *Q* is used). See the illustration below.



Distance from Flange Top to Flange/Head Intersection - Enter the distance from the top of the floating head flange to the intersection of the dished head and the flange in the uncorroded condition, and then click **Compute**. The software considers the corrosion allowance, calculates *hr* in the corroded condition, and places its value in **Distance from Flange Centroid to Head Centerline (hr)**. Other than for this purpose, this value is not used by the software.

Compute - If **Distance from Flange Top to Flange/Head Intersection** is known, you can enter it (in the uncorroded condition) and click **Compute**. The software considers the corrosion allowance, calculates *hr* in the corroded condition, and places its value in **Distance from Flange Centroid to Head Centerline (hr)**.

Perform Soehren's Calc - Select to perform Soehren's calculation, a more detailed analysis of the interaction between the spherical cap and the flange. The stresses calculated are frequently acceptable for heads or flanges that are slightly less thick than required by the normal code rules. This analysis can only be done for type **d** floating heads. Par. 1-6(h) of the code allows this type of analysis.

Dim Q - Enter the distance *Q* from the flange bolting face to the intersection of the attached head inside diameter and the flange. *Q* is used in the Soehren's calculation, but not in the code calculation (where *hr* is used). See the illustration above.

Backing Ring Data (Spherical Cover/Backing Ring Tab)

Is There a Backing Ring? - Select if a backing ring is used. A backing ring is a second flange used to sandwich the tubesheet of a floating head heat exchanger.

Backing Ring Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Backing Ring Inside Diameter - Enter the inside diameter of the backing ring. This value is usually slightly smaller than the outside diameter (OD) of the flange.

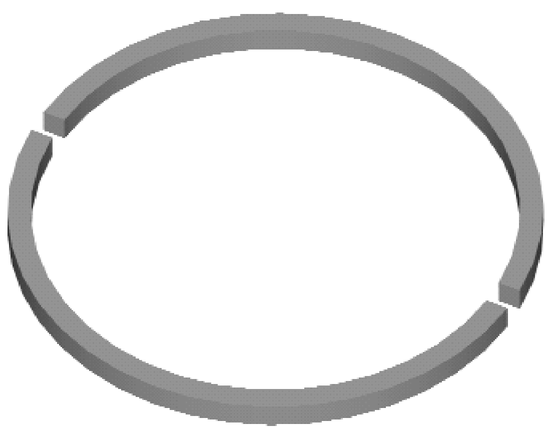
Backing Ring Outside Diameter - Enter the outside diameter of the backing ring. This value is usually slightly larger than the inside diameter (ID) of the flange.

Backing Ring Thickness - Enter the actual through thickness of the backing ring. For doubly split rings (when **2** is selected for **Number of Splits in Backing Ring)**, this is the thickness of each piece.

Number of Splits in Backing Ring - Select the number of splits in the ring, if any, for loose-type flanges. Select **0**, **1**, or **2**. Split flanges are typically ring-type flanges. A split is used when the flange must be completely removable from the vessel.

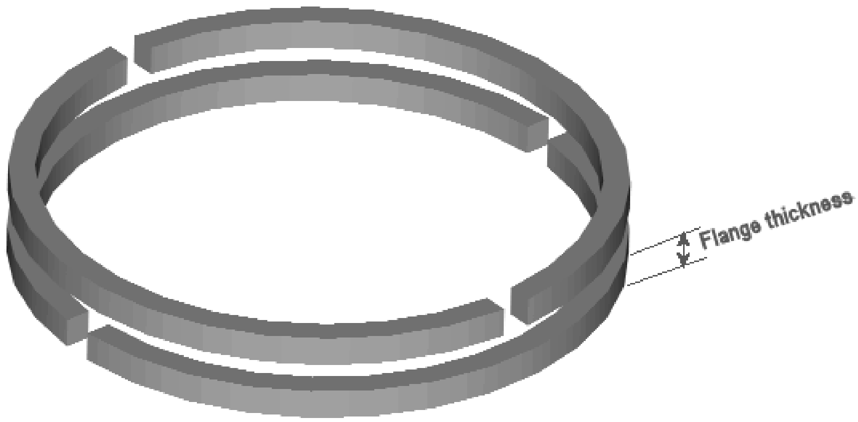
- Ring with a single split:

  When the flange is split into two pieces by a single split, the split is along the diameter, and the design bending moment for the flange is multiplied by 2.0.



- Ring with a double split:

A ring with two splits has two stacked rings, with each half split along the diameter. The software analyzes each ring as if it were a solid flange (without splits) using 0.75 times the design bending moment. **Backing Ring Thickness** is the thickness of each piece. The thickness of the total ring is twice this value. The pair of rings is assembled so that the splits in one ring are at 90º from the splits in the other ring.



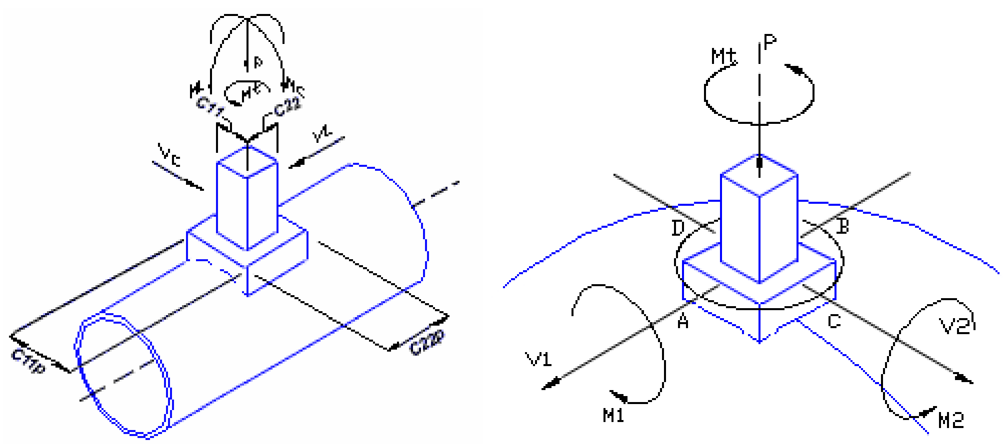
NOTE TEMA RCB-5.141 shows different styles of backing devices. Styles A and D have a split ring, but the moment used to design them is not increased. When you have one of these styles, enter zero as the number of splits to get the same effect in CodeCalc (available by clicking **Component Analysis**).

Generic Clip

Home tab: **Details** > **Generic Clip**

Adds a clip to the vessel. In the dialog, enter clip support information on cylinders as well as elliptical, torispherical and spherical heads. Clips are used to carry load, such as from piping, ladders, and platforms. These loads, along with pressures, cause local stress at the clip support location. WRC 107/537 is used to calculate the local stresses and compare them to the allowables.

Sustained, expansion, and occasional loads on the clip must be determined from a separate analysis and entered. Clips supporting piping generally have loads in all categories, while platform clips generally have only sustained and occasional loads.



From Node - Displays the **From Node** for the selected element.

Description of Clip - Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

Distance from "From" Node - Enter the distance from the **From Node** to the center of the clip.

Layout Angle - Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the clip at the point where it is attached to the shell, following the same conventions used for a nozzle. For more information, see **Layout Angle**.

Is the Clip Circular? - Select if the clip has a circular cross-section.

Clip Parameters

Circumferential Length (C11) - Enter the length that the clip extends around the circumference of the cylinder element. For spherical head elements that can be analyzed using this method, the clip must be square in cross-section.

Longitudinal Length (C22) - Enter the length that the clip extends along the length of the cylinder element. For spherical head elements that can be analyzed using this method, the clip must be square in cross-section.

Thickness - Enter the distance that the pad extends radially from the shell surface or reinforcing pad. The thickness is not used to calculate stresses, but is used to calculate the weight and draw the 3D image.

NOTE When **Is the Clip Circular?** is selected, the clip parameters are **Clip Outside Diameter**, **Clip Outside Projection**, and **Wall Thickness**.

Pad Parameters

Reinforcing pad used? - Select if there is a pad under the clip.

Circumferential Length (C11p) - Enter the length that the pad extends around the circumference of the cylinder element. For spherical head elements that can be analyzed using this method, the pad must be square in cross-section.

Longitudinal Length (C22p) - Enter the length that the pad extends along the length of the cylinder element. For spherical head elements that can be analyzed using this method, the pad must be square in cross-section.

Thickness - Enter the distance that the pad extends radially from the shell surface. The thickness is not used to calculate stresses, but is used to calculate the weight and draw the 3D image.

NOTE When **Is the Clip Circular?** is selected, the pad parameters are **Pad Diameter** and **Thickness**.

Fatigue Parameters

Use Kn and Kb - Select to use the Kn and Kb parameters, based on the fillet radius for the clip or pad and used for fatigue analysis.

Clip Fillet Radius - Enter the clip fillet weld radius. See Appendix B in the WRC 107/537 bulletin for illustrations and more information.

Pad Fillet radius - Enter the pad fillet weld radius. See Appendix B in the WRC 107/537 bulletin for illustrations and more information.

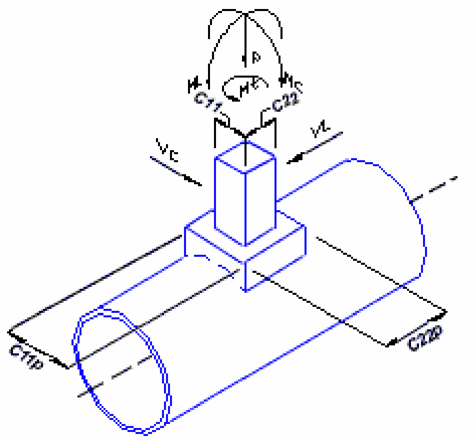
Local Attachment Loads at the Shell Surface

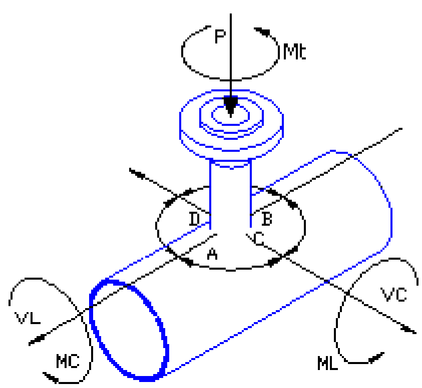
You can enter values in the following load sets:

- **Sustained** - (SUS) Primary loads, typically weight + pressure + forces.
- **Expansion** - (EXP) Secondary, self-limiting thermal expansion loads.
- **Occasional** - (OCC) Irregularly occurring, short-term loads, such as wind loads, seismic loads, and water hammer.

These local loads are used to analyze the stresses at the base of the clip in the shell. They are not used by the software to produce a global bending moment over the entire vessel cross section (Use **Force and Moment** to do this). The positive orientation of the loads is shown below. These loads are generally calculated, given as defaults or calculated by a stress analysis program such as CAESAR II.

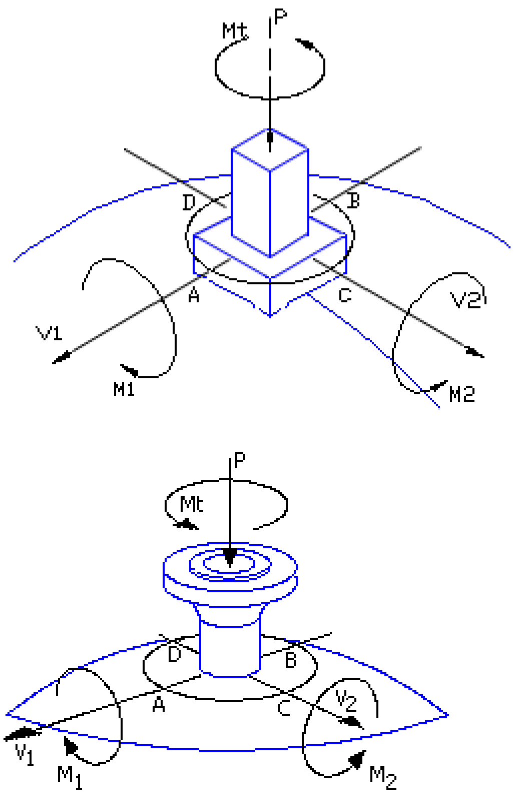
The following force/moment convention is used for a square or circular clip on a **cylindrical** element:





P - Radial Force
Vc - Circ. Shear Force
Vl - Long. Shear Force
Mc - Circ. Moment
Ml - Long. Moment
Mt - Torsional Moment

The following force/moment convention is used for a square or circular clip on a **spherical** vessel:



P - Radial Force
V2 - Shear Force, D to C
V1 - Shear Force, B to A
M1 - Moment, B axis
M2 - Moment, C axis
Mt - Torsional moment

Previous - If you created more than one clip on the element, click to go back to the previous clip.

Go To Next Clip - If you created more than one clip on the element, click to go to the next clip.

Add New Clip - Click to add a new clip to the element.

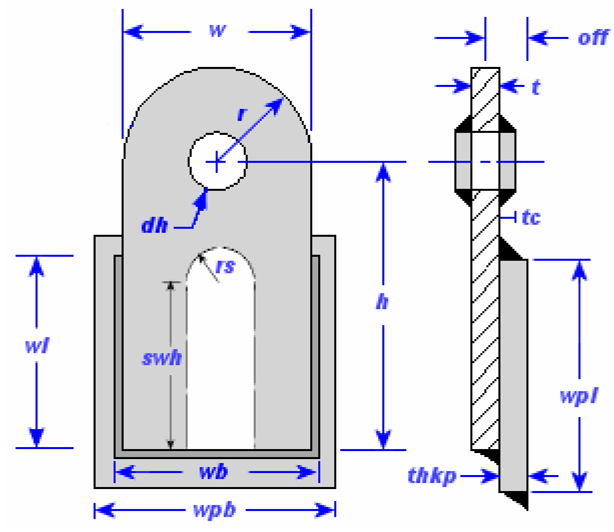
Delete - Deletes all data for the current clip.

Lifting Lug Data

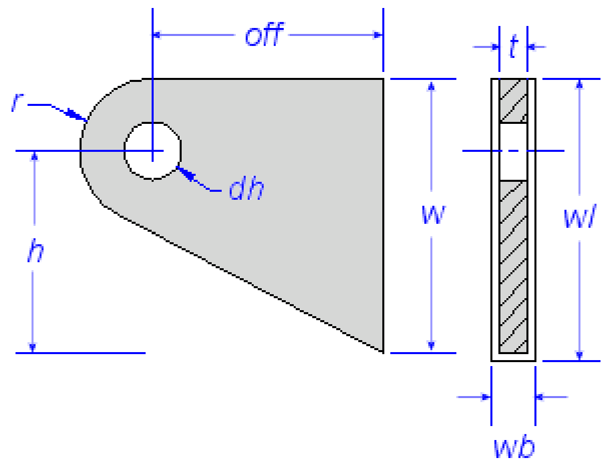
Home tab: **Details** > **Lifting Lug Data**

Adds lugs to the selected element. You can create:

- Flat-type lugs on vertical vessels, generally place near the top head.



- Perpendicular, ear-type lugs for horizontal vessels.



The software calculates the reactions on each lug, the stresses in each lug, and the stress in the welds.

Delete - Deletes all data for the lug.

Common Detail Parameters (on page 52)

Layout Angle - Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the lug at the point where it is attached to the shell, following the same conventions used for a nozzle. For more information, see **Layout Angle** on the **Nozzle Orientation (Nozzle Main Tab)** (on page 61).

Lug Contact Width (w) - For a flat lug, enter the width of the lug at the base. For a perpendicular lug, enter the longer length of the lug along the surface.

Diameter of Hole in Lug (dh) - Enter the diameter of the hole cut or drilled into most lifting lugs.

Radius of Semi-circular Arc (r) - Enter the radius of the semi-circular part of the lifting lug where the hole is located. This is typically circular on flat lugs and semi-circular on perpendicular lugs.

Height from Bottom to Center of Hole (h) - Enter the distance along the axis of the vessel from the center of hole to the bottom of the lug.

Offset from Vessel OD to Center of Hole (off) - Enter the distance from the center of the hole to base of the lifting lug. For perpendicular lugs this is to the vessel OD. If the orientation is flat this is 1/2 the thickness of the lug.

Lug Fillet Weld Size (tf) - Enter the fillet weld leg size. For stress analysis of the welds, the leg dimension is converted into the throat dimension.

Length of Weld Along Side of Lifting Lug (wl) - Enter the length of the long welds on the side of the lifting lug. The software multiplies this value by two when determining the weld area.

Length of Weld Along Bottom of Lifting Lug (wb) - Enter the length of the short weld. This is usually the bottom weld.

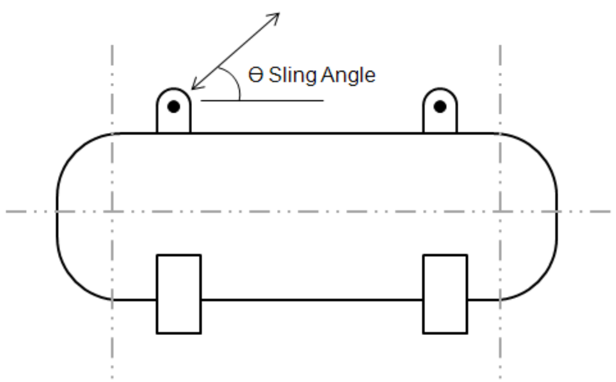
Collar Thickness (tc) - If the lug has a collar, enter the thickness. The thickness is measured from the outside surface of the lug to the edge of the collar.

Collar Diameter (dc) - Enter the diameter of the collar. This value is mainly used for documentation.

Lug Thickness (t) - Enter thickness of the lifting lug plate.

Lug Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

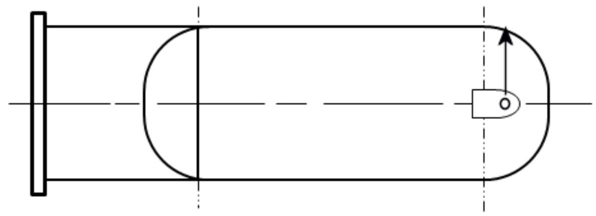
Sling Angle from Horizontal - Enter the sling angle in degrees:



Occasional Load Factor - Enter an occasional load factor, used in many construction codes to increase the allowable stress for an event that is considered occasional in nature. Such occasional loads are wind, seismic, and the lifting of a vessel. The occasional load factor is multiplied by other terms in the allowable stress equation to get the overall allowable. If you do not want to use this value, enter **1**.

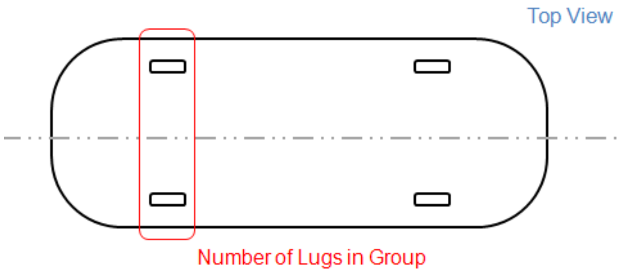
Impact Factor - Enter the impact factor to account for lifting a vessel when it may be pulled quickly with sudden force. This value typically ranges from **1.5** to **2.0**, although values as high as

3.0 may be entered. The software multiplies the lifting loads by the impact factor. The theoretical maximum value for this value is 2.0.



Number of Lugs in Group - Enter the number of lugs in this group:

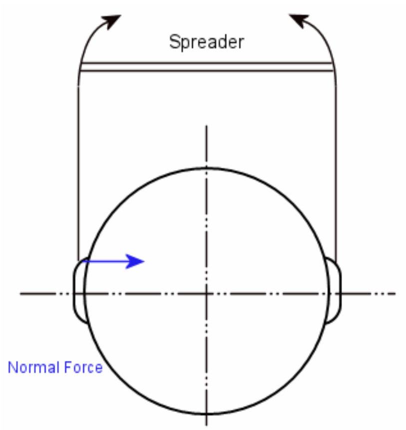
- For a horizontal vessel, there is typically one lifting lug on each side of the vessel. The number of lugs in a group is **1**. If there are two lugs on each end (such as for a large vessels), the value is **2**.



- For a vertical vessel, there are typically two flat lugs located near the top of the vessel. The number of lugs in a group is 2. A value of **1** for a vertical vessel should never be used.

Additional Normal Force (Fn) or **Tangential Force on Lug** - Enter an additional normal force acting on the lifting lug. This force is typically acting when there is no spreader bar used in the lifting procedure. For a horizontal vessel, enter the tangential force acting on the lug.

The normal or tangential force causes weak axis bending on the lug. Even a moderate force can cause a high bending stress. This is to be avoided. If there is an additional force, enter the value for each lug. The software applies the impact factor to this value.



-

Is there a pad? - Select if there is a pad under the lug.

Circumferential Width (wpb) - Enter the width of the pad along the circumference of the vessel.

Longitudinal Length (wpl) - Enter the length of the pad along the long axis of the vessel.

Pad Thickness (thkp) - Enter the thickness of the pad plate.

Pad Fillet Weld Size (tfp) - Enter the weld leg length for the pad fillet weld.

Is the lug slotted? - Select if the lifting lug is slotted.

Slot Radius (rs) - Enter the slot radius.

Slot Weld Height (swh) - Enter the slot weld height along the axis of the vessel from the center of slot hole to the bottom of the lug.

ASME Appendix 9 Jacket

Home tab: **Details** > **ASME Appendix 9 Jacket**

Adds ASME Appendix 9. Type 1 cylindrical jackets to the shell on the selected cylinder element, according to ASME Code, Section VIII, Division 1, Appendix 9.

NOTE This command is only available when **Cylinder** is selected as the **Element Type**, and **ASME Section VIII-Division 1** was selected as the design code for **File** > **New** (on page 48).

A Type 1 jacket is specified as either a jacket or a vapor belt. Vapor belts cover perforated areas of some vessels but are more typically found on shell and tube heat exchangers. Jackets can have separate design conditions from the parent cylindrical element. Vapor belts have the same design conditions as the parent cylindrical element. The software calculates the required thickness and MAWP for the jacket and closure bars. The MAWP of vapor belts can influence nozzle design as well as the overall MAWP of the vessel. The software calculates the stiffened length of the inner vessel to account for the closure bars.

Both jackets and vapor belts can have nozzles attached. If a nozzle has an internal projection that cuts into the parent shell, the nozzle reinforcement calculations are for the parent shell. If the nozzle does not have an internal projection, the calculation is for the jacket. According to Appendix 9, nozzle calculations are not required for both the jacket and inner vessel.

The internal pressure in a vapor belt adds to the vacuum (external) pressure in the inner shell. This extra pressure is reflected in the external pressure report generated during analysis.

IMPORTANT

- Appendix 9 jackets are joined by inter-connecting pipework, affecting static pressure head considerations.
- Enter the specific gravity of the fluid in the jacket or vapor belt. The software uses this value to calculate the weight of the fluid in the annular area.

Delete - Deletes all jacket data.

From Node - Displays the From Node for the selected element. The **From Node** is the software-generated node number describing the starting location of the element. The value cannot be modified.

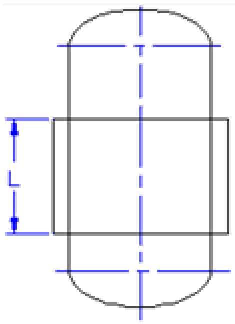
Jacket Length L - Enter the length of the jacket along the cylinder element.

Dist. from 'From' Node to Jacket Bottom - Enter the distance from the From node to the bottom of the jacket.

Jacket Description -Enter a description for the jacket.

Is this a Vapor / Distribution Belt (cut outs in shell)? - Select if the jacket is a vapor belt or distribution belt.

Select Jacket Figure (Fig. 9-2) - Displays the **Type 1** jacket configuration. Other jacket configurations in ASME Section VIII, Division 1, Appendix 9, Fig 9-2 are not supported.



Jacket Longitudinal Efficiency - Enter the welded joint efficiency, as defined in ASME Section VIII, Division 1, Table UW-12. For the weld on a **Type 1** jacket (welded from both sides or with a removable backing strip), use the following efficiencies:

- Full radiography: 1.00
- Spot x-ray: 0.85
- No radiography: 0.70

Jacket Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Design Temperature, Internal - Enter the design temperature for internal pressure. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Design Temperature, External - Enter the design temperature for external pressure. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Jacket Pressure, Internal - Enter the design pressure for internal pressure analysis. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Jacket Pressure, External - Enter the design pressure for external pressure analysis. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Jacket Thickness - Enter the jacket thickness.

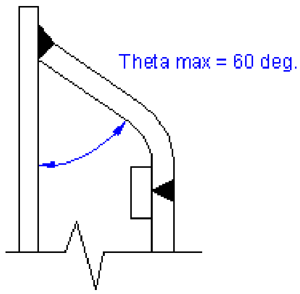
Jacket Corrosion Allowance - Enter the thickness of the corrosion allowance for the jacket.

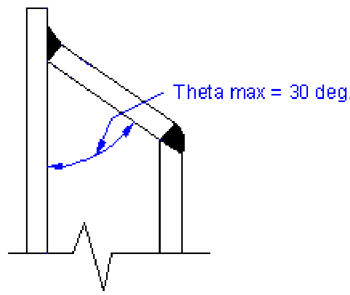
Jacket Length for External Pressure - Enter the length of the jacket to use for external pressure calculations.

Jacket Inside Diameter - Enter the inside diameter of the jacket.

Specific Gravity of Contents - Enter the specific gravity of any fluid contained within the jacket. This value is usually **1.0**.

Half Apex Angle - When **Type (b-2)**, **Type (c)**, or **Type (k)** is selected for **Select Closure Type (fig. 9-5)**, enter the half apex angle θ_{max} (or theta max). For example:



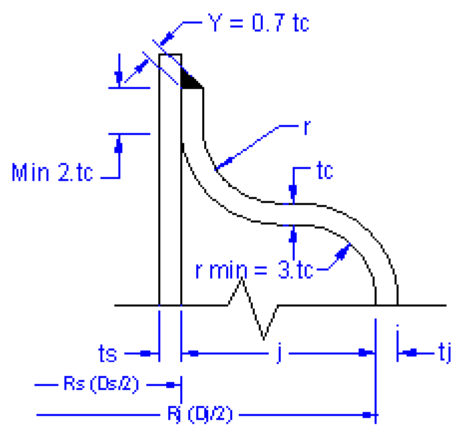


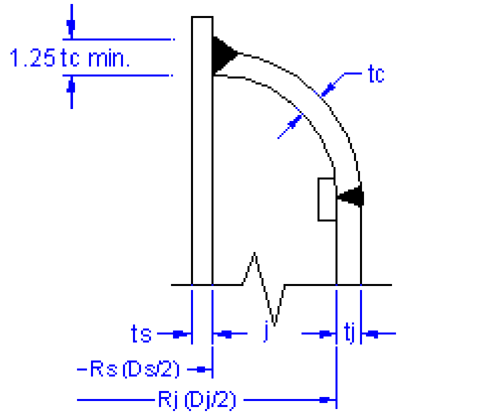
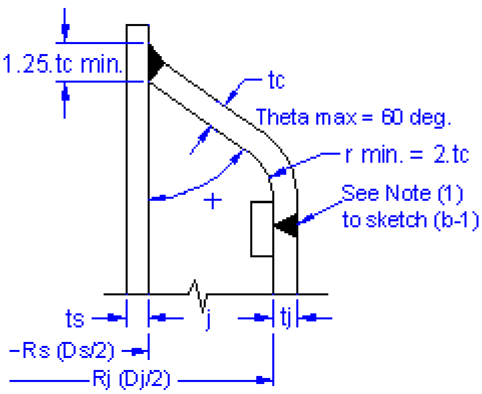
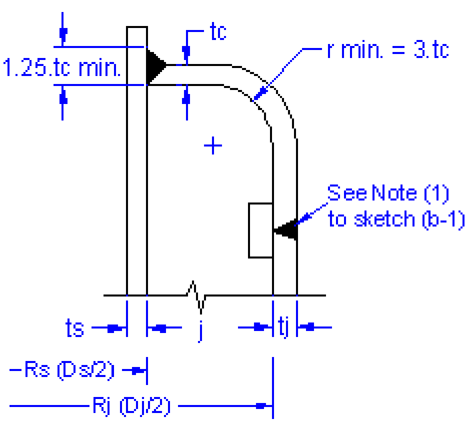
For more information, see ASME Section VIII, Division 1, paragraph UG-33, Figure UG-33.1 for half apex angles on typical geometries.

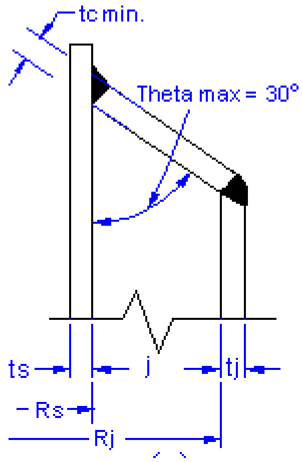
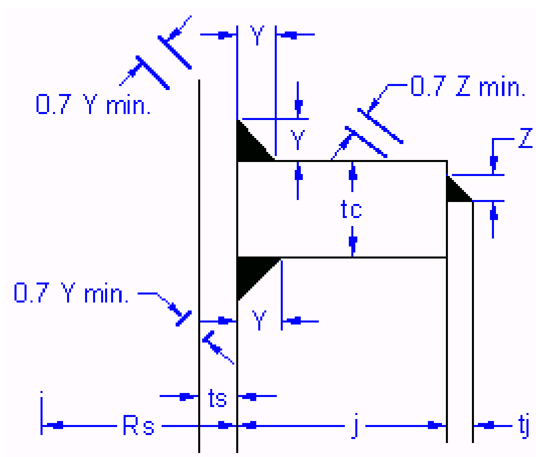
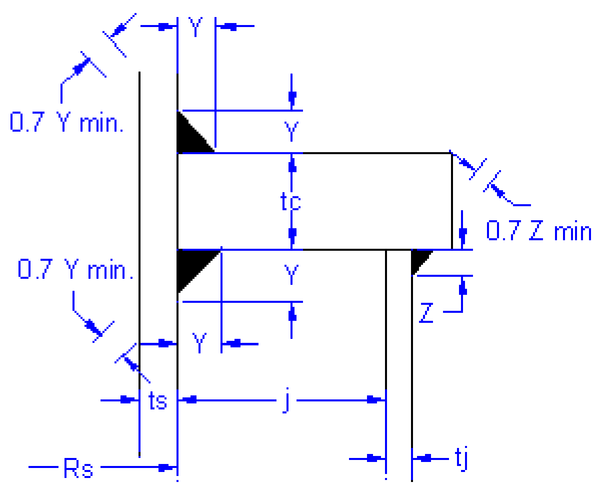
NOTE For most internal pressure calculations, the half apex angle should not be greater than 30º, though the software calculates results for up to 60º. For external pressure calculations, the angle must not be greater than 60º.

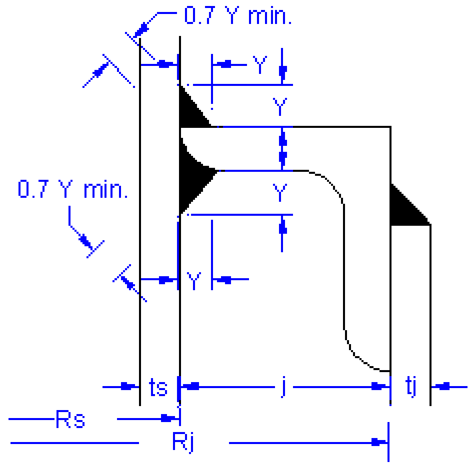
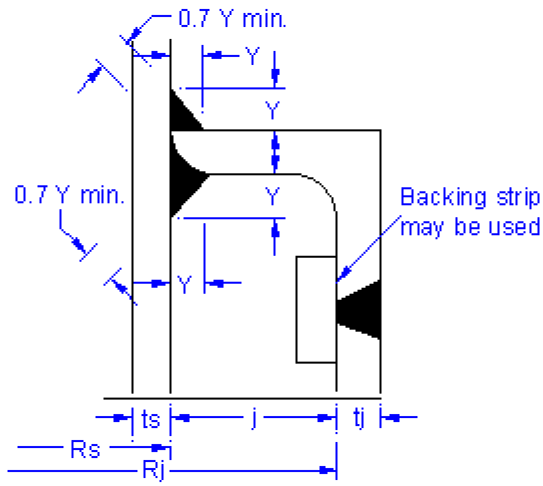
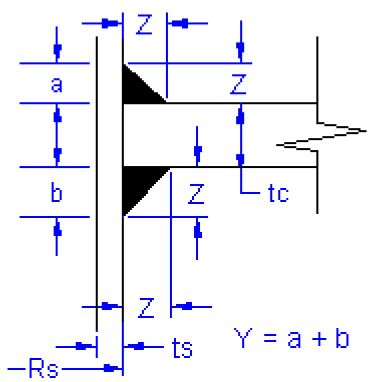
Select Closure Type (Fig. 9-5) - Select one of the following closure types, according to Fig. 9-5 of Appendix 9:

Type (a)9

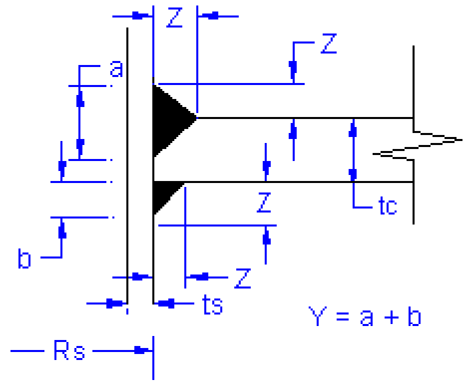


Type (b-1)	
Type (b-2)	
Type (b-3)	

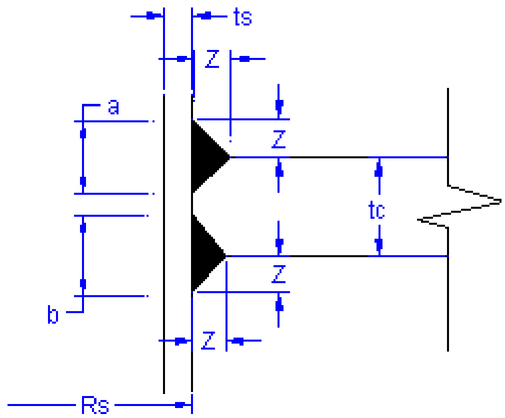
Type (c)	
Type (d-1)	
Type (d-2)	

Type (e-1)	
Type (e-2)	
Type (f-1)	

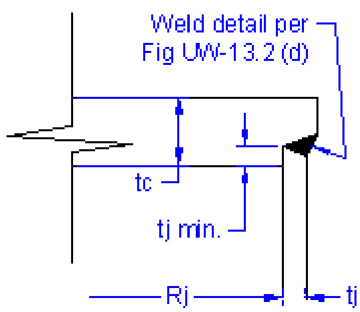
Type (f-2)



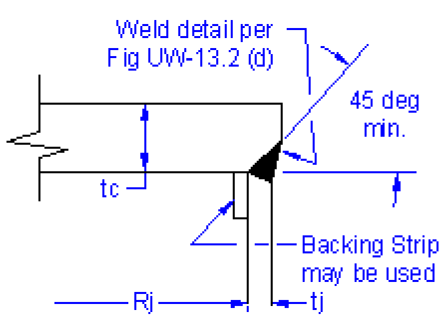
Type (f-3)



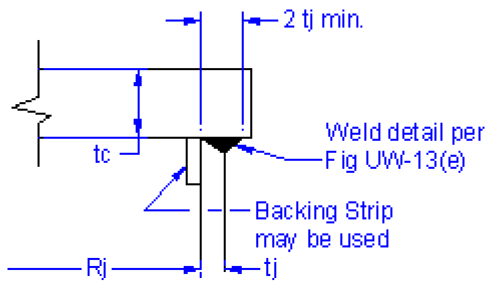
Type (g-1)



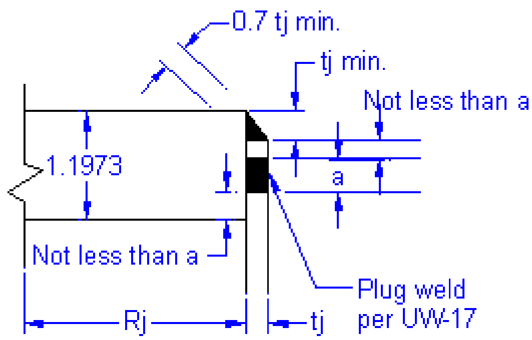
Type (g-2)



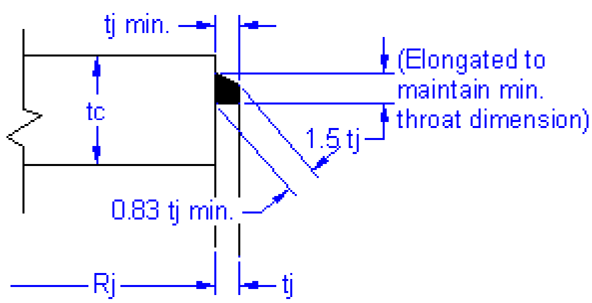
Type (g-3)



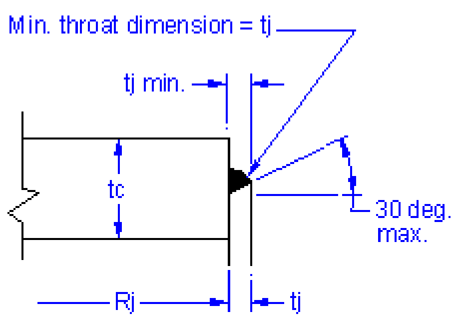
Type (g-4)



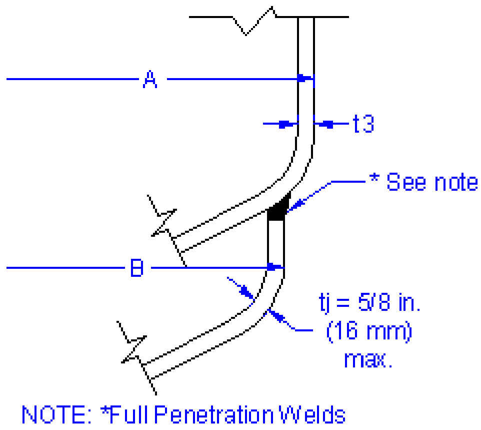
Type (g-5)



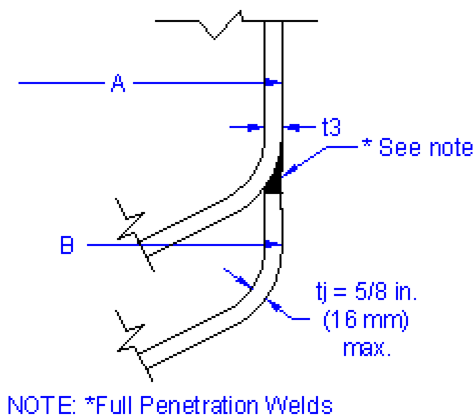
Type (g-6)



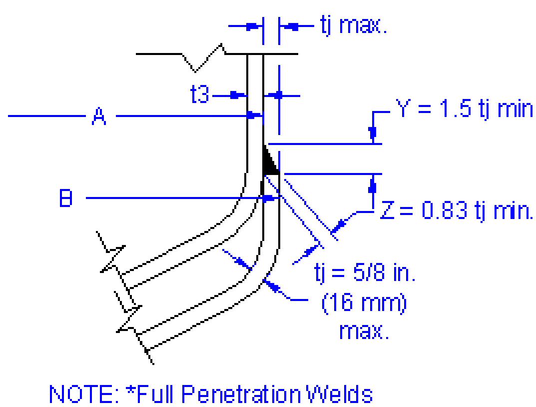
Type (h, i-1(a))



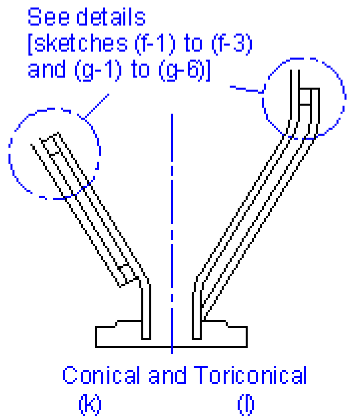
Type (h, i-1(b))



Type (h, i-2)



Type (k)



Closure Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Closure Thickness - Thickness of the plate used for the closure.

Total Corrosion Allowance - Enter the thickness of the corrosion allowance for the closure.

API-579 Flaw/Damage Input/Analysis

Home tab: **Details** > **API-579 Flaw/Damage Input/Analysis**

Adds API-579 Flaw/Damage Input/Analysis information on the selected element.

Previous - Select to view the previous flawed defined on the selected element.

Add New Flaw - Select to define another flaw on the selected element.

Delete - Select to delete the current flaw from the element.

From Node - Displays the **From Node** for the selected element.

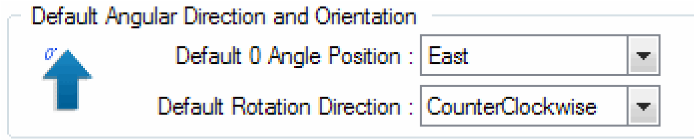
Distance from 'From' Node - Enter the distance between the **From Node** and the flaw that you are defining.

New Damage Description - Type a description for the damage.

Layout Angle - Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the weight at the point where it is attached to the shell.

For a horizontal vessel, the zero degree line is at 12:00 (looking at a clock); 90 degrees is at 3:00. Entering these layout angles is important if the horizontal vessel has a liquid level and the nozzles are being designed using **Design P + Static Head**, selected for **Nozzle/Clip Design Pressure Options** on the **Load Cases** tab. For a vertical vessel, the angle is more arbitrary. For purposes of rendering the graphics, the assumption is that the zero degree line is at 3:00 and 90 degrees is 12:00.

The position of the zero degree reference direction (North, West or East) and the angular rotation (clockwise or counterclockwise) are set in **Tools** tab, **Set Configuration Parameters**, *Set Default Values Tab (Configuration Dialog)* (see "*Set Default Values Tab (Configuration Tab)*" on page 207).



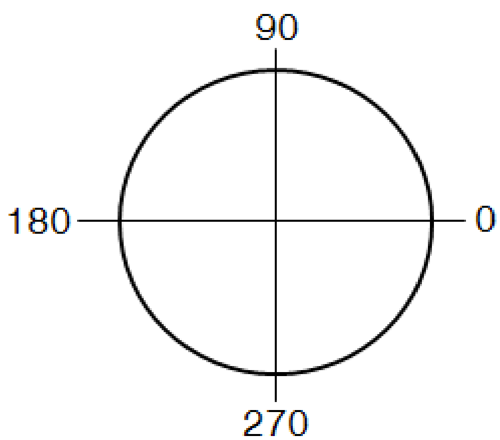
If the new orientation is different from the orientation of the current model, the software asks to update the model with the new settings. If the settings are updated, the weights and areas of platforms must be recalculated:

- Click **List Dialog** (on page 193). On the **Detail Listing** dialog box, click the **Platform** tab so that platform data is visible. Click **Platform Wind Area** and **Platform Weight** to recalculate.
- Click **Platform Input** for each platform. Tab through the fields in the **Platform** dialog box, causing the weight and area to be recalculated

NOTE The angular settings apply to nozzles, clips, legs, lifting lugs, support lugs, base rings, platforms, weights and half-pipe jackets.

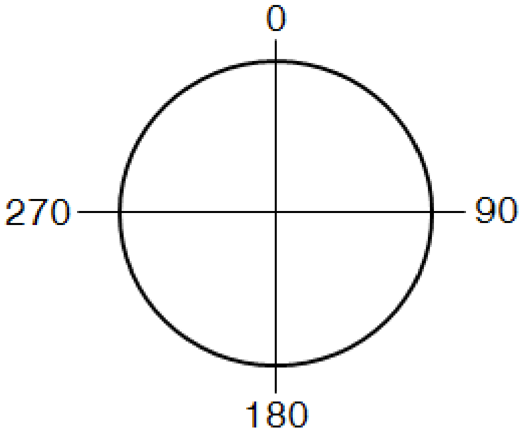
Examples

Default Orientation



Vertical Vessel (Top or Plan View)
Horizontal Vessel (Left End View)

Angle Position: North
Rotation Direction: Clockwise



Vertical Vessel (Top or Plan View)
Horizontal Vessel (Left End View)

Flaw Type - Select the type of damage observed from the following flaw types.

- **Part 4 (General Metal Loss)** - Select this option when the general metal loss (uniform or local, inside or outside) exceeds, or is predicted to exceed, the corrosion allowance before the next scheduled inspection. Assessments are based on point thickness readings and thickness profiles (see paragraph 4.3.3), whether the metal loss is uniform or local, and the degree of conservatism acceptable for the assessment. You can use the methodology shown in Figure 4.2 to determine the assessment procedure for the evaluation.
- **Part 5 (Local Thinning Area)** - Select this option when the metal loss on the surface of the component is localized, and the length metal loss is about the same as the width.
- **Part 5 (Groove like Flaw)** - Select this option when the flaw is either:
  - **Groove** – A local, elongated, thin-spot caused by directional erosion or corrosion. The length of the metal loss is significantly greater than the width.

    NOTE A sharp radius might be present at the base of a groove-like flaw.
  - **Gouge** – A local, elongated, mechanical removal or relocation of material from the component surface resulting is a reduction in wall thickness at the flaw. The length of the gouge is much greater than the width. The material might have been cold worked in the formation of the flaw. Gouges are typically caused by mechanical damage, for example, denting and gouging of a section of pipe by mechanical equipment during the excavation of a pipeline. Gouges are frequently associated with dents due to the nature of mechanical damage. If a gouge is present, the assessment procedures of Part 12 shall be used.
- **Part 6 (Pitting)** - Select this option when one of the four types of pitting is present:
  - widely scattered pitting over a significant region of the component,
  - a local thin area (LTA) located in a region of widely scattered pitting,
  - localized regions of pitting, and

- pitting confined within a region of an LTA.

A flowchart shown in Figure 6.2 provides details of the required assessment procedures. Depending on the type of pitting damage, you must use for evaluation either assessment methods in Part 6 or a combination of assessment methods in Part 5 and Part 6.

Assessment Level - Select the assessment level. See the description below that corresponds to your selection for Flaw Type.

Part 4 (General Metal Loss)

Select **Level 1 and Level 2** if these four points are true. Otherwise select **Level 1 only** or **Level 2 only**.

- The original design criteria was in accordance with a recognized code or standard (see Part 1, paragraphs 1.2.2 or 1.2.3).
- The metal loss region has relatively smooth contours without notches (that is, negligible local stress concentrations).
- The component is not in cyclic service.

  TIP A component is not in cyclic service if the component is subjected to less than 150 cycles throughout its previous operating history and future planned operation, or the component satisfies the cyclic service screening procedure in Annex B1 paragraph B1.5.2. A cycle is defined as pressure or temperature variations including operational changes and start-ups and shut-downs.
- The following limitations on component types and applied loads are satisfied:
  - Level 1 Assessment – Type A Components subject to internal pressure or external pressure (that is, supplemental loads are assumed to be negligible).
  - Level 2 Assessment – Type A or B Components (see Part 4, paragraph 4.2.5) subject to internal pressure, external pressure, supplemental loads (see Annex A, paragraph A.2.7), or any combination thereof.

Part 5 (Local Metal Loss)

Select **Level 1 and Level 2** if these five points are true. Otherwise select **Level 1 only** or **Level 2 only**.

- The original design criteria were in accordance with a recognized code or standard (see Part 1, paragraphs 1.2.2 or 1.2.3).
- The material is considered to have sufficient material toughness. If there is uncertainty regarding the material toughness, then a Part 3 assessment should be performed. If the component is subject to embrittlement during operation due to temperature or the process environment, a Level 3 assessment should be performed. Temperature or process conditions that result in material embrittlement are discussed in Annex G.
- The component is not in cyclic service.

  TIP A component is not in cyclic service if the component is subjected to less than 150 cycles throughout its previous operating history and future planned operation, or the component satisfies the cyclic service screening procedure in Annex B1 paragraph B1.5.2. A cycle is defined as pressure or temperature variations including operational changes and start-ups and shut-downs.
- The following limitations on component types and applied loads are satisfied:

  - Level 1 Assessment – Type A Components subject to internal pressure or external pressure (that is, supplemental loads are assumed to be negligible).
  - Level 2 Assessment – Type A or B Components (see Part 4, paragraph 4.2.5) subject to internal pressure, external pressure, supplemental loads (see Annex A, paragraph A.2.7), or any combination thereof.
- A flaw characterized as a groove in accordance with paragraph 5.2.1.b has a groove radius that satisfies the requirements in paragraph 5.4.2.2.f.

Part 6 (Pitting)

Select **Level 1 and Level 2** if these six points are true. Otherwise select **Level 1 only** or **Level 2 only**.

- The original design criteria were in accordance with a recognized code or standard (see Part 1, paragraphs 1.2.2 or 1.2.3).
- The material is considered to have sufficient material toughness. If there is uncertainty regarding the material toughness, then a Part 3 assessment should be performed. If the component is subject to embrittlement during operation due to temperature and/or the process environment, a Level 3 assessment should be performed. Temperature and/or process conditions that result in material embrittlement are discussed in Annex G.
- The component is not in cyclic service.

  TIP A component is not in cyclic service if the component is subjected to less than 150 cycles throughout its previous operating history and future planned operation, or the component satisfies the cyclic service screening procedure in Annex B1 paragraph B1.5.2. A cycle is defined as pressure or temperature variations including operational changes and start-ups and shut-downs.
- The following limitations on component types and applied loads are satisfied:
  - Level 1 Assessment – Type A Components subject to internal pressure or external pressure (that is, supplemental loads are assumed to be negligible).
  - Level 2 Assessment – Type A or B Components (see Part 4, paragraph 4.2.5) subject to internal pressure, external pressure, supplemental loads (see Annex A, paragraph A.2.7), or any combination thereof.
- Additional requirements for Level 1 Assessments are:
  - The pitting damage is arrested.
  - The pitting damage is located on only one surface (either inside or outside) of the component.
  - The pitting damage is composed of many pits; individual pits or isolated pares of pits should be evaluated using the assessment procedures in Part 5.
- Additional requirements for Level 2 Assessments are:
  - The pitting damage is characterized by localized regions of pitting, an LTA located in a region of widely scattered pitting, or pitting that is confined within an LTA.
  - The pitting damage is located on either one surface or both surfaces of the component and the pitting damage is not overlapping (see Figure 6.15)
  - The pitting damage is composed of many pits; individual pits or isolated pairs of pits should be evaluated as LTAs using the assessment procedures in Part 5.

A **Level 2 only** assessment should be performed if:

- An appropriate pit comparison chart cannot be found (see paragraph 6.3.3.1).
- A more detailed assessment of widespread pitting (inclusion of the pit-couple orientation) is required.

Future Corrosion Allowance (FCA) - Enter the future corrosion allowance (the projected future metal loss). The future corrosion allowance should be based on past inspection information or corrosion rate data relative to the component material in a similar environment. Corrosion rate data can be obtained from API Publication 581 or other sources (see paragraph A.2.7 of Annex A). The FCA is calculated by multiplying the anticipated corrosion rate by the future service period considering inspection interval requirements of the applicable inspection code.

Note for Part 6 – Per paragraph 6.2.7, The future corrosion allowance (FCA) shall be based on the projected future metal loss in the pitting region. The FCA is not applied to the depth or diameter of the pits.

Remaining Strength Factor (RSFa) - Enter the allowable remaining strength factor (RSFa). The recommended RSFa for all major design codes per the 2007 edition of API 579 is 0.90. See Table 2.3.

Point Thickness Readings (Part 4 only) - Select this option to use point thickness readings in the assessment. This option is only available when Flaw Type is set to **Part 4 (General Metal Loss)**.

If you select this option, you are confirming the assumption of uniform metal loss. Point thickness readings may be used to characterize the metal loss in a component if there are no significant differences in the thickness reading values obtained at inspection monitoring locations.

A minimum of 15 thickness readings should be used unless the level of NDE used can confirm that the metal loss is general. In some cases, additional readings might be required based on the size of the component, the construction details used, and the nature of the environment resulting in the metal loss. A sample data sheet to record thickness readings is shown in Table 4.2.

If the Coefficient of Variation (COV) of the thickness readings is greater than 10%, then thickness profiles shall be considered for use in the assessment (see paragraph 4.3.3.3). The COV is defined as the standard deviation divided by the average.

Critical Thickness Profile - Select this option to use thickness profiles to characterize the metal loss in a component. You must use this option if there is a significant variation in the thickness readings, which can indicate the metal loss might be localized. Thickness profiles should be used to character the remaining thickness and size of the region of metal loss.

Pitting Data (Part 6 Only) - Select this option to consider pitting data. This option is only available when Flaw Type is set to **Part 6 (Pitting)**.

- **Level 1 Assessment** - The surface damage is measured in terms of pitted area, and the maximum pit depth is used to quantify the extent of pitting damage. You can use the standard pit charts (see Figures 6.3 to 6.10) to compare the actual damage on the component to the damage represented on the pit chart. The pit chart and an estimate of the maximum pit depth are used to determine acceptability. A cross sectional UT thickness scan can determine the pitting profile. Guidelines for determining the maximum pit depth are in paragraph 6.3.4.1 of the code.
- **Level 2 Assessment** - The surface damage is measured using a pit-couple (two pits separated by a solid ligament, see Figure 6.11). The metal loss of each pit in a pit-couple is

modeled as an equivalent cylinder. The diameter and depth of each pit, and the distance between the pit centers are also required. The orientation of the pit-couple in the biaxial stress field can also be included in the assessment (see Figure 6.11). The depth and diameter of a pit should be carefully measured because of the variety of pit types that can occur in service (see Figure 6.12). If the pit has an irregular shape, a diameter and depth that encompasses the entire shape should be used in the assessment.

Edit Measurement Data - Activates the API 579 Point Thickness Readings dialog box. Use this dialog to enter thickness readings.

Thickness Approach (Parts 4 and 5 only) - Select this option to calculate the remaining life of a component based on a minimum required thickness for the intended service conditions, thickness measurements from an inspection, and an estimate of the anticipated corrosion rate. This method is acceptable for calculating the remaining life for Type A Components (see paragraph 4.2.5). Because the thickness-based approach (this option) might produce non-conservative results for the remaining life when applied to Type B or Type C Components (see paragraph 4.2.5), you need to use the MAWP Approach for those components.

MAWP Approach - Select this option to calculate the remaining life of Type A, B, and C components using a systematic method. You must select this option for Type B and C components. This approach also ensures that the design pressure is not exceeded during normal operation if the future corrosion rate is accurately established.

Re-rate MAWP - Select this option to re-rate the maximum allowable working pressure. By increasing or decreasing the process temperature, pressure, or both and if the degradation mode is temperature or pressure sensitive, a process change might minimize the progression of the damage. However, the component must be evaluated so that the design still meets the changed conditions.

NOTE A reduction in the pressure or temperature might result in a reduction of the minimum required wall thickness, thereby increasing the lifespan of the component.

Additional Level 5 Assessment (CTPs only) - Select this option if the component does not meet the Level 1 Assessment requirements and you want the metal loss region evaluated using the Part 5 Assessment procedures for local metal loss using the same thickness profile. This option is not applicable for point thickness readings.

Previous Average Measured Thickness [tam_prev] - Enter the average measured thickness from the previous inspection. The software uses this value in the Remaining Life calculations.

Corrosion Rate [Crate] - Enter the future corrosion rate determined from previous thickness data, corrosion design curves, or experience in similar services.

Pit Depth Corrosion Rate [PPRpit-depth] - Enter the estimated rate of change of pit characteristic depth. Pits may corrode in many forms and sometimes the pit depth is not impacted by corrosion. If you leave this input at 0, the software does not use a pit depth corrosion rate in the remaining life pitting calculations. However, if corrosion affects both the pit diameter and the depth and you specify those values, the software takes both propagation rates into consideration for the remaining life calculation.

Pit Diameter Corrosion Rate [PPRpit-dia] - Enter the rate of change of the pit characteristic diameter. Pits may corrode in many forms, and sometimes the pit diameter is not impacted by corrosion. If you leave this input at 0, the software does not use the pit diameter corrosion rate in the remaining life pitting calculations. However, if corrosion affects both the pit diameter and the depth and you specify those values, the software takes both propagation rates into consideration for the remaining life calculation.

Overriding values - Activates the API Overriding Values dialog box. Use this dialog to specify settings for the corrosion allowance on remaining life, design condition specifics, and overriding force and moment details.

API Overriding Values Dialog Box **on page 189****Orientation w/respect to Long. Axis (beta)** - Enter the angle to the groove-like flaw from the longitudinal axis.

Length of Groove-like Flaw (gl) - Enter the length of the flaw.

Width of Groove-like Flaw (gw) - Enter the width of the flaw.

Radius at Base of Groove-like law (gr) - Enter the radius at the base of the flaw.

Local Thinning Area Longitudinal Dimension (s) - Enter the longitudinal extent or length of the region of local metal loss at the time of the inspection.

Local Thinning Area Circumferential Dimension (c) - Enter the circumferential extent or length of the region of local metal loss at the time of the inspection.

Distance to nearest Major Structural Discontinuity [Lmsd] - Enter the distance to the nearest major structural discontinuity. This box indicates that the software checks the limiting flaw size criteria.

Uniform Metal Loss [LOSS] - Specify the amount of uniform metal loss away from the local metal loss location at the time of the assessment. The software uses this box determine a suitable wall thickness to use in the assessment.

Maximum Pit Depth (wmax) - Enter the pit depth.

Widespread Pitting - Select this option when the pitting is widespread across the element.

Localized Pitting - Select this option if the pitting is localized.

LTA in Region of Widespread Pitting - Select this option when the region of local metal loss (LTA) is located in an area of widespread pitting. When you select this option, the software determines a combined Remaining Strength Factor uses that value to assess the damage.

Pitting Confined in Local Thin Area - Select this option when the pitting damage is confined within the local thin area.

API Overriding Values Dialog Box

Click **Overriding Values** on the **API-579 Flaw/Damage Input/Analysis** dialog box to activate the API Overriding Values dialog box. Use this dialog to specify overriding values information for the selected element in areas such as the corrosion allowance on remaining life, design condition specifics, and overriding force and moment details.

Input/Output Panel

The following commands are available on the **Input/Output** panel on the **Home** tab.

	Input - Shows and hides the data input tabs located in the bottom-left of the interface. For more information, see *Input* (on page 190).
	Component Analysis - Activates CodeCalc. For more information, see *Component Analysis* (on page 190).
	Review Database - Opens the output database in Microsoft Access. For more information, see *Review Database* (on page 190).

Input

Home tab: **Input / Output > Input**

Activates and hides the input tabs that appear on the bottom-left corner of the interface.

General Input - Enter data for a vessel element. For more information, see *General Input Tab* (on page 243).

Report Headings - Enter page heading text and cover sheet text for reports. For more information, see *Report Headings (Heading Tab)* (on page 323).

Design/Analysis Constraints - Enter data such as pressures and temperatures, hydrotest data, and wall thicknesses. For more information, see *Design Constraints Tab* (on page 325).

Load Cases - Enter stress combination and nozzle pressure load cases. For more information, see *Load Cases (Load Cases Tab)* (see "*Load Cases Tab*" on page 335).

Wind Loads - Select a wind design code and enter data required by that code. For more information, see *Wind Loads (Wind Data Tab)* (on page 345).

Seismic Loads - Select a seismic design code and enter data required by that code. For more information, see *Seismic Loads (Seismic Data Tab)* (on page 393).

Component Analysis

Home tab: **Input / Output > Component Analysis**

Opens CodeCalc and perform analyses separate from the PV Elite model. You can perform analyses such as local loads on nozzles and tubesheet calculations. For more information, see CodeCalc Help.

Review Database

Home tab: **Input / Output > Review Database**

Opens the database in Microsoft Access so you can review the contents. You need to create the database first using **Create Database** in the **Auxiliary** panel. For more information, see *Create Database* (on page 195).

Utilities Panel

Provides miscellaneous edit commands for functions such as insert, delete, update, and flip to edit elements. The following commands are available on the **Utilities** panel on the **Home** tab.

	Insert Element - Insert an element before or after the current element.
	Delete Element - Delete the currently selected element.
	Propagate Element Diameter - Propagate element diameter to connected elements.
	Share Information - Share information between elements.
	Flip Element Orientation - Flips the orientation of the current element. Use this command when you want to change the orientation of just a single element. Use **Flip Model Orientation** on the **Tools** tab to flip the entire model.
	Select Material - Select a material from the materials database.
	Zoom Mode - Zoom in and out of elements in 2D.
	View Element - View 2D element plan or layout view.
	Compute Ligament Efficiencies - Calculates ASME VIII-1 UG-53.x or 4.10.x ligament efficiencies for tube spacing. For more information, see *Compute Ligament Efficiencies* (on page 226).

Flip Element Orientation

Home tab: **Utility** > **Flip Element Orientation**

Vessels are defined one element to the next (from bottom-to-top for vertical vessels and from left-to-right for horizontal vessels). If the vessel begins with a skirt element, it is a vertical vessel. Vertical vessels on legs and horizontal vessels start with a head element. If that first element is improperly oriented for the vessel that you want to model, use this command to correct the orientation.

After the second element is added, the vessel can no longer be flipped between horizontal and vertical using this command. However, you can use this command later if heads, body flanges, or cone elements need to be flipped. To flip the entire model after the second element has been added, use **Flip Model Orientation** on the **Tools** tab.

Auxiliary Panel

The following commands are available from the **Auxiliary** panel on the **Home** tab.

	**Pipe properties** - Use this dialog to select piping properties for the Shell, Nozzle, or Leg from the currently selected piping database. Please note that the **Deduct Mill Tolerance from Thickness?** has no effect on nozzles or pipe legs. For more information, see *Seamless Pipe Selection Dialog Box* (on page 193).
	**List dialog** - Opens the Detail Listing dialog box. For more information, see *List Dialog* (on page 193).
	**Write Foundation 3D file** - Selecting this function creates a Foundation 3D file after the model is analyzed. For more information, see www.dimsoln.com.
	**Export to DXF file** - Exports the vessel geometry to a Release 12 Data Exchange File (CAD file). For more information, see *Export to DXF File* (see "*Setting Up the Required Parameters*" on page 194).
	**Rigging Results** - View rigging results.
	**Create Database** - Create database of input files. For more information, see *Create Database* (on page 195).
	**Element Properties** - Display list of element weights, volumes, and surface areas.
	**Set Configuration Parameters** - Set configuration parameters. For more information, see *Configuration* (on page 200).
	**Create/Review Units** - Create or review unit files. For more information, see Create/Review Units.
	**Calculator** - Opens the Windows calculator.
	**Switch Datum Input** - Enable/Disable nozzle data entry from the datum line.

Seamless Pipe Selection Dialog Box

Pipe Schedule - Select the pipe schedule from the drop-down menu.

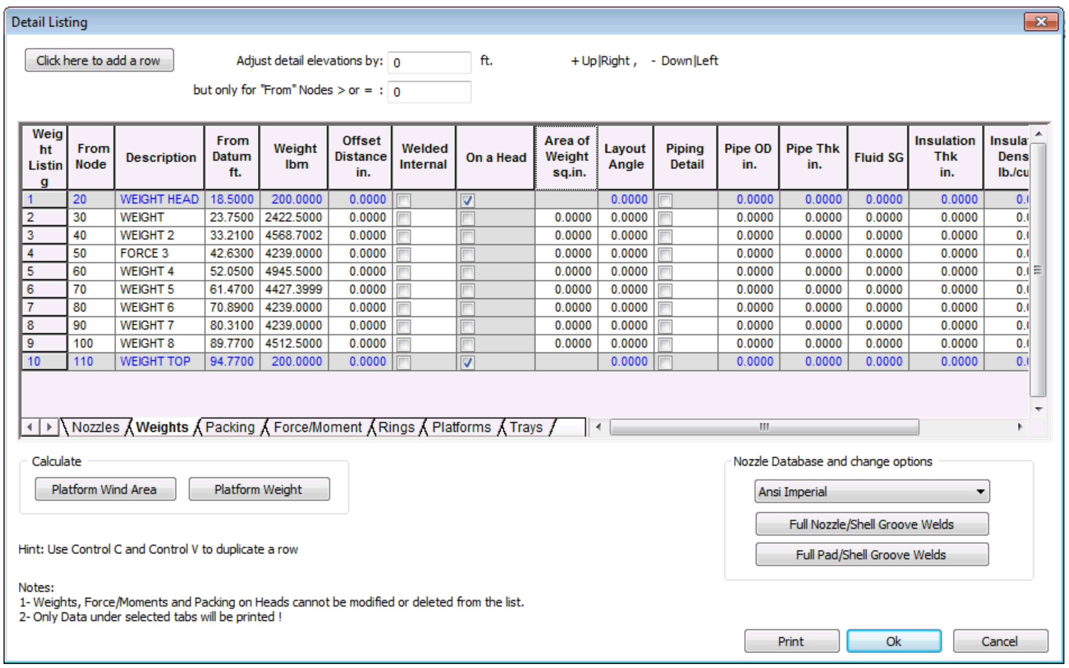
Nominal Pipe Diameter - This value automatically gets populated once the pipe schedule is selected. This value can then be modified if needed.

Deduct Mill Tolerance from Thickness? - Check this box if you want to deduct the mill tolerance from the thickness.

List Dialog

Home tab: **Auxiliary > List Dialog**

Opens the **Detail Listing** dialog box, where you add details for platforms, nozzles, weights, packing, forces/moments, trays, and pressure rings.



The location of the detail can be specified from the datum position. Select the type of detail to edit by clicking its tab. Enter the needed data for each row. Press **+** to add a row.

The software automatically assigns the value for **From Node**.

IMPORTANT **Description** is required. If you do not enter one, the software ignores the row and the data is lost.

All other data must be entered as required.

Rows of data can be duplicated from one row to the next. Click on the listing number of the item to copy to highlight the row. Copy the row and paste it to a blank row. Change any data that might be different for that detail.

Adjust detail elevations by - Enter a value to shift the position of all details by the specified elevation distance. A negative value will move details down or left. A positive value moves the details up or right. This option is useful when all of the details such as rings, nozzles, and trays need to be adjusted by a specified amount. This may happen if an element is inserted into the model after it has been completed and the detail elevations need to be kept constant.

CAUTION If the adjustment moves a detail (such as a tray) into an element (such as a body flange), the software does not allow this, and the detail is lost and cannot be recovered.

But only for "From" nodes > or = - Enter the **From Node** number where you would like the change in position to start. All details on this element and the following elements are affected. A value of zero affects all elements.

Setting Up the Required Parameters

Home tab: **Auxiliary > Export to DXF File**

Instructs **PV Elite** to generate DXF files during an analysis run. Optionally, you can use **File > Import/Export > Export Vessel Geometry to R12 DXF File** to set this option. If the scale factor is not set, the **DXF Options** dialog appears prompting for the scale factor and any other necessary options. These options should be entered after the vessel has been completely modeled because the scaling factor is based on the overall height and length of the vessel. It is best to check the scaling factor at the conclusion of the data input and before the model is analyzed.

Create a Default Border - Select to put a border around the drawing. The border style differs based on the border size. You can create your own border styles. The borders are located in the **PV Elite\System** folder. They are named ANSI_A.txt and so forth. These text files are essentially the core of ACAD Release 12 DXF files. See the user border creation section for more information.

Create a Nozzle Schedule - Select to create a Nozzle Schedule. The nozzle schedule contains information pertaining to the size and thickness of nozzles, their mark number and the necessity of reinforcing.

Create a Bill of Material - Select to generate a Bill of Material for the major components of the vessel, such as shells, heads, and conical sections.

OD Lines Shown Only - Normally the DXF file will contain ID as well as OD lines for the major shell sections. If you do not want to see the ID lines, then check this box.

Show Dimensions - Select if you want tail dimensions for the major shell courses. The element diameters and thicknesses are shown in the BOM.

Drawing Size - Select A, B, C or D. Each size has a different style.

Scale Factor - Enter the scale factor. We recommend letting the program select this value by clicking **Compute and Insert Scale Factor**. We then recommend rounding up to the nearest typical scale factor.

User Border Creation

In order to do the following, you must use Windows Explorer, AutoCad, and Notepad.

1. Start AutoCAD and open your border. The border should be ANSI standard dimensions (8½ by 11, and so forth) scaled for the non-printable area of the paper.
2. After the border drawing is open, save it as a release 12 DXF file.
3. After the file has been saved it will be necessary to edit it with a text editor such as Notepad. Because the main drawing will have a DXF header, it will be necessary to delete the one in

the border drawing. The DXF header ends on about line 960 with the word Entities. Delete through this line.

4. Next delete the last four lines in the file. This is the end of file marker.
5. Save the file with a txt extension.
6. Next rename the file in the **PV Elite\System** folder that you will be replacing. We suggest putting a new extension on it.
7. Save/Copy your border in the **PV Elite\System** folder and then rename it replacing our default border. You should now have new ANSI_?.txt file in the **PV Elite**\System subdirectory.

Review our border drawing text files before you start. Note that the border drawings must not contain any block attributes. These are not supported in our current implementation.

Create Database

Home tab: **Auxiliary > Create Database**

Creates a Microsoft Access database that contains all the input values.

Use **Review Database** in the **Input / Output** panel to review the database. For more information, see *Review Database* (on page 190).

Analyze Panel

The following commands are available on the **Analyze** panel on the **Home** tab.

	Analyze - Analyze the vessel and produce reports and forms. For more information, see *Analyze* (on page 195).
	Error Check - Error check the vessel input. For more information, see *Error Check Only* (on page 196).
	Review - Review the analysis data output for the vessel from the last analysis. For more information, see *Review Reports* (on page 196).
	Review the DXF File - Opens the .dxf file of the model in any software installed on your computer system that is capable of viewing .dxf-formatted files. For more information, see *Review the DXF File* (see "*DXF File Generated by PV Elite During Runtime*" on page 196).

Analyze

Home tab: **Analyze > Analyze**
Keyboard: **F12**

Analyzes the current model and creates the output files. Click reports in the **Report List** to see results of the analysis. For more information about the reports, see *Review Reports* (on page 196) and *Output Processor* (on page 481).

For more information about how PV Elite performs an analysis, see *PV Elite Analysis* (on page 471).

TIP Although not required, we recommend that you run *Error Check Only* (on page 196) before you run an analysis.

Error Check Only

Home tab: **Analyze > Error Check Only**

Runs only error checking on the model without running the analysis. Select **Warnings and Errors** in the **Report List** to see results of the error check.

Error Checking

The input processor makes many data consistency checks during the input session. For example, the processor creates an error message if you try to specify a nozzle 20 feet from the bottom of a 10-foot shell element. However, not all data can be confirmed on input so a general error processor is run prior to the analysis. This error processor can be run in a stand-alone from the **Analyze** panel, *Error Check Only* (on page 196) .

In addition to the notes that are presented on the screen during error checking, these error messages appear in the output report and are accessible through the output review processor.

NOTE As with all engineering and designing, the vessel analyst must use common sense to insure the model is basically correct. This is a great advantage of the 3D graphics as it reveals obvious errors.

Review Reports

Home tab: **Analyze > Review**

Displays the results of your analysis and output that results to a Microsoft Word file or an ASME Form using Microsoft Excel. For more information about reports, see *Output Processor* (on page 481).

DXF File Generated by PV Elite During Runtime

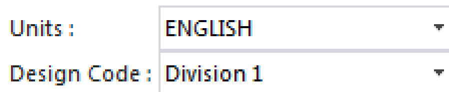
Home tab: **Analyze > Review the DXF File**

Opens the DXF file using drawing software installed on your computer that supports DXF files. If this command is available, the DXF file for this job was created during the last run. Clicking this command submits the file to Windows, which in turn launches your drawing software. If the input is altered, the analysis must be run in order to generate a new DXF file.

Use *Export to DXF File* (see "*Setting Up the Required Parameters*" on page 194)  to define the DXF settings to use.

Units/Code Panel

The following commands are available on the **Units/Code** panel on the **Home** tab.



Units - Selecting a units file converts all the properties in the model to that set of units. This can also be changed by **Tools > Select Units** (on page 207).

Design Code - Determines which design code is used for analysis.

SECTION 5

Tools Tab

The following utility commands are available on the **Tools** tab:

	Set Configuration Parameters - Set configuration options for this analysis. For more information, see *Configuration* (on page 200).
	Select Units - Select a new units file. For more information, see *Select Units* (on page 207).
	Create\Review Units - Review the current units file, or create a new units file. For more information, see *Create / Review Units* (on page 208).
	Units Conversion - Opens the **PV Elite Units Conversion Utility** that you can use to values from one unit of measure to another.
	Edit/Add Materials - Edit a materials database. For more information, see *Edit/Add Materials* (on page 210).
	Calculator (on page 226) - Open the Windows Calculator.
	Re-Number Nodes - Resequences the **From Node** and **To Node** numbers of the elements in the vessel.
	Flip Model Orientation - Flips the orientation of the entire model. Use **Flip Element Orientation** on the **Home** tab to flip just a single element.
	Enter U-1 Form Information - Create ASME U-1 form. For more information, see *Enter U-1 Form Information* (on page 226).
	Compute Ligament Efficiencies - Calculate ligament efficiencies for tube spacing. For more information, see *Compute Ligament Efficiencies* (on page 226).
	Display Driver - Select your display driver. If you have having display issues, try the other option.

Configuration

Tools tab: **Set Configuration Parameters**
Home tab: **Auxiliary > Set Configuration Parameters**

Sets job-specific parameters in the **Configuration** dialog box. Many parameters affect the analysis results. Always review the configuration at the start of a new job.

Topics

Job Specific Setup Parameters Tab (Configuration Dialog)

Print Water Volume in Gallons - Select to print out the element volume in gallons, instead of the default **Volume** cubic diameter unit. This command is independent of the selected units system.

Round Thickness to Nearest Nominal Size - Select to round thicknesses to the nearest 1/16 of an inch (if you are in English units) or the nearest 1mm (if you are in SI or MM units). The software increases the thickness of an element if you so specify in **Design/Analysis Constraints** (see "**Design Constraints Tab**" on page 325) and the element thickness is inadequate.

Print Equations and Substitutions - Select to print formulas and substitutions for internal and external pressure calculations in analysis reports.

Increase Blind Flange Thickness for Reinforcement - Select to bypass reinforcement of a single opening of a flat end connection, according to Section VIII Division 1, paragraph UG-39(d)(2). This effectively increases the required thickness of the blind flange cover. This option can only be used if there is only one nozzle located in the blind flange.

Print Flange Calcs for External Pressure - Select to always print external pressure calculations on flanges in addition to the default internal pressure calculations. When this option is *not* selected, the software does not print external pressure calculations unless the required thickness generated by the external pressure exceeds the thickness generated by the internal pressure.

Do not use the bolt correction factor for flange design - Select this option to avoid using the ASME Section VIII, Appendix 2 Equation (7) or the PD-5500 bolt space correction factors in the design of heat exchanger flanges and tubesheets, like Taylor-Forge. Using the correction factor is an industry-standard practice that is applied in other pressure vessel design codes. When the actual bolt spacing exceeds the allowable bolt spacing, the software multiplies the correction factor by the moment to design a thicker flange. In these cases, do *not* select this option.

Use ASME Code Case 2260/2261 - Select to use modified equations to calculate the required thickness of elliptical/torispherical heads, as defined in ASME Section VIII, Division 1, Code Case 2260, May 20, 1998, Alternate Design Rules for Ellipsoidal and Torispherical Formed Heads. A thinner head is typically designed with these rules.

Use Eigen Solver - Select to use an Eigen solver for natural frequency calculations.

The natural frequency of a structure can be calculated using more than one method. When this option is not selected, the traditional method is to use the analysis technique of Freese or Rayleigh-Ritz. For a skirt-supported free standing vessel, this method provides acceptable results. When the support configuration is not a skirt/base type, such as legs, lugs or intermediate skirt, this method may not provide accurate results. By default, this option is selected and the Eigen solver is used.

The natural frequency Eigen solver uses numerical methods to solve the general equation of motion. Namely, the software solves the following matrix problem:

$$[[K] - w^2[M]]\{a\} = \{0\}$$

which for the general case is a set of *n* homogeneous (right-hand side equal to zero) algebraic system of linear equations with *n* unknown displacements a_i and an unknown parameter w^2. This is known as an Eigen problem. This iterative solution, for which not all $a_i = 0$, requires that the determinant of the matrix factor of *{a}* be equal to zero, in this case:

$$abs([[K] - w^2[M]) = 0$$

After building stiffness *[K]* and mass *[M]* matrices of the model with appropriate boundary conditions (such as, anchors at skirts, bottom of legs, and at support lugs), the software can extract modes that are meaningful in the solution of the dynamics problem, particularly the modal response spectrum analysis. Using this generic-frequency Eigen solution method, the software can accurately extract modes of vibration for models that do not fit neatly into the cantilever beam model required for the Freese integration method. The natural frequency of the vessel is used in several of the wind and seismic codes.

NOTE If the selected earthquake code uses response spectrum, the software automatically uses the Eigen Solver, even if this option is not selected.

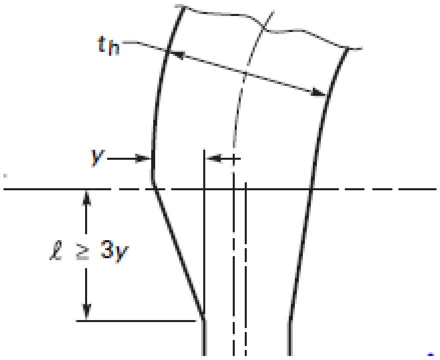
Use Pre-99 Addenda (Division 1 only) - Select if you are re-rating an older vessel to the pre-99 ASME addenda, and would like to use the older material allowables. As of January 2000, the 1999 addenda of ASME Division 1, Section 2, Part D is mandatory. The revision includes changes to the material properties of many materials, such as increases of allowable stresses in some ranges. By default, this option is not selected and the higher allowable stress database is used.

IMPORTANT Select this option before any vessel modeling occurs. For an existing file, you must access the material database for each existing element to update material properties. Other design codes are not affected.

Use 2004 A-06 Addenda for Division 2 - Select if the model is ASME Section VIII< Division 2, to use the older code rules in the analysis.

IMPORTANT Select *prior* to entering any vessel data.

For ASME VIII, Shell Head Joints are Tapered - Select if the shell to head junction is tapered as shown in the figure (ASME VIII-1 Figure UW-13.1).



For ASME VIII-1, Use Table G instead of exact equation for "A" - PV Elite can determine the strain factor 'A' used in ASME VIII-1 for external pressure calculations by either:

- Formula
- Table G in Section II Part D

Both selections yield a nearly identical result in most cases. However, PV Elite runs faster when using the formula option. The default is to use the data from Table G.

No MDMT Calculations - Select to use the MDMT check in the analysis.

No MAWP Calculations - Select to use the MAWP check in the analysis.

Use Bolt Load Instead of Bolt Area Times Bolt Stress - Select to use the calculated value of the bolt load instead of the bolt area times its allowable stress in the design of annular baserings. This leads to a less conservative basering/bolting/chair cap thickness calculation.

Use Flange Bolt Stress Ratio for ASME VIII-1 Hydro - Select to allow specific control of the use of the bolt stress ratio for calculated custom (Appendix 2) flanges. The software considers the stress ratio in bolts of flanges that are designed according to Appendix 2 for hydrotests UG-99 and UG-100. ASME Interpretation VIII-1-83-260 allows this. Flanges that are not calculated (ANSI flanges) are not allowed this exemption.

Syntax Highlighting in Output Reports - Select to highlight failures and problem areas in data reports.

No Extended ASCII Characters in Output - Select to replace extended ASCII characters with multiple characters of the same value in equations. For example, ½ is replaced with 1/2. Some non-English versions of Windows do not display these characters correctly.

Metric Output is in Consistent Units - Select to allow the software to change units so that they are displayed consistently. For example, units of stress may be displayed in MPa and pressure in Bar. For coherent output these units should be the same.

Use Code Case 2286

Select to use Code Case 2286-1, a set of alternative rules for determining allowable external pressure and compressive stresses for cylinders, cones, spheres, and formed heads. This code case is applicable for ASME Section VIII Divisions 1 and 2. In the current ASME code rules, there are no provisions for computation of allowables when external pressure acts with bending and other compressive loads simultaneously.

These rules were first published in the late 1990's and reaffirmed on January 1, 2004. Review the code case before selecting this option. The following sections are covered in PV Elite:

2.2 Stress Reduction Factors

3.1 External Pressure

3.2 Uniform Axial Compression

3.2.1 Local Buckling

3.2.2 Column Buckling

3.3 Axial Compression due to Bending Moment

3.4 Shear (Allowable in Plane Shear Stress)

4.1.1 Allowable Circumferential Compression Stresses for Cones

4.1.3 Cone-Cylinder Junction Rings

4.2.1 Allowable Longitudinal and Bending Stresses for Cones

4.2.2 Unstiffened Cone-Cylinder Junctions

4.2.3 Cone-Cylinder Junction Rings

4.3.1 Allowable in Plane Shear Stress for Cones

5.1.1 Allowables for Uniform Axial Compression and Hoop Compression

5.1.2 Allowables for Uniform Axial Compression and Hoop Compression (Column Buckling)

5.2 Allowables for Axial Compression due to Bending Moment and Hoop Compression

5.3 Allowables for Hoop Compression and Shear

5.4 Allowables for Uniform Axial Compression, Axial Compression due to Bending Moment, Shear in the presence of Hoop Compression

5.4.1 Local Buckling (Section 5.4)

5.4.2 Column Buckling (Section 5.4)

5.5 Allowables for Uniform Axial Compression, Axial Compression due to Bending Moment, Shear in the absence of Hoop Compression

5.5.1 Local Buckling (Section 5.5)

5.5.2 Column Buckling (Section 5.5)

6.0 Sizing of Rings

6.1 Sizing of Small Rings

6.4 Local Stiffener Geometry Requirements

8.1.1 Spherical Shell with Equal Biaxial Stresses

Does the use of 2286 guarantee thinner required thicknesses and lower stresses due to combined loading?

No, it does not. The results are a function of the geometry. However, the results for allowable external pressure are generally higher than those computed using the typical UG-28 external pressure methods.

What happens during the analysis when this option is selected?

The software follows several steps in determining values that are needed during the 2286 calculations. One of these steps is to determine the lengths, such as LB1 and LB2. These are the distances between major lines of support, such as heads, body flanges and conical sections. The software does not distinguish large rings that act as bulkheads, according to paragraph 6.1(b). For cones, **Line of Support Options** in **Vessel Data** must be set to **Both Ends a Line of Support**. The **ASME Steel Stack** option in **Design/Analysis Constraints (see "Design Constraints Tab" on page 325)** is not compatible with this option and is also ignored if both are checked.

The first calculations that appear for 2286 are in the **External Pressure Calculations** report, with the calculation of the allowable pressure at the given thickness. After completing this step, the software iterates to determine the required thickness for the given external pressure. If the element is a cylinder, the maximum length between stiffeners is calculated. These results are displayed in the summary at the end of the **External Pressure Calculations** report. Because Factor A and Factor B are not applicable, they are set to "No Calc".

After completing the external pressure calculations, the software calculates individual stresses for each stress category and summarizes them in the **Stress Due to Combined Loads** report. The allowable stresses are calculated for the combined loads (including external pressure). Unity checks according to 5.4 and 5.5 are made. The software also compares direct axial, shear and bending stresses to their respective allowables. If any of these are higher than the combined unity check, the maximum value is reported as the Unity Check.

A supplemental table in the **Stress Due to Combined Loads** report displays the results of λ_c, *L* and the calculated allowable stresses for each element for each load case. In some cases, the allowables may not be calculated, especially when there is no external pressure or when the load is only tensile. Some elements, such as welded flat heads and flanges, are not applicable and do not have any results. If this is the case, the printed stress value may read "No Calc" or may be blank.

If there any conical transitions in the model, the software calculates the necessity of junction rings and their requirements according to section 4.2.3. Cone-to-cylinder junction rings must satisfy inertia requirements from equations (4-1) and (4-5). The net area of the junction must be greater than or equal to the result of equation (4-4). The maximum distance from the cone-cylinder junction is defined in paragraph 4.2.3. The distance is defined as, "The nearest surface of the stiffening ring shall be located with a distance of T_r or 1 in., whichever is greater from the cone junction." T_r is the contact or stem width of the ring. Because the software handles arbitrary sections, the check value is one inch (25.4 mm). If the ring is farther than one inch from the junction, it is ignored.

For ASME VIII-1, Compute "K" in corroded condition - The ‘K’ value is the stress concentration factor used in determining the required thickness of Elliptical Heads. If the elliptical head has an internal corrosion allowance, the computed value of K decreases resulting in a lower required thickness. If that is what you want, select this option.

Use ASME Code Case 2695 (Div 1 allowables using Div 2 Part 4.5) - ASME Code Case 2695 allows the use of ASME VIII-2 formulas in a Division 1 design. In addition, the allowable stress values from Division 1 must also be used. Using the Code Case generally results in a more economical design, especially in the area of head thickness and nozzle reinforcement. As when using any Code Case, all of the restrictions must be understood so read the Code Case before using this option.

Nozzle Analysis Directives

No Corrosion on Inside Welds - Select if inside nozzle welds do not corrode or you do not wish to remove the corrosion allowance when computing the area. By default, the software always corrodes the inner fillet weld when calculating the area available in the inside weld. The default method is the most conservative because the area under the weld is corroded in accordance with figure UG-37 of the ASME Code. This option is not valid when using PD:5500.

Use AD-540.2 sketch b and not sketch d for normal limit (pre 2007) - Select to use sketch (b) for the calculation of the vertical nozzle thickness limit, according to ASME Section VIII, Division 2, paragraph AD-540.2. Sketch (b) shows an integral connection with a smooth radius. Sketch (d) shows a similar geometry with an alternative pad plate and fillet. By default, the software uses sketch (d) to calculate the vertical thickness limit.

Compute Increased Nozzle Thickness - Select to calculate the minimum nozzle wall to account for external loading. In many cases pressure vessels are designed and built long before the piping system is attached to them. This means that the nozzle loadings are unknown. When this option is selected, then the minimum nozzle thickness *trn* is maximum of:

```
trn = (.134, trn for internal pressure) <= Nps 18
trn = (OD/150, trn for internal pressure) > Nps 18
```

Using this requirement in addition to UG-45 provides some additional metal to work with to satisfy thermal bending stresses in the nozzle. You can also specify the minimum wall thickness of the nozzle *trn* in **Nozzle**. If you do so, that vale overrides this calculation.

NOTE These formulae are not in the ASME Code. They are used in industry.

Compute and Print Areas for Small Nozzles - Select to calculate the nozzle reinforcement areas and MAWPs of small nozzles when the requirements of code paragraph UG-36 are met. UG-36 discusses the requirement of performing area replacement calculations when small nozzles are involved. Openings in vessels not subject to rapid fluctuations in pressure do not require reinforcement other than that inherent in the construction under the following conditions:

- 3.5 inch finished opening in a shell or head 0.375 inches required thickness or less
- 2.375 inch finished opening in a shell or head greater than 0.375 inches required thickness

If your geometry meets this criteria and this option is not selected, then the nozzle reinforcement areas and MAWPs are not calculated. If you require a single nozzle to be checked, place the text **#SN** in the nozzle description to force the software to calculate the areas for the selected small nozzle.

Compute Chord Length in Hillside Direction - Select to use the chord length to calculate the included angle for hillside nozzle calculations. By default, the software uses the actual length of removed material.

Compute Areas per PD 5500 3.5.4.9 - Select to perform the pressure times area calculations according to PD 5000:2003, 3.5.4.9. The standard calculations according to design section 3 are always Calculated.

Nozzle opening MWAP is not restricted by the Shell (ASME) - When using ASME VIII-1 UG-37 for nozzle reinforcement, the MAWP of the opening is iteratively computed based on several items including UG-45, UG-37 areas, and other considerations. The MAWP of the opening is the minimum of the overall UG-37 calculation and the MAWP of the parent component to which the nozzle is attached. In some cases, you might want to know the MAWP of the junction without regard to the parent's MAWP. If that is the case, select this option.

ASME MDMT Directives

Use the MAWP to Compute the MDMT - Select to use the MAWP for the vessel when determining the required thickness to use in the temperature reduction calculation according to UCS 66.1 (Div. 1), or Section 3.11.2.5 (Div. 2 2007 Edition or later). The MAWP also includes MAWPs determined during nozzle calculations. If this option is not selected, the design pressure on the element is used to determine the MDMT reduction for that element.

NOTES

- If the pressure specified on each element is the MAWP, do not select this option. Otherwise, the temperature reduction is conservatively low.
- For PD 5500, EN-13445, and Division 2 pre-2007, this option is ignored.

Do Not Use Nozzle MDMT Interpretation VIII-1-01-37 - Select to control the MDMT calculation of the nozzle to shell junction, according to Section VIII, Division 1 (1998 Edition, 2000 Addenda), Figure UCS-66.l, Interpretation VIII-1-01-37, March 9, 2001. The interpretation states that if a nozzle neck with a nominal noncorroded thickness that is heavier than that of the shell is attached to the shell with a corner joint, then the shell becomes the governing thickness as defined in UCS-66(a)(1)(b). When evaluating the nozzle joint per UCS-66(b) of Section VIII, Division 1, the tr and tn thicknesses are those of the shell.

Allowable Tower Deflection - Enter a value for the allowable vertical tower deflection, if the default of 6 inches per 100 feet does not meet your design specification.

Wind Shape Factor - Enter a value for the wind shape factor, if your design specification requires a specific value that does not correspond to the software-calculated value. For cylindrical structures, the value is typically 0.7.

Ope. Nat. Freq. (Hz) Optional - Enter a value for the operating natural frequency. This is typically a value that you calculate separately when you want to use a method different than the Rayleigh method used by the software. In general, the Rayleigh method is suitable for most vessel designs, and you do not need to enter a value.

Empty Nat. Freq. (Hz) Optional - Enter a value for the empty natural frequency. This is typically a value that you calculate separately when you want to use a method different than the Rayleigh method used by the software. In general, the Rayleigh method is suitable for most vessel designs, and you do not need to enter a value.

Test Nat. Freq. (Hz) Optional - Enter a value for the test natural frequency. This is typically a value that you calculate separately when you want to use a method different than the Rayleigh method used by the software. In general, the Rayleigh method is suitable for most vessel designs, and you do not need to enter a value.

ASME VIII-1 Matl Database Year - Select the database the software uses for tables of allowable stress versus temperature. This entry is only valid for Section VIII Division 1 and is not available if **Use Pre-99 Addenda (Division 1 only)** is selected.

IMPORTANT Select this option before entering other data.

Metric Constant Selection - ASME Code Section VIII Division 1 contains a number of constants used for comparison in certain calculations. These values are presented in both Imperial and Metric units. An example would be the minimum thickness per UG-16(b) of 1/16" (1.5mm) or 3/32" (2.5mm). Note that the values in mm have been rounded. PV Elite can work with either depending on the selection made in this pull down.

- **Imperial** – Use the Imperial constants
- **Metric** – Use the Metric constants

- **Determine at runtime** – Have PV Elite determine which value to use depending on the set of currently selected units. For new files, the default is the **Determine at Runtime**.

Output Language - Select the language to use for the Input Echo report. Make this selection before you analyze the model.

DXF Options Tab (Configuration Dialog)

Create a Default Border - Select to put a border around the drawing. The border style differs based on the border size. You can create your own border styles. The borders are located in the **PV Elite\System** folder. They are named ANSI_A.txt and so forth. These text files are essentially the core of ACAD Release 12 DXF files. See the user border creation section for more information.

Create a Nozzle Schedule - Select to create a Nozzle Schedule. The nozzle schedule contains information pertaining to the size and thickness of nozzles, their mark number and the necessity of reinforcing.

Create a Bill of Material - Select to generate a Bill of Material for the major components of the vessel, such as shells, heads, and conical sections.

OD Lines Shown Only - Normally the DXF file will contain ID as well as OD lines for the major shell sections. If you do not want to see the ID lines, then check this box.

Show Dimensions - Select if you want tail dimensions for the major shell courses. The element diameters and thicknesses are shown in the BOM.

Drawing Size - Select A, B, C or D. Each size has a different style.

Scale Factor - Enter the scale factor. We recommend letting the program select this value by clicking **Compute and Insert Scale Factor**. We then recommend rounding up to the nearest typical scale factor.

Set Default Values Tab (Configuration Tab)

Sets the initial values for commonly used properties. PV Elite defaults fields using the values that you define here. You can edit the default value during creation.

Select Units

Tools tab: **Select Units**

Selects a new units file and changes the units system of the current job. For example, if your current job is in English units and you would like to change the units to millimeters, then use this option. After selecting a new units file, the current input values are converted into that set of units. To get a set of reports in the new units, run **Analyze (on page 195)** again.

Delivered units files have the .fil extension and are in the C:\Users\Public\Public Documents\Intergraph CAS\PVELITE\2013\System folder. Many unit systems are delivered, such as English, MM, SI, Inches, and Newtons. Unicode systems are delivered for use in China, Japan, Taiwan, and Korea, where multibyte character sets are used.

Create / Review Units

Tools tab: Create\Review Units

The **Create/Review Units File** utility allows you to create a new custom units file or edit an existing units file for use with PV Elite or CodeCalc. The utility is available on the **Tools** tab > **Create/Review Units** . You can also double-click **MakeUnit.exe** in the product delivery folder.

Delivered units files have the .fil extension and are in the C:\Users\Public\Public Documents\Intergraph CAS\PVELITE\2013\System folder. Many unit systems are delivered, such as English, MM, SI, Inches, and Newtons. Unicode systems are delivered for use in China, Japan, Taiwan, and Korea, where multibyte character sets are used. You can save new units files to the system folder or to another folder.

TIPS

- Use **Tools** tab > **Configuration** to specify the units file to use at startup.
- Use **Tools** tab > **Select Units** to select a new units file. The data in your job file is immediately converted to the new units.

Units File Dialog Box (on page 209)

What do you want to do?

- *Create a new units file* (on page 208)
- *Edit an existing units file* (on page 209)

Create a new units file

1. On the **Tools** tab, click **Create/Review Units** .

   *The units file dialog box displays. **Constant** has a default value of **1** for each type of unit.*

2. Do one of the following for each type of unit:

   - Select defined values for **Constant** *or* **User Unit**.
   - Type values for or **Constant** *and* **User Unit**.

3. Click **Save and Exit** .

   *The **Save As** dialog box displays.*

4. Select a folder path and type a file name.
5. Click **Save**.

   *The **Save As** dialog box and the **Units File** dialog box close.*

Edit an existing units file

1. On the **Tools** tab > click **Create/Review Units** .

   The units file dialog box displays.

2. Click **Open** .

   *The **Open** dialog box displays.*

3. Select an existing .fil units file and click **Open**.

4. Change unit types as needed by doing one of the following:

   - Select defined values for **Constant** *or* **User Unit**.
   - Type values for or **Constant** *and* **User Unit**.

5. Click **Save and Exit** .

   *The **Save As** dialog box displays.*

6. Select a folder path and type a file name. You can also use the same file name to replace the open file with the new unit values.

7. Click **Save**.

   *The **Save As** and **Units File** dialog boxes close.*

Units File Dialog Box

Specifies units and constants for a units file.

Name - Displays the type of unit, such as **Length**, **Area**, or **Pressure**.

System Unit - Displays the default system unit used as a multiplier for conversions, such as **feet**, **sq-inches**, and **psig**.

Constant - Select a defined conversion constant used as a multiplier for conversions, or type your own value.

User Unit - Select a defined unit for the conversion from the drop-down, or click in the box and type your own unit.

IMPORTANT

- If you select a defined **Constant**, the software changes **User Unit** to the correct unit. If you select a defined **User Unit**, the software changes **Constant** to the correct value.
- If you type your own value for **Constant** and **User Unit**, you must manually ensure that the combination provides the needed conversion.

Open - Open an existing units file for editing.

Save - Saves the units file.

Save and Exit - Saves the units file and closes the dialog box.

Help - Opens the help.

Tools tab: **Edit/Add Materials**

- **Edit Material**
- **Edit PD:5500 Material Database**
- **Edit EN-13445 User Material Database**

The **Material Database Editor** (on page 210) utility opens. You can edit existing material and create new material in the selected material database. The new material does not affect the current job. To change an element or detail to the new material, open the **Material Database Dialog Box** (on page 485) for each element or detail.

Material Database Editor

The **Material Database Editor** utility allows you to add custom materials to a delivered ASME, PD:5500, or EN-13445 material database for use with PV Elite or CodeCalc. For a YouTube demonstration, visit: http://www.youtube.com/watch?v=GEtIRO4PwCw. While the video is centered around PV Elite, it works much the same way in CodeCalc.

The utility is available from:

- **Tools** > **Edit/Add Materials**
- **MatEdit.exe**, found in the *[Product Folder]*\Intergraph CAS\PV Elite*[Version Number]*\ folder

When you use this utility, material database files with the .bin extension are created in the *[Product Folder]*\Intergraph CAS\PV Elite*[Version Number]*\System folder. These files contain only the custom materials you have added. The custom materials can then be merged into the main material databases.

NOTES

- The delivered databases contain allowed material for the current codes. You typically only add custom material if you are required to use an outdated material, or need to add material from a different code.
- Have the appropriate code available when adding new material. You will enter code-based material properties such as Chart Data, Material Band, and S Factor. The properties needed vary with the database that you are editing.

Material Properties (on page 212)

What do you want to do?

- *Create a new custom material* (on page 211)
- *Create a custom material based on an existing material* (on page 211)

Create a new custom material

1. Click **Tools** > **Edit/Add Materials** and select the **ASME**, **PD:5500**, or **EN-13445** material database.
2. Click **Add** .

   *A new row named **New Material** appears in the grid of the **Material Database** view in the right pane.*
3. In the **Material Properties** view in the left pane, type values for the new material.

   IMPORTANT As you type values, check the **Stress vs. Temperature** graph in the right pane. Stress must not increase as temperature decreases.
4. Repeat these steps for each new material that you want to add.
5. Click **Save** to save the new material to a user database file.
6. Click **Merge** to add the user database to the material database of the software.

NOTE After merging, the custom material now appears at the bottom of the material database list for any command using the material database in PV Elite or CodeCalc.

Create a custom material based on an existing material

1. Click **Tools** > **Edit/Add Materials** and select the **ASME**, **PD:5500**, or **EN-13445** material database.
2. Click **Edit** .

   *The contents of the software database appear in the grid of the **Material Database** view in the right pane.*
3. Select a material for the **Material Database** grid.
4. Click **Select** and click **Yes** on the confirmation dialog box.

   *The copied material appears in a new row in the grid of the **Material Database** view.*
5. In the **Material Properties** view in the left pane, type new values as needed.

   IMPORTANT

   - You must change **Material Name** so that the name is unique in the user database *and* in the material database after merging.
   - As you type values, check the **Stress vs. Temperature** graph in the right pane. Stress must not increase as temperature decreases.
6. Click **Save** to save the new material to a user database file.

7. Click **Merge** to add the user database to the material database of the software.

NOTE After merging, the custom material now appears at the bottom of the material database list for any command using the material database in PV Elite or CodeCalc.

Material Properties

The following code-based values are typically used as material properties.

Material Name - Type an allowed external pressure chart name. The software uses the chart name to calculate the B value for all external pressure and buckling calculations. If you type a valid value for **Material Name**, the software will look into its database and determine the external pressure chart name for this material and enter it into this cell. The program will also determine this chart name when you select a material name from the material selection window.

The following are the allowed external pressure chart names:

Carbon Steel

CS-1	Carbon and Low Alloy Sy<30000
CS-2	Carbon and Low Alloy Sy>30000
CS-3	Carbon and Low Alloy Sy<38000
CS-4	SA-537
CS-5	SA-508, SA-533, SA-541
CS-6	SA-562 or SA-620

Heat-Treated Steel

HT-1	SA-517 and SA-592 A, E, and F
HT-2	SA-508 Cl. 4a, SA-543,B,C

Stainless Steel (High Alloy)

HA-1	Type 304
HA-2	Type 316, 321, 347, 309, 310, 430B
HA-3	Type 304L
HA-4	Type 316L, 317L
HA-5	Alloy S31500

Non-Ferrous Material

NFA-1	AL3003, O and H112
NFA-2	AL3003, H20
NFA-3	AL3004, O and H112
NFA-4	AL3004, H34
NFA-5	AL5154, O and H112
NFA-6	C62000 (Aluminum Bronze)
NFA-7	AL1060, O
NFA-8	AL5052, O and H112
NFA-9	AL5080, O and H112
NFA-10	AL5456, O
NFA-11	AL5083, O and H112
NFA-12	AL6061, T6, T651, T6510 and T6511
NFA-13	AL6061, T4, T451, T4510 and T4511
NFA-20	AL5454, O and H112
NFC-1	Annealed Copper
NFC-2	Copper Silicon A and C
NFC-3	Annealed 90-10 Copper Nickel
NFC-4	Annealed 70-30 Copper Nickel
NFC-5	Welded Copper Iron Alloy Tube
NFC-6	SB-75 and SB-111 Copper Tube
NFN-1	Low Carbon Nickel
NFN-2	Ni
NFN-3	Ni Cu Alloy
NFN-4	Annealed Ni Cr Fe

NFN-5	Ni Mo Alloy B
NFN-6	Ni Mo Cr Fe
NFN-7	Ni Mo Cr Fe Cu
NFN-8	Ni Fe Cr Alloy 800
NFN-9	Ni Fe Cr Alloy 800H
NFN-10	Ni Moly Chrome Alloy N10276
NFN-11	Ni Cr Fe Mo Cu Alloys G and G-2
NFN-12	Cr Ni Fe Mo Cu Co, SB-462, 463, and so on.
NFN-13	Ni Fe Cr Si Alloy 330
NFN-20	Ni Cr Mo Grade C-4
NFN-15	Ni Mo Alloy X
NFN-16	Ni Mo Alloy B2
NFN-17	Ni Cr Mo Co N06625 (Alloy 625)
NFN-18	Ni Mo Cr Fe Cu (Grade G3)
NFN-19	Ni Mo Cr Fe Cu (Grade G3, >3/4)
NFN-20	Work Hardened Nickel
NFT-1	Unalloyed Titanium, Grade 1
NFT-2	Unalloyed Titanium, Grade 2
NFT-3	Titanium, Grade 1
NFZ-1	Zirconium, Alloy 702
NFZ-2	Zirconium, Alloy 705

Elastic Modulus Reference

The elastic modulus reference number is a value that points to or corresponds to a set of data set forth in ASME Section II Part D, tables TM-1, 2 and so on. Unfortunately, many materials have a composition or UNS number that does not match the criteria of what is supplied in the

ASME Code. In these cases, the reference number will be brought in as zero. If this happens, you will need to enter in an appropriate value.

Reference Number	**Table**	**Description/UNS Number**
1	TM-1	Carbon Steels with C<= 0.3%
2	TM-1	Carbon Steels with C> 0.3%
3	TM-1	Material Group A
4	TM-1	Material Group B
5	TM-1	Material Group C
6	TM-1	Material Group D
7	TM-1	Material Group E
8	TM-1	Material Group F
9	TM-1	Material Group G
10	TM-1	S13800
11	TM-1	S15500
12	TM-1	S45000
13	TM-1	S17400
14	TM-1	S17700
15	TM-1	S66286
16	TM-2	A03560
17	TM-2	A95083
18	TM-2	A95086
19	TM-2	A95456
20	TM-2	A24430
21	TM-2	A91060
22	TM-2	A91100

Reference Number	Table	Description/UNS Number
23	TM-2	A93003
24	TM-2	A93004
25	TM-2	A96061
26	TM-2	A96063
27	TM-2	A92014
28	TM-2	A92024
29	TM-2	A95052
30	TM-2	A95154
31	TM-2	A95254
32	TM-2	A95454
33	TM-2	A95652
34	TM-3	C93700
35	TM-3	C83600
36	TM-3	C92200
37	TM-3	C92200
38	TM-3	C28000
39	TM-3	C28000
40	TM-3	C65500
41	TM-3	C66100
42	TM-3	C95200
43	TM-3	C95400
44	TM-3	C44300
45	TM-3	C44400

Reference Number	Table	Description/UNS Number
46	TM-3	C44500
47	TM-3	C64200
48	TM-3	C68700
49	TM-3	C10200
50	TM-3	C10400
51	TM-3	C10500
52	TM-3	C10700
53	TM-3	C11000
54	TM-3	C12000
55	TM-3	C12200
56	TM-3	C12300
57	TM-3	C12500
58	TM-3	C14200
59	TM-3	C23000
60	TM-3	C61000
61	TM-3	C61400
62	TM-3	C65100
63	TM-3	C70400
64	TM-3	C19400
65	TM-3	C60800
66	TM-3	C63000
67	TM-3	C70600
68	TM-3	C97600

Reference Number	Table	Description/UNS Number
69	TM-3	C71000
70	TM-3	C71500
71	TM-4	N02200
72	TM-4	N02201
73	TM-4	N04400
74	TM-4	N04405
75	TM-4	N06002
76	TM-4	N06007
77	TM-4	N06022
78	TM-4	N06030
79	TM-4	N06045
80	TM-4	N06059
81	TM-4	N06230
82	TM-4	N06455
83	TM-4	N06600
84	TM-4	N06617
85	TM-4	N06625
86	TM-4	N06690
87	TM-4	N07718
88	TM-4	N07750
89	TM-4	N08020
90	TM-4	N08031
91	TM-4	N08330

Reference Number	Table	Description/UNS Number
92	TM-4	N08800
93	TM-4	N08801
94	TM-4	N08810
95	TM-4	N08825
96	TM-4	N10001
97	TM-4	N10003
98	TM-4	N10242
99	TM-4	N10276
100	TM-4	N10629
101	TM-4	N10665
102	TM-4	N10675
103	TM-4	N12160
104	TM-4	R20033
105	TM-5	R50250
106	TM-5	R50400
107	TM-5	R50550
108	TM-5	R52400
109	TM-5	R56320
110	TM-5	R52250
111	TM-5	R53400
112	TM-5	R52402
113	TM-5	R52252
114	TM-5	R52404

Reference Number	Table	Description/UNS Number
115	TM-5	R52254
116	TM-5	R60702
117	TM-5	R60705
118	TM-1	12Cr-13Cr Group F
119	TM-1	20+Cr Material Group G
220	TEMA	Ni-Mo Alloy B
221	TEMA	Tantalum
222	TEMA	Tantalum with 2.5% Tungsten
223	TEMA	7 MO (S32900)
224	TEMA	7 MO PLUS (S32950)
225	TEMA	17-19 CR Stn Steel
226	TEMA	AL-6XN Stn Steel (NO8367)
227	TEMA	AL-29-4-2
228	TEMA	SEA-CURE
229	TEMA	2205 (S31803)
230	TEMA	3RE60 (S31500)

Thermal Expansion Coefficient Reference

Reference Number	Table	Description/UNS Number
1	TE-1	Carbon & Low Alloy Steels, Group 1
2	TE-1	Low Alloy Steels, Group 2
3	TE-1	5Cr-1Mo and 29Cr-7Ni-2Mo-N Steels
4	TE-1	9Cr-1Mo

Reference Number	**Table**	**Description/UNS Number**
5	TE-1	5Ni-¼Mo
6	TE-1	8Ni and 9Ni
7	TE-1	12Cr,13Cr and 13Cr-4Ni Steels
8	TE-1	15Cr and 17Cr Steels
9	TE-1	27Cr Steels
10	TE-1	Austenitic Group 3 Steels
11	TE-1	Austenitic Group 4 Steels
12	TE-1	Ductile Cast Iron
13	TE-1	17Cr-4Ni-4Cu, Condition 1075
14	TE-1	17Cr-4Ni-4Cu, Condition 1150
15	TE-2	Aluminum Alloys
16	TE-3	Copper Alloys C1XXXX Series
17	TE-3	Bronze Alloys
18	TE-3	Brass Alloys
19	TE-3	70Cu-30Ni
20	TE-3	90Cu-10Ni
21	TE-4	N02200 and N02201
22	TE-4	N04400 and N04405
23	TE-4	N06002
24	TE-4	N06007
25	TE-4	N06022
26	TE-4	N06030
27	TE-4	N06045

Reference Number	Table	Description/UNS Number
28	TE-4	N06059
29	TE-4	N06230
30	TE-4	N06455
31	TE-4	N06600
32	TE-4	N06625
33	TE-4	N06690
34	TE-4	N07718
35	TE-4	N07750
36	TE-4	N08031
37	TE-4	N08330
38	TE-4	N08800,N08801,N08810,N08811
39	TE-4	N08825
40	TE-4	N10001
41	TE-4	N10003
42	TE-4	N10242
43	TE-4	N10276
44	TE-4	N10629
45	TE-4	N10665
46	TE-4	N10675
47	TE-4	N12160
48	TE-4	R20033
49	TE-5	Titanium Gr 1,2,3,7,11,12,16 and 17
50	TE-5	Titanium Grade 9

Reference Number	Table	Description/UNS Number
51	TEMA	5Cr-1/2Mo
52	TEMA	7Cr-1/2Mo & 9Cr-1Mo
53	TEMA	Ni-Mo (Alloy B)
54	TEMA	Nickel (Alloy 200)
55	TEMA	Copper-Silicon
56	TEMA	Admiralty
57	TEMA	Zirconium
58	TEMA	Cr-Ni-Fe-Mo-Cu-Cb (Alloy 20Cb)
59	TEMA	Tantalum
60	TEMA	Tantalum with 2.5% Tungsten
61	TEMA	17-19 CR (TP 439)
62	TEMA	AL-6XN
63	TEMA	2205 (S31803)
64	TEMA	3RE60 (S31500)
65	TEMA	7 MO (S32900)
66	TEMA	7 MO PLUS (S32950)
67	TEMA	AL 29-4-2
68	TEMA	SEA-CURE
69	TEMA	80-20 Cu-Ni (C71000)

Minimum Thickness (in.) - Type the minimum allowable thickness for the material. If the material has no minimum thickness, type **-1**.

Maximum Thickness (in.) - Type the maximum allowable thickness for the material. If the material has no maximum thickness, type **-1**.

Creep Temperature (F) - Type the temperature at which the material is governed by time dependent properties.

MDMT Exemption Temperature (F) - When the material uses an impact tested product specification, type the impact temperature. Otherwise, type **1**.

Product Form

Type an integer that designates the product form of the material.

Form Value	Product Form
1	Plate
2	Forgings
3	Seamless pipe
4	Welded pipe
5	Welded tube
6	Seamless tube
7	Bolting
8	Castings
9	Fittings
10	Seamless/welded pipe
11	Seamless/welded tube
12	reserved
13	Seamless pipe and tube
14	Pipe
15	Bar
16	Sheet
17	Tube
18	Forged pipe
19	Seamless/welded fitting
20	Drawn seamless tube

21	Condenser & heat exchanger tubes
22	Seamless extruded tube
23	Rod
24	Seamless and welded fittings
25	Welded fittings
26	Seamless fittings
27	Finned tube
28	Seamless U-bend tube
29	Welded condenser tube

Impact Reduction Temperature (F) - When the material is eligible for a -5ºF temperature reduction according to UCS-66(g), type **-5**. Otherwise, type **0.**

Material Band

The material band is used to determine the modulus of elasticity and coefficient of thermal expansion for that type of material.

Material Band	**Basic Material Type/composition**
M0	Carbon steel
M1	Carbon manganese steel
M2	Carbon molybdenum steel
M4	Low alloy MG Cr Mo V steel
M5	3.5Ni
M6	9Ni
M7	1-1.5Cr .5Mo
M8	.5Cr .5Mo .25V
M9	2.25Cr 1Mo

M10	5Cr .5Mo
M11	9Cr1Mo
M12	12Cr1Mo1V

Calculator

Tools tab: **Calculator**

Opens the Windows-supplied calculator utility. Use **Ctrl-C** and **Ctrl-V** to copy and paste values between the calculator and PV Elite.

Enter U-1 Form Information

Tools tab: **Enter U-1 Form Information**

Opens the **Additional Vessel Information** dialog box into which you enter additional information to produce an ASME U-1 form for the vessel. After analysis, an intermediate results file (.pvu) is created. This file is read by a Microsoft Excel macro when **Create ASME Form** is clicked in **Review Reports**, and the worksheet fields are populated with the calculated results. For more information, see *Review Reports* (on page 196).

Compute Ligament Efficiencies

Tools tab: **Compute Ligament Efficiencies**

Opens the **ASME VIII-1 UG-53 Ligament Efficiency Scratch Pad** dialog box which is used to calculate ASME VIII-1 UG-53.x or 4.10.x ligament efficiencies for tube spacing.

Ligament Efficiency Calculation per Figure - Select one of the following:

- UG-53.1 or 4.10.1
- UG-53.2 or 4.10.2
- UG-53.3 or 4.10.3
- UG-53.4 or 4.10.4

and Figure - When **UG-53.4 or 4.10.4** is selected for **Ligament Efficiency Calculation per Figure**, select one of the following as the secondary figure:

- UG-53.5 or 4.10.5
- UG-53.6 or 4.10.6

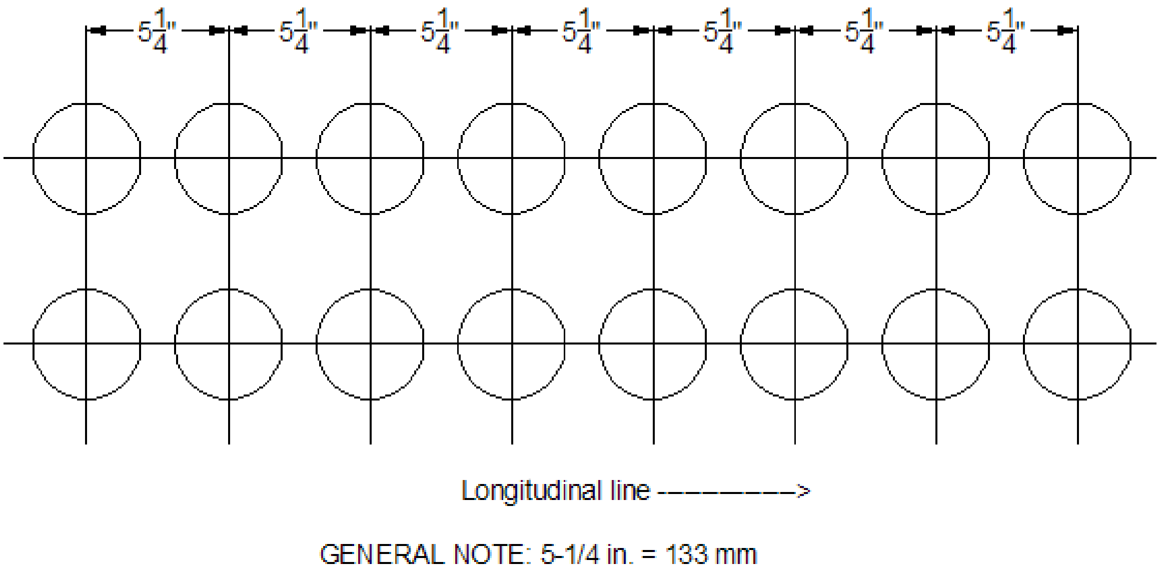
Diameter of Tube Holes (d), **Longitudinal Pitch of Tube Holes (p)**, **Unit Length of Ligament (p1)**, **Diagonal Pitch of tube Holes (p' or p*)**, **Number of Tube Holes in Length p1 (n)**, and **Angle of Diagonal with Longitudinal line (theta)** - Based on the selected ligament efficiency figure, enter values for each dimension.

After all values are entered, the efficiencies are calculated and displayed as a percentage at the bottom of the dialog box. If there is an error, it is also displayed.

All figures are shown below.

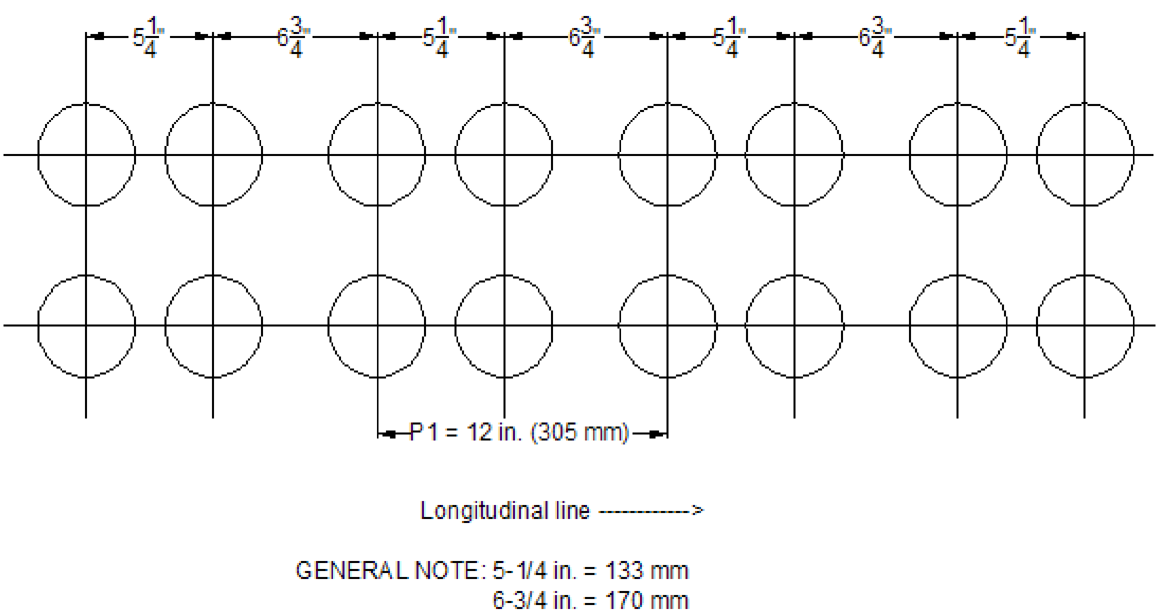
UG-53.1 or 4.10.1

FIG-53.1 Example of Tube Spacing with Pitch of Holes Equal in Every Row



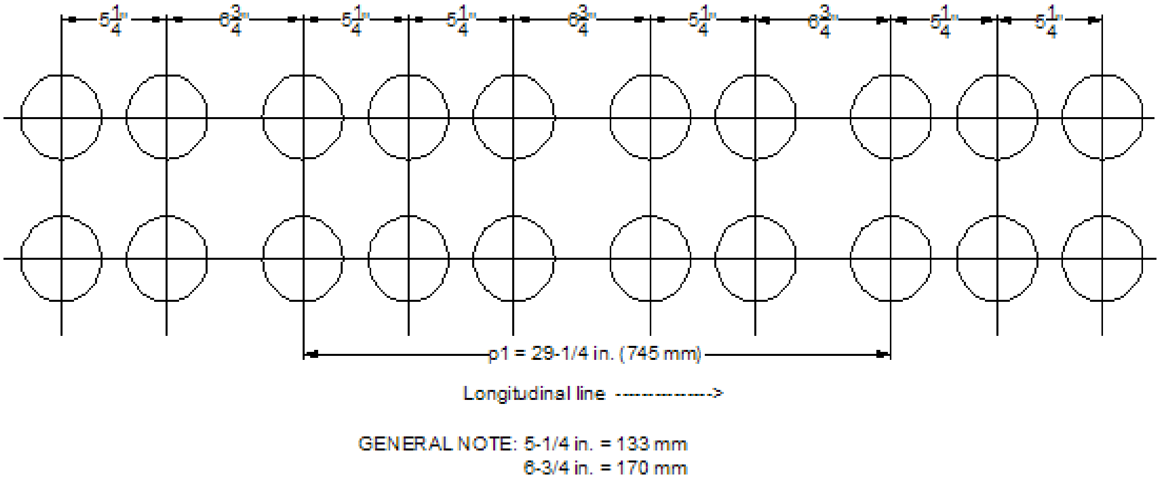
UG-53.2 or 4.10.2

FIG-53.2 Example of Tube Spacing with Pitch of Holes Unequal in Every Second Row



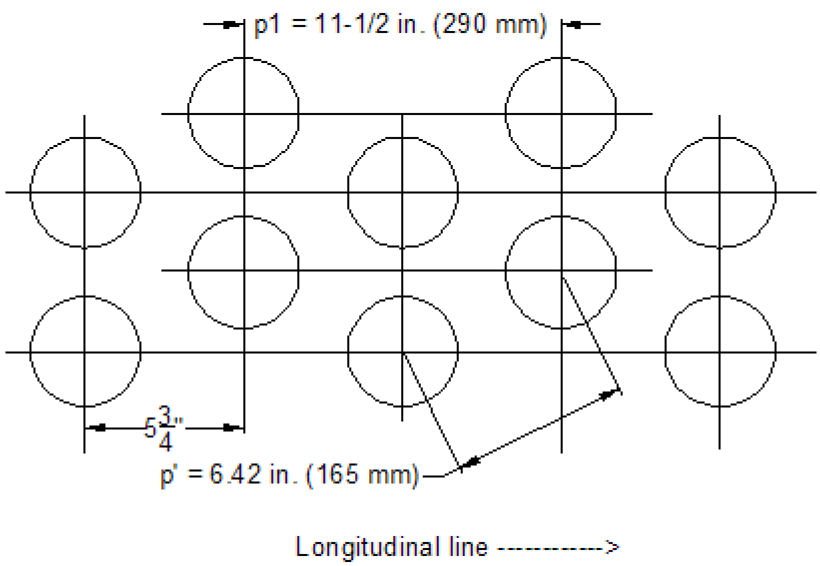
UG-53.3 or 4.10.3

FIG-53.3 Example of Tube Spacing with Pitch of Holes Varying in Every Second and Third Row



UG-53.4 or 4.10.4

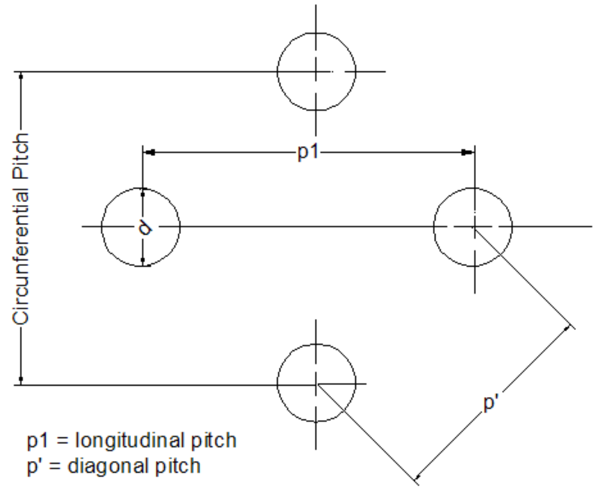
FIG-53.4 Example of Tube Spacing with Pitch with Tube Holes on Diagonal Lines



$$\frac{p1 - nd}{p1} = \text{efficiency of ligament}$$

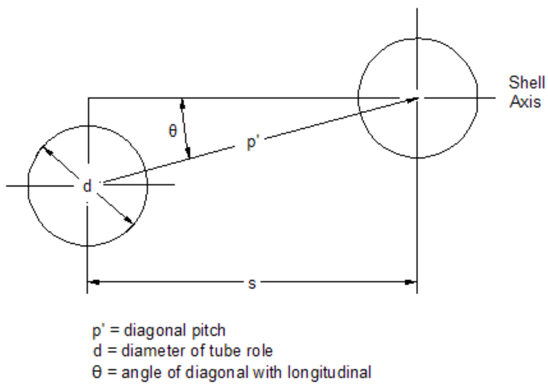
UG-53.5

FIG-53.5 Diagram for Determining the Efficiency of Longitudinal and Diagonal Ligaments Between Openings in Cylindrical Shells



UG-53.6

FIG-53.6 Diagram for Determining Equivalent Longitudinal Efficiency of Diagonal Ligaments Between Openings in Cylindrical Shells

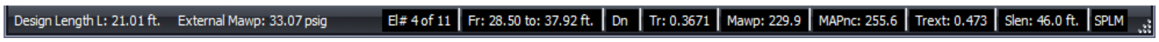


SECTION 6

View Tab

The following commands are available on the **View** tab.

Toggle Status Bar - Click to display or hide the status bar at the bottom of the software window. The status bar displays information about the selected vessel element.



Rigging Results - Displays the calculated rigging results.

Reset Pane Layout - Restores the software views to the default settings.

SECTION 7

3D Tab

The 3D tab contains commands that control the display of elements in the 3D View. The 3D View shows the actual vessel geometry in three dimensions. In addition to showing the outer surfaces, the model can also be viewed in wire frame and hidden line mode. Different shading modes such as flat shaded, Gourard and Phong are supported. Other operations, such as panning, zooming and model rotation are also supported.

Right-click anywhere on the 3D view to display the context menu. These same commands are also on the **3D** tab. The toolbar for performing some basic operations appears on the right side view. For more information, see *3D Graphics Toolbar* (on page 234).

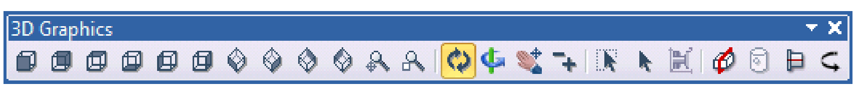
TIP When in 3D viewing mode, the dialog box for a detail can be activated by double-clicking a detail. It might be necessary to first select **Select by Single Click**.

	Visibility - Select and clear the different options to display and hide that element in the 3D graphics view. For example, you might want to hide the skirt to get a better view of a nozzle at the bottom of a vessel. The elements are not deleted from the model, just hidden.
	Wireframe - Shows lines and curves to show the object's borders and all its edges.
	Flat Shaded - Shows the objects with shaded surfaces and smooth edges.
	Gouraud Shaded - Shows the objects with shaded surfaces and smooth edges.
	Hidden Line - Shows the objects using lines and curves to show the object's borders.
	View Orthographic - Displays all objects the same relative size regardless of the distance from the observer.
	View Perspective - Specifies that the view should display the vessel using perspective rendering. Perspective rendering is characterized by: Objects being drawn smaller as their distance from the observer increases. The size of object dimensions along the line of sight are smaller relative to the dimensions across the line of sight.
	Delete Cutting Plane Grid - Removes a cutting plane from the view that you placed using the **Insert Cutting Plane** command on the 3D Graphics Toolbar.

	Fonts and Colors - Activates the Plot Properties dialog with which you can customize the colors of elements in the display.
	Fixed View - Turn on the fix the view.
	Shadow - Turns the vessel shadow on and off.
	Show Materials - Assigns a unique color for each material used in the vessel, and displays those colors in the view and legend grid.
	Show Wall Thickness - Assigns a unique color for each wall thickness value defined in the vessel, and displays those colors in the view and legend grid.
	Show Temperature - Assigns a unique color for each temperature value defined in the vessel, and displays those colors in the view and legend grid.
	Show Pressure - Assigns a unique color for each pressure value defined in the vessel, and displays those colors in the view and legend grid.

3D Graphics Toolbar

The 3D graphics toolbar controls how your model displays in the 3D View. By default, this toolbar displays vertically on the right side of the graphics window. You can toggle the 3D graphics toolbar off and on using the quick access toolbar customization command (black drop-arrow) in the top left-corner of the main window.



	Pre-defined Views - Changes the current view to front, back, top, bottom, left, right view or a standard isometric view.
	Zoom Extents - Resizes the model so that it fits in the current window.
	Zoom Window - Use the mouse to draw a window around the portion of the model that you want to zoom in on. This is a rubber band zoom. Alternately, spin the mouse wheel to zoom in and out.
	Orbit - Rotates the model in any direction using the mouse. Click the right mouse button and move the mouse to rotate the model.
	Turntable Orbit - Rotates the model about the Y-axis.

	Pan - Translates the model in the direction the mouse is dragged. Pressing the mouse wheel and holding it down while moving the mouse will also pan the model.
	Zoom Camera - Zooms in or out. Click this button, then press the left mouse button and move the mouse diagonally across the screen to zoom in or out. Alternately, spin the mouse wheel to zoom in and out.
	Select by Window - Selects details that are inside a fence that you define.
	Select By Click - Allows the selection of a detail for further manipulation.
	Translate Detail - Translates the selected detail in the view.
	Insert Cutting Plane - Inserts a cutting plane when you click this button and then click anywhere in the window. You can then rotate the cutting plane after it has been initiated. The rotating plane exposes the various layers of the vessel. The visibility of the cutting plane can then be turned off after the view is set. To restore the model, right- click in the 3D window and choose **Delete Cutting Plane**.
	Transparency - The main exterior shells of the model are transparent.
	Show Nozzle List - Displays list of nozzles in a list box. The list allows a nozzle to be located in the model for editing.
	Options - Element and detail colors are supported using the **Options** selection. After being set, the software recalls them in subsequent sessions. This option is also available when you right-click on the model window and select **Properties**. The **Options** dialog box displays as shown below. If any of the colors are changed, click **Apply** to update the new color selections.

SECTION 8

Diagnostics Tab

The following commands are available on the **Diagnostics** tab.

Crc Check - Performs a cyclic redundancy check on each of the delivered software DLLs and checks that executable files are correctly copied to the hard drive of your computer. Use this command if your software is behaving erratically.

Program Scanner - Checks the software build version of each executable file. See the *Intergraph web site* www.coade.com/pvelite.htm for the latest build information.

SECTION 9

Esl Tab

The following commands are available on the **Esl (External Software Lock)** tab.

Show Data - Displays the encrypted data on your external software lock (ESL) key that allows you to check the status of the device. The data can also be saved to a log file. This information is useful for updating the software and for remaining current with your Intergraph license.

Access Codes -Creates access codes needed to update the ESL when a new version of the software has been released.

Authorization Codes (on page 240) - Allows ESL update codes to be entered.

Check HASP - See Admin Control Center.

Install HASP - See Admin Control Center.

Admin Control Center - Displays all information related to the HASP Driver. The **Sentinel Keys** option shows all available keys, whether local or on the network. The **Access Log** option displays all instances of a license being used on the network key on the host computer.

Phone Update

In general, each time a new version of PV Elite is released, data on the ESL must be updated to allow the new release to run properly. If the ESL is not updated, an error displays and software commands may not be active.

After installing the new version, use this option to generate four access codes. Phone, email, or fax the codes to Intergraph support. Updated codes are then provided that you enter using **Authorization Codes (on page 240)** command.

NOTES

- The access codes constantly change.
- The time and date on the computer must be correct.

Authorization Codes

Enter one or more sets of update codes, usually to allow access to the latest software version. These codes can also change the client name, or change the last usage date.

Each set of codes received should be entered on a single line. Click **OK** to activate the new codes.

SECTION 10

Help Tab

The following commands are available on the **Help** tab.

Tip of the Day - Displays a tip about the software.

Help Topics - Opens the PV Elite help.

View Documentation - Opens the printable *PV Elite User's Guide, QA Manual,* and *QA certificate.*

Quick Start - Opens the printable *PV Elite Quick Start.*

Foundation 3D Help - Opens the help menu for all Foundation 3D related help topics.

Check for Updates - Checks your version of the software against the most current version. You must be connected to the Internet for this option.

E-mail Support - Creates an email with your system and software information. You can type your support question and send to Intergraph Support *ppmcrm@intergraph.com*. However, eCustomer is the best method to reach Intergraph support. eCustomer provides a comprehensive knowledge base of information and allows Intergraph to track all customer queries, including bug reports, user issues, and user-generated ideas for improvement of the software.

On-Line Registration - Registers this application. You must be connected to the Internet for this option. Intergraph does not give email addresses to third party solicitors. This information is solely used to inform customers regarding updates and release of the software.

On-line Help - Starts an interactive help session with Intergraph Support. You must be connected to the Internet for this option.

What's New - Displays new features for this version of the software.

NOTE For information about your software version, see the **Help** option on the **File** tab.

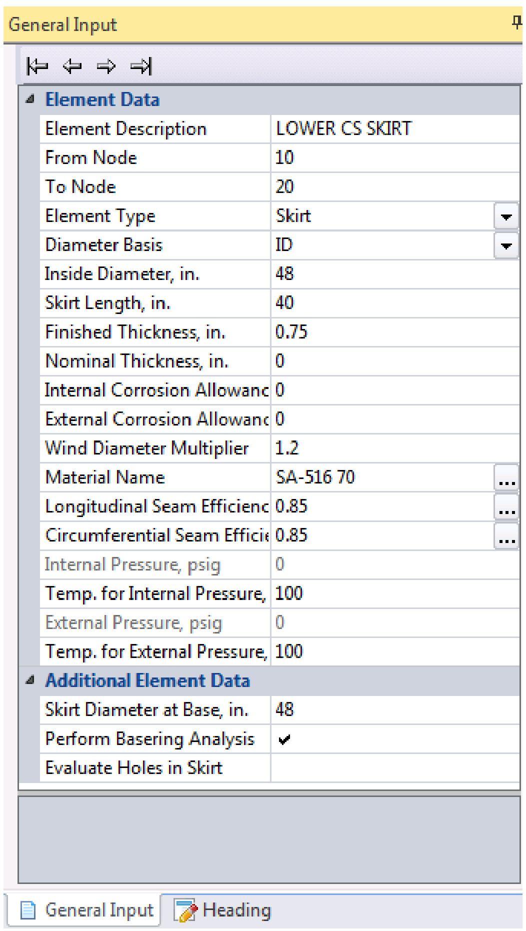
SECTION 11

General Input Tab

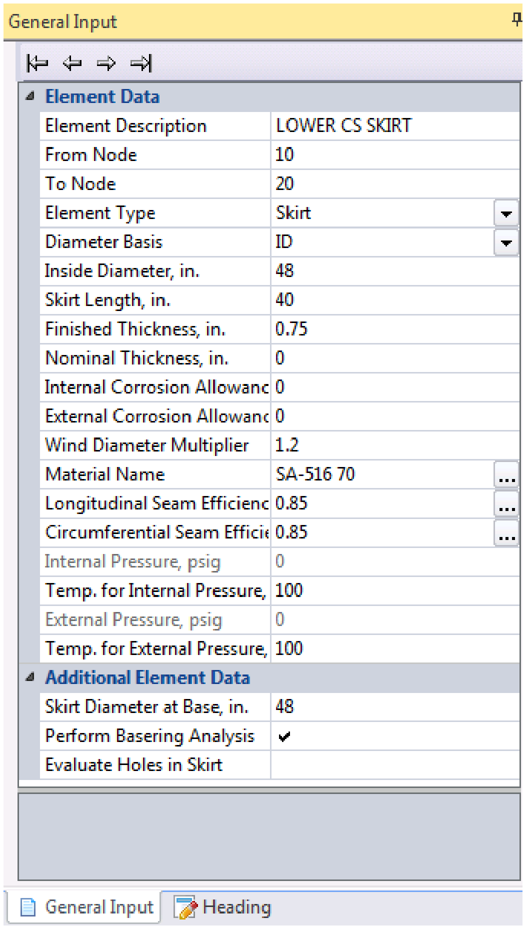
Select a vessel element in the graphics view to make it current, and then edit data for that element on the **General Input** tab.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > General Input** .

   *The **General Input** tab appears at the bottom-left of the PV Elite window.*



3. Use the element navigation arrows to navigate quickly between elements in the vessel.



In This Section

Element Data (General Input Tab)

All elements share a common set of parameters described below. A few of the parameters might be disabled depending on the active element.

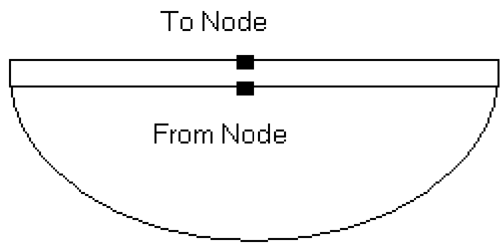
Element Description - Enter a description for the element. The description can be up to 48 characters in length and can consist of both letters and numbers. The description is used in output reports. This entry is optional.

When descriptions are available, the software prints descriptions instead of node numbers. Descriptions are also seen in 3D graphics when you click **Flip to 3D View** on the **Auxiliary** toolbar.

NOTE This value is not a function of the selected vessel code (such as, PD:5500, EN or ASME).

From Node - Displays the software-generated node number describing the starting location of the element. The **From Node** value for this element is also used to define starting locations for details such as nozzles, insulation, and packing that are associated with this element.

The software defines a vertical vessel from bottom to top. If the vertical vessel is on skirt, the first element is the skirt. If it is on legs or lugs, the first element is a head and the legs or lugs are defined as details on the appropriate head or shell elements.



The software defines a horizontal vessel from left to right. The first element in a horizontal vessel is usually a head, and the support saddles are defined as details on the appropriate shell elements.

NOTE This value is not a function of the selected vessel code (such as, PD:5500, EN or ASME).

To Node - Displays the software-generated node number that describes the ending location of the element. The **To Node** value is incremented by 10 above the **From Node** value. **To Node** is **From Node** of the following element. For example, if the value of **To Node** for a head is 50, then the value of **From Node** for the shell is 50.

NOTE This value is not a function of the selected vessel code (such as, PD:5500, EN or ASME).

Element Type - Displays the type of the element. You can change the type by selecting one of the following from the list.

- **Cylinder** - A cylindrical shell
- **Elliptical** - An elliptical head
- **Torispherical** - A Torispherical (F&D) head
- **Spherical** - A spherical head
- **Conical** - A conical head or shell segment
- **Welded Flat** - CA welded flat head
- **Body Flange** - A body flange or blind flange
- **Skirt** - A skirt with an optional basering

CAUTION If the element type is changed, all detail data, such as nozzles, are lost.

NOTE This value is not a function of the selected vessel code (such as, PD:5500, EN or ASME).

Diameter Basis - Select the type of diameter to use for the element. Select **ID** for the inside diameter. Select **OD** for the outside diameter. **ID** and **OD** are available for ASME Division 1. Only **OD** is available for ASME VIII-1.

NOTES

- The ASME code provides different equations for required thickness based on whether the geometry is specified on **ID** or **OD**. By using the **ID** basis, the software computes a thinner required thickness, Tr, for the nozzle, such as in high-pressure, thick-wall geometries.
- If you are modeling a cylinder with welded flat heads on either end, and the welded flat heads sit just inside the cylinder shell, set **Diameter Basis** to **ID** and specify the **Inside Diameter** value on the welded flat heads to be the same size as the **Inside Diameter** of the cylinder. Once you make these changes, if the flat head element still displays as sitting on the cylinder shell (instead of inside of the shell), select **Flip Orientation** twice. The software refreshes the model display to show the welded flat head inside the cylinder shell.

Inside Diameter - Enter a value for the inside diameter of the element, when **ID** is selected for **Diameter Basis:**

- Cylinders - Enter the diameter of the cylinder.
- Elliptical, torispherical and spherical heads - Enter the diameter of the straight flange.
- Skirts - Enter the diameter at the top of the skirt.
- Welded flat heads - Enter the large diameter of the flat head. The value is not a function of the selected code (such as, PD:5500, EN or ASME).

NOTE This option is not available when either **Body Flange** or **Conical is** selected for **Element Type**.

Outside Diameter - Enter a value for the outside diameter of the element, when **OD** is selected for **Diameter Basis:**

- Cylinders - Enter the diameter of the cylinder.
- Elliptical, torispherical and spherical heads - Enter the diameter of the straight flange.
- Skirts - Enter the diameter at the top of the skirt.
- Welded flat heads - Enter the large diameter of the flat head. The value is not a function of the selected code (such as, PD:5500, EN or ASME).

NOTE This option is not available when **Body Flange** or **Conical** are selected for **Element Type**.

"From" End Diameter - Enter a value for the inside or outside diameter at the **From Node** end of the cone, as needed by the selection for **Diameter Basis.** This option is only available when **Conical** is selected for **Element Type**.

Flange ID - Enter a value for the inside of a flange element. This option is only available when **Body Flange** is selected for **Element Type**

- Body flanges - Enter the inside diameter of the body flange.
- Blind flanges - Enter the inside diameter of the flange to which it is bolted. The inside diameter of a blind flange is zero, but the software uses this value to sketch the graphics.

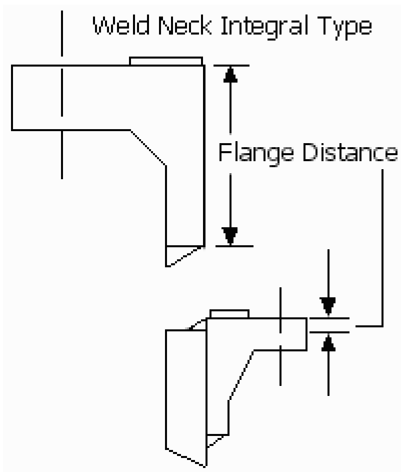
Cylinder Length - Enter the distance between **From Node** and **To Node**. For a cylindrical shell, enter the length of the shell from seam to seam. This option is only available when **Cylinder** is selected for **Element Type**.

Straight Flange Length - Enter the distance between **From Node** and **To Node**. For an elliptical, torispherical, or spherical head, enter the length of the straight flange. The software automatically includes the volume of the head and the depth of the head in calculations. This option is only available when **Elliptical**, **Torispherical**, or **Spherical** are selected for **Element Type**.

Overall Flange Length - Enter the distance between **From Node** and **To Node.**

For an integral weld neck type body flange, enter the through thickness of the flange including the weld neck, if any. If the flange is a slip-on, lap joint or similar, the distance is the length of the flange that protrudes past the shell to which it is attached. This value cannot be zero.

For a bolted blind flange, enter the thickness of the head/flange.



This option is only available when **Body Flange** is selected for **Element Type**.

Flat Head Thickness - Enter the distance between **From Node** and **To Node**. For a welded flat head, enter the thickness of the head/flange.

This option is only available when **Welded Flat** is selected for **Element Type**.

Cone Length - Enter the distance between **From Node** and **To Node**.

For a conical head or shell segment, enter the length of the cone (including toriconical sections, if any) from seam to seam.

For a conical bottom head that is skirt-supported, enter the distance from the top of the skirt to the top of the conical head. You must also enter **Cone Length** in the **Additional Element Data** section. That is the length used in volume calculations and external pressure calculations.

This option is only available when **Conical** is selected for **Element Type**.

Skirt Length - Enter the distance between **From Node** and **To Node**. For a skirt support, enter the distance from the bottom of the base ring to the head tangent line. Because the software does not add the basering thickness to the element length, including the basering thickness in the skirt length provides the correct element elevations. This option is only available when **Skirt** is selected for **Element Type**.

Finished Thickness - Enter the finished thickness of the element. This is **Nominal Thickness** minus mill tolerance and thinning due to forming as described below. Finished thickness is a required input for each vessel element but you can allow PV Elite to increase the element thickness so that each element passes the requirements for internal pressure, external pressure, the combined loads of pressure, dead and live loads. Remember that the status bar lists internal pressure information about the current element including the required thickness.

For elliptical, torispherical and spherical heads, you may have to reduce the nominal thickness of the plate to account for thinning of the head due to forming.

For cylindrical shells made from pipe, you must subtract the maximum possible mill undertolerance from nominal pipe wall thickness. Use the pipe selection button to select standard pipe and insert the thickness into this field.

For welded flat heads this is simply the thickness of the plate from which the head is made.

For a skirt, this is typically the nominal thickness minus any mill undertolerance, and any thinning. For cylindrical skirts made from pipe, you will have to subtract the maximum possible mill undertolerance from the nominal pipe wall thickness.

For a body flange, this is the thickness of the flange. The ASME, EN and PD:5500 codes depict the thickness at the edge of the flange.

NOTE Do not include corrosion allowance. It is automatically subtracted from this thickness by the software when values are included for **Internal Corrosion Allowance** and **External Corrosion Allowance**.

Nominal Thickness - Enter the nominal (design) thickness of the element. For most calculations, the software uses **Finished Thickness** to determine MAWP and other results. However, when calculating element weight, it is theoretically more accurate to use the nominal thickness before the element it is formed. This entry is optional. If left blank, the software uses **Finished Thickness** to determine weight.

Normally, this value would only be applicable to formed heads, but the software allows this entry on all elements for greatest flexibility. When using **Nominal Thickness** to compute a result, the software always takes the greatest of the finished and nominal thicknesses. If the software designs the thickness, you are responsible for adjusting the value of nominal thickness before final calculations are made.

NOTES

- In May 2008, a change was made regarding the calculation of the inside diameter of a cylinder whose **Diameter Basis** is **OD**. In this case, the cylinder ID (usually pipe) is calculated as the nominal OD - 2(Nominal wall thickness). This change affects only nozzle reinforcement calculations. If the diameter basis for the shell is specified as **ID**, there is no change.
- Use of **Nominal Thickness** applies to the selected vessel code (such as, PD:5500, EN or ASME).

Internal Corrosion Allowance - Enter the internal corrosion allowance for this element. Each dimension of the element (diameters and thicknesses) is modified by the corrosion allowance. Some elements in jacketed vessels may have both an internal and **External Corrosion Allowance**.

NOTE Use of **Internal Corrosion Allowance** applies to the selected vessel code (such as, PD:5500, EN-13445 or ASME).

External Corrosion Allowance - Enter the external corrosion allowance for this element. Most vessels do not have an external corrosion allowance specification, but some, such as jacketed vessels, need the consideration of an external corrosion allowance.

If an external corrosion allowance is specified, the software changes the dimension as needed. For example, the OD of a cylinder would be reduced by two times the external corrosion allowance for the external pressure calculation. For flanges, the large and small end hub dimensions are corroded in addition to the flange thickness. The external corrosion allowance is added to the final required thickness of the element.

NOTE Use of **External Corrosion Allowance** applies to the selected vessel code (such as, PD:5500, EN-13445 or ASME).

Wind Diameter Multiplier - Enter the wind load diameter multiplier. The value is multiplied by the element outside diameter in order to determine the overall diameter to be used in wind load calculations. The element outside diameter includes insulation.

When using a number greater than one, carefully account for the tributary area of external attachments such as nozzles, piping, or ladders. The typical multiplier used to determine wind load diameter is 1.2. Thus if the actual element OD is 50 inches, the overall wind load diameter for this element would be 50 * 1.2 = 60 inches.

The value of the wind load multiplier can be as low as 0. If a value of zero is used, then there will be no wind load on the element. This feature is useful when sections of vessels are not exposed to the wind.

NOTES

- If the element contains a platform, use **Platform Input** . The software automatically accounts for the load on the platform due to the wind pressure at the elevation of the platform. **Wind Diameter Multiplier** does not need to account for the platform.
- Use of **Wind Diameter Multiplier** applies to the selected vessel code (such as, PD:5500, EN-13445 or ASME)

Material Name - Enter the name of the material for this element. The software contains a database with most of materials in ASME Code, Section II, Part D, Table 1A, 1B, 5A and 5B. In addition, material for PD 5500 and En-13445 are included.

NOTES

- Alternatively, you can click **Select Material** on the **Utilities** toolbar to select a material directly from the **Material Database Dialog Box** (on page 485).

To modify the material properties of the selected element, click ... to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Longitudinal Seam Efficiency - Enter the efficiency of the welded joint for a shell section with welded longitudinal seams. This is the efficiency of the longitudinal seam in a cylindrical shell or any seam in a spherical shell. Elliptical and torispherical heads are typically seamless but may require a stress reduction which may be entered as a joint efficiency. Refer to Section VIII, Div. 1, **Table UW-12** (below) for help in determining this value. If you know the paragraph reference from UW-12, click ... to open the **Degree of Radiographic Examination and PWHT** dialog box and select the radiography and heat treatment for the joint. The longitudinal and circumferential joint efficiencies are then automatically selected by the software.

The joint efficiency in this (and all other) ASME code formulas is a measure of the inspection quality on the weld seam. In general, weld seams that receive full radiography have a joint efficiency of 1.0. Weld seams that receive spot radiography have a joint efficiency of 0.85. Weld seams that receive no radiography have a joint efficiency of 0.7. Seamless components usually have a joint efficiency of 1.0.

In addition to the basic rules described previously, the code requires that no two seams in the same vessel differ in joint efficiency by more than one category of radiography. For example, if circumferential seams receive no radiography (E = 0.7), then longitudinal seams have a maximum E of 0.85, even if they receive full radiography. In practice, circumferential seams, which are usually less highly stressed, may be spot radiographed (E = 0.85), while longitudinal seams are fully radiographed. This results in the same metal thickness at some savings in inspection costs.

For PD5500 and Section VIII Div. II (pre-2007) this value is not used by the software except for skirt-to-shell welds. If Division II, 2007 edition or later is used, the joint efficiency will be 1 or 0.85 depending on the joint category, type of weld, and the extent of NDE. See section 4.3 and table 7.2 in Division 2 for more information.

Circumferential Seam Efficiency - Enter the efficiency of the welded joint for a shell section with welded circumferential seams. This is the efficiency of the circumferential seam in a cylindrical shell or any seam in a spherical shell. Elliptical and torispherical heads are typically seamless but may require a stress reduction which may be entered as a joint efficiency. Refer to Section VIII, Div. 1, **Table UW-12** (below) for help in determining this value. If you know the paragraph reference from UW-12, click  to open the **Degree of Radiographic Examination and PWHT** dialog box and select the radiography and heat treatment for the joint. The longitudinal and circumferential joint efficiencies are then automatically selected by the software.

The joint efficiency in this (and all other) ASME code formulas is a measure of the inspection quality on the weld seam. In general, weld seams that receive full radiography have a joint efficiency of 1.0. Weld seams that receive spot radiography have a joint efficiency of 0.85. Weld seams that receive no radiography have a joint efficiency of 0.7. Seamless components usually have a joint efficiency of 1.0.

In addition to the basic rules described above, the code requires that no two seams in the same vessel differ in joint efficiency by more than one category of radiography. For example, if circumferential seams receive no radiography (E = 0.7), then longitudinal seams have a maximum E of 0.85, even if they receive full radiography. In practice, circumferential seams, which are usually less highly stressed, may be spot radiographed (E = 0.85), while longitudinal seams are fully radiographed. This results in the same metal thickness at some savings in inspection costs.

For PD5500 and Section VIII Div. II (pre-2007) this value is not used by the software except for skirt-to-shell welds. If Division II, 2007 edition or later is used, the joint efficiency will be 1 or 0.85 depending on the joint category, type of weld, and the extent of NDE. See section 4.3 and table 7.2 in Division 2 for more information.

Table I: MAXIMUM ALLOWABLE JOINT EFFICIENCIES FOR ARC AND GAS WELDED JOINTS from Table UW-12

			Degree of Radiographic Examination			
Type No.	**Joint Description**	**Limitations**	**Weld Joint Category**	**(a) Full[2]**	**(a) Spot[3]**	**(c) None**
(1)	Butt joints as attained by double welding or by other means which will obtain the same quality of deposited weld metal on the inside and outside weld surfaces to agree with the requirements of UW-35. Welds using metal backing strips which remain in place are excluded.	None	A, B, C, D	1.00	0.85	0.70
(2)	Single welded butt joint with backing strip other than those included under (1)	(a) None except in (b) below (b) Circumferential butt joints with one plate offset; see UW-13(b)(4) and Fig. UW-13.1, sketch (k)	A, B, C, D A, B, C	0.90 0.90	0.80 0.80	0.65 0.65
(3)	Single welded butt joint without use of backing strip.	Circumferential butt joints only, not over 5/16" in. (16 mm) thick and not over 24 in. (610 mm) outside diameter	A, B, C	NA	NA	0.60
(4)	Double full fillet lap joint	(a) Longitudinal joints not over 3/8 in. (10 mm) thick (b) Circumferential joints not over 5/8 in. (16 mm) thick	A B & C[6]	NA	NA	0.55

			Degree of Radiographic Examination			
Type No.	**Joint Description**	**Limitations**	**Weld Joint Category**	**(a) Full[2]**	**(a) Spot[3]**	**(c) None**
(5)	Single full fillet lap joints with plug welds conforming to UW-17	(a) Circumferential joints[4] for attachment of heads not over 24 in. (610 mm) outside diameter to shells not over 1.2 in. (13 mm) thick (b) Circumferential joints for the attachment to shells of jackets not over 5/8 in. (16 mm) in nominal thickness where the distance from the center of the plug weld to the edge of the plate is not less than 1 1/2 times the diameter of the hole for the plug	B	NA	NA	0.50
(6)	Single full fillet lap joints without plug welds	(a) For the attachment of heads convex to pressure to shells not over 5/8 in. (16 mm) required thickness, only with use of fillet weld in inside of the shell; or (b) for attachment of heads having pressure on either side, to shells not over 24 in. (610 mm) inside diameter and not over 1/4 in. (6 mm) required thickness with fillet weld on outside of head flange only	A, B A, B	NA	NA	0.45

			Degree of Radiographic Examination			
Type No.	**Joint Description**	**Limitations**	**Weld Joint Category**	**(a) Full[2]**	**(a) Spot[3]**	**(c) None**
(7)	Corner joints, full penetration, partial penetration, and/or fillet welded	As limited by Fig. UW-13.2 and Fig UW-16.1	C[7], D[7]	NA	NA	NA
(8)	Angle joints	Design per U-2(g) for Category B and C joints	B, C, D	NA	NA	NA

Table Notes:
1 - The single factor shown for each combination of joint category and degree of radiographic examination replaces both the stress reduction factor and the joint efficiency factor considerations previously used in this Division
2 - See UW-12(a) and UW-51
3 - See UW-12(b) and UW-52
4 - Joints attaching hemispherical heads to shells are excluded
5 - E = 1.0 for butt joints in compression
6 - For Type No. 4 Category C joint, limitations not applicable for bolted flange connections.
7 - There is no joint efficiency E in the design formulas of this Division for Category C and D joints. When needed, a value of E not greater than 1.00 may be used.

Element is Post Weld Heat Treated - Select on the **Degree of Radiographic Examination and PWHT** dialog box if the element receives post-weld heat treatment. The code makes post weld heat treatment mandatory under certain conditions, such as lethal contents [UW-2] and thick materials [UCS-56]. In the case of carbon and low alloy steels (P1 to P10 materials) operating at low temperatures, the code gives credit if post-weld heat treatment is performed where it is *not* mandatory. Clause UCS-68 of the code discusses this in detail.

If the thickness ratio in Figure UCS-66.1 does not exceed 0.35, the code gives a temperature reduction credit of 30º F (16.66º C). The software determines the MDMT as follows:

1. Determine the minimum design metal temperature (MDMT) from figure UCS-66 using, the appropriate curve for the chosen material.
2. Determine the further reduction in temperature from figure UCS-66.1 where the calculated thickness is less than the actual part thickness.
3. If the ratio as determined by figure UCS-66.1 is less than 0.35, then the MDMT may then be reduced by a further 30º F (16.66º C).

For more information, see UW-2, UW-40. UCS-56, UCS-66 and UCS-68 of the code.

NOTE **Impact Testing (UG-84 and UCS-66)** - Under certain circumstances, the code makes impact testing of materials and weld metal mandatory. This is the case when the design temperature is lower than that determined using the code rules. The code provides rules in UCS-66 for determining the minimum design metal temperature (MDMT) of any part of the vessel. Where vessels are required to operate at a lower temperature than that calculated using the UCS-66 rules, then the materials must be impact tested according to UG-84. If the required energy levels are achieved for the impact test, then the vessel may be operated at the lower temperature.

To include impact testing in your design, go to the **Materials Name** dialog box and check the box under "**Has this material been impact tested?**"

Internal Pressure - Enter the design internal pressure for the element. This pressure need not include any pressure due to liquid head, which is calculated automatically by the software when liquid is defined on this element.

Temperature for Internal Pressure - Enter the design temperature for internal pressure. This value is used as the metal design temperature for the element, especially in determining allowable stress values.

When you edit material properties by selecting a new material name, the program uses this temperature to determine the operating allowable stress value for the material. An exception is stiffening rings. The allowable stress for stiffeners uses **Temperature for External Pressure**.

NOTE For PD:5500 and EN-13445, this value must *never* be less than **Temperature for External Pressure**. The internal design temperature is the value that the software uses to determine the allowable stress for the element.

External Pressure - Enter the design pressure for external pressure analysis. This must be a positive value, such as 15 psig (0.103 MPa). If you enter a zero, the software determines the External MAWP, not the required thickness due to external pressure. For a skirt, zero is the only valid value, because there cannot be external pressure on a skirt.

External pressure definitions are the same for PD:5500, ASME and EN-13445. For example:

- 0 - No External Pressure
- 15 psig (0.1034 MPa) - Full Vacuum
- 0.3 psig - Partial Vacuum

NOTE If the element is an internal head, the external pressure equals the internal pressure on the previous element. If a different value is entered, the software will run, but prompts you with a warning message.

Temperature for External Pressure - Enter the design temperature for external pressure. This value is used as the metal design temperature for external pressure calculations. The software uses the external design temperature and **External Pressure Chart Name** on the **Material Properties Dialog Box** (on page 534) to determine the allowable external pressure from the material tables.

When the software uses the materials tables to determine the allowable axial compression for a vessel, it uses the maximum of **Temperature for Internal Pressure** and **Temperature for External Pressure**. Axial compression may act in combination with either internal or external pressure unless **Vary Compressive Allowable for Internal/External Cases** is selected on the **Load Cases** tab.

NOTES

- For Divisions 1 and 2, this value must not exceed the maximum allowed by the external pressure charts in ASME Section II Part D.
- For PD 5500 and EN-13445, this value is not used to look up values for the allowable stress of the element. The internal temperature is used. If the vessel is a core or vacuum vessel and there is no internal pressure, **Temperature for Internal Pressure** should be set to the same value as **Temperature for External Pressure**.

Additional Element Data (General Input Tab)

Some element types require additional parameters.

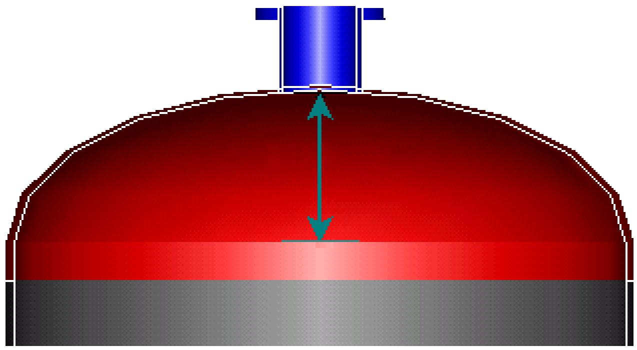
Topics

Elliptical (Additional Element Data)

Defines additional data for elliptical head elements.

Head Factor - Enter the aspect ratio for the elliptical head. The aspect ratio is the ratio of the major axis to the minor axis for the ellipse. For a standard 2:1 elliptical head the aspect ratio is 2.0.

Inside Head Depth - Enter the inside depth of the elliptical head. This value is for a new head and does not include corrosion allowance. The software calculates the outer depth, *h*, and uses this term to calculate the required thickness of the ellipse. This option is only available for the PD:5500 code and you must select **Elliptical** for the **Element Type**.



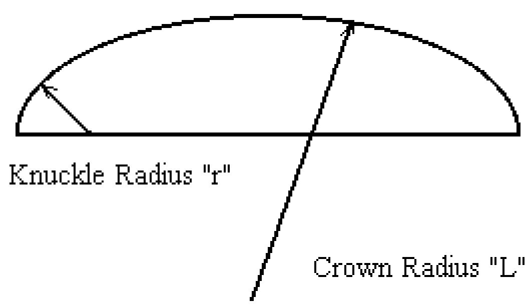
Sump Head - Select if the head is attached to the end of a nozzle. You must also select a **Parent Nozzle**. All of the nozzles on the vessel must be defined before the sump head. The best strategy is to completely define all other elements and details and then create the sump head element last. This option is only available when **Elliptical**, **Torispherical**, or **Spherical** are selected for **Element Type**.

Parent Nozzle - Select the type of nozzle for the sump head. Select **Inlet**, **Outlet**, **Shell Inlet**, or **Shell Outlet**. This option is only available when **Sump Head** is selected.

Torispherical (Additional Element Data)

Defines additional data for torispherical head elements.

Inside Crown Radius - Enter the crown radius *L* of the torispherical head. For a standard ASME flanged and dished head, this is equal to the outside diameter of the shell. See the ASME Code, Section VIII, Division 1, Appendix 1-4, figure 1-4(b). For PD:5500, this value is equal to the outside diameter measured to the tangent between crown and knuckle, as shown in Figure 3.5.2.1.



Inside Knuckle Radius - Enter the knuckle radius *r* for the toroidal portion of the torispherical head. For a standard ASME flanged and dished head, this is equal to six percent of the crown radius. Allowable values range from six percent of the crown radius to 100 percent of the crown radius (hemispherical head). See the ASME Code, Section VIII, Division 1, Appendix 1-4, figure 1-4(b).

Sump Head - Select if the head is attached to the end of a nozzle. You must also select a **Parent Nozzle**. All of the nozzles on the vessel must be defined before the sump head. The best strategy is to completely define all other elements and details and then create the sump head element last. This option is only available when **Elliptical**, **Torispherical**, or **Spherical** are selected for **Element Type**.

Parent Nozzle - Select the type of nozzle for the sump head. Select **Inlet**, **Outlet**, **Shell Inlet**, or **Shell Outlet**. This option is only available when **Sump Head** is selected.

Spherical (Additional Element Data)

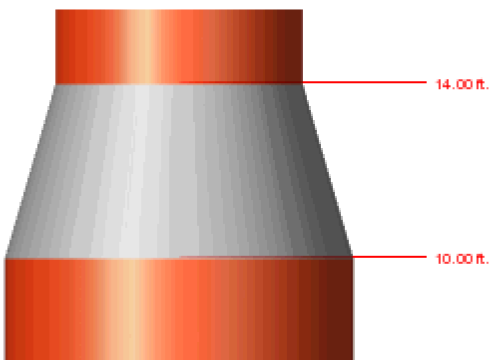
Defines additional data for spherical head elements.

Sump Head - Select if the head is attached to the end of a nozzle. You must also select a **Parent Nozzle**. All of the nozzles on the vessel must be defined before the sump head. The best strategy is to completely define all other elements and details and then create the sump head element last. This option is only available when **Elliptical**, **Torispherical**, or **Spherical** are selected for **Element Type**.

Parent Nozzle - Select the type of nozzle for the sump head. Select **Inlet**, **Outlet**, **Shell Inlet**, or **Shell Outlet**. This option is only available when **Sump Head** is selected.

Conical (Additional Element Data)

Defines additional data for conical head or shell elements.



'To' End Inside Diameter or **'To' End Outside Diameter** - Enter a value for the inside or outside diameter of the cone at the **To Node** end, as needed by the selection for **Diameter Basis.** For a conical head, either the **From Node or the To Node** has a diameter equal to zero or two times the small end knuckle radius. This is not the diameter at the point where a knuckle or flare intersects the conical section, but at the point where the knuckle or flare intersects the cylindrical section.

Cone Length - Enter the seam-to-seam or design length of the cone along the axis of the vessel. The software calculates the effective length of the cone for internal and external pressure calculations.

For cones without a knuckle or flared section, you must enter the design length of the cone. For cases where there is a knuckle or a flare, you must enter the seam-to seam-length. You can optionally enter in the **Half Apex Angle**.

Half Apex Angle - Enter a value if both **Cone Length** (in the **Element Data** section) and **Cone Length** (in the **Additional Element Data** section) are specified and you want to override the angle calculated by the software. If **0** is entered or no value is entered, the software uses the calculated value. This entry is optional.

Refer to the ASME code, Section VIII, Division 1, paragraph UG-33, figure UG-33.1 for a sketch of the half apex angle for some typical geometries. For internal pressure calculations, the half apex angle should not be greater than 30º, though the software will give results for up to 60º. For external pressure calculations, the angle should not be greater than 60º.

For cones without a knuckle, the software calculates discontinuity stresses according to the analysis technique by H. Bednar. For cones whose half apex angles are not within those limits prescribed by the code, this may help you decide if the geometry is acceptable. If the cone and attached cylinders do not have a common centerline, it may be necessary to calculate the greater of the angles and manually enter the value. For cones with a flare or knuckle, PV Elite calculates the half apex angle with the given seam-to-seam cone length. The overall cone length must include the knuckle dimensions.

Line of Support Options - Select a value to determine how the cone is taken as a line of support for external pressure calculations, according to ASME code Section VIII Division 1, Figure UG-28.1. Select one of the following:

- **Not a Line of Support** - The external pressure length for the surrounding sections includes the length of the cone.
- **Both Ends a Line of Support** - The external design length L does not include the cone length.
- **Small End a Line of Support** - The cone length up to small end is included.
- **Large End a Line of Support** - The cone length up to large end is included.

Based on the selection, it may not be necessary to perform moment of inertia calculations according to Appendix 1-8 (End is not a line of support).The code also does not force you to choose one option over the other; in some cases you may find that rings are not required at all. Optimize the design by experimenting with the options.

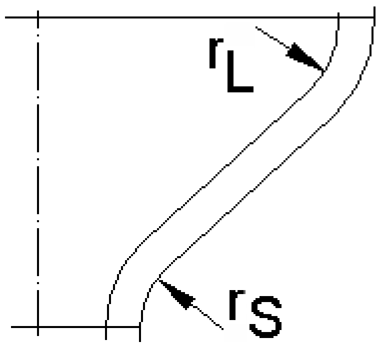
Has Flare or Knuckle? - Select if the cone is toriconical and has a flare at the small end or a knuckle at the large end. The **Toricone Dialog Box** (on page 259) opens. See ASME code, Section VIII, Division 1, Paragraph UG-33, Figure UG-33.1 for an illustration of a toriconical section. This option is only available when **Conical** is selected for **Element Type**.

Is Concentric? - Select if the conical sections are concentric. Clear if the conical sections are eccentric. The software uses concentric sections in vertical geometries and eccentric conical sections in kettle type reboilers.

Shell Section - Select if the cone is a typical transition between two cylinders. Clear if this is a bottom or top conical head. This option applies to skirt-supported geometry.

Toricone Dialog Box

Defines knuckle data on a cone element. This dialog box opens when **Has Flare or Knuckle?** is selected.



Inside Knuckle Radius (rL) - Enter the bend radius of the toroidal knuckle at the large end. ASME Section VIII, Division 1, Paragraph UG-31(h) requires this radius to be no less than six percent of the outside diameter of the head, nor less than three times the knuckle thickness.

For ASME Section VIII Division 2 vessels, also choose the type of curvature of the large end knuckle: **Hemispherical**, **(2:1) Elliptical**, or **Torispherical**.

Knuckle Thickness - Enter the minimum thickness after forming the toroidal knuckle at the large end.

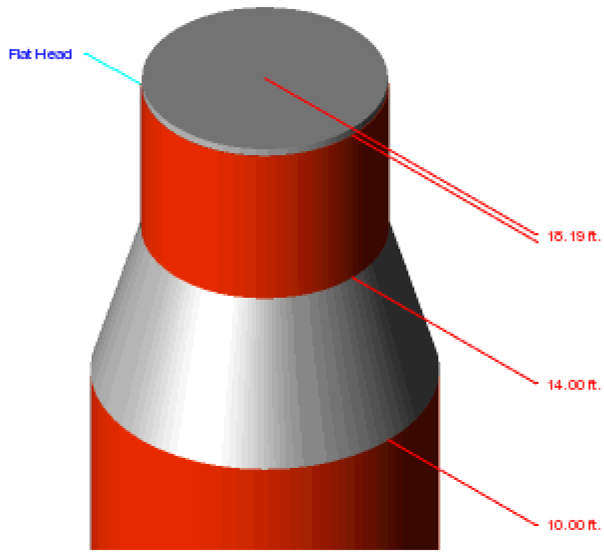
Crown Radius - When **Torispherical** is selected as the large end knuckle curvature, enter the radius of the torisphere crown. For standard geometry, click ... and the software calculates the value. ASME Section VIII, Division 1, Paragraph UG-31(h) requires this radius to be no less than six percent of the outside diameter of the head, nor less than three times the knuckle thickness.

Small end Knuckle Radius (rS) - Enter the bend radius of the toroidal knuckle at the small end. ASME Section VIII, Division 1, Paragraph UG-31(h) requires this radius to be no less than six percent of the outside diameter of the head, nor less than three times the knuckle thickness.

Small End Knuckle Thickness - Enter the minimum thickness after forming the toroidal knuckle at the small end.

Welded Flat (Additional Element Data)

Defines additional data for welded flat head elements.



Welded Flat Head Attachment Sketch - Select the flat head attachment sketch, according to ASME code Section VIII, Division 1, Paragraph UG-34, Figure UG-34. Select from the following:

- **(a)** - Welded cover
- **(b-1)** - Head welded to vessel with generous radius
- **(b-2)** - Head welded to vessel with small radius
- **(c)** - Lap welded or brazed construction
- **(d)** - Integral flat circular heads
- **(e)**, **(f)**, **(g)** - Plate welded inside vessel (check 0.33*m)
- **(h)** - Plate welded to end of shell
- **(i)** - Plate welded to end of shell (check 0.33*m)
- **(m)**, **(n)**, **(o)** - Plate held in place by screwed ring
- **(q)** - Plate screwed into small diameter vessel
- **(r)**, **(s)** - Plate held in place by beveled edge

The sketch is for welded covers, not bolted blind covers.

Attachment Factor - Enter the welded flat head attachment factor for the sketch selected in **Welded Flat Head Attachment Sketch,** according to ASME code Section VIII, Division 1, Paragraph UG-34, Figure UG-34. Some typical attachment factors are below:

- **(b-1)** - 0.17 (Head welded to vessel with generous radius)
- **(b-2)** - 0.20 (Head welded to vessel with small radius)
- **(c)** - 0.20 (Lap welded or brazed construction)
- **(d)** - 0.13 (Integral flat circular heads)

- **(e)**, **(f)**, **(g)** - 0.20 (Plate welded inside vessel (check 0.33*m))
- **(h)** - 0.33 (Plate welded to end of shell)
- **(i)** - 0.20 (Plate welded to end of shell (check 0.33*m))
- **(j)**, **(k)** - 0.30 (Bolted flat heads (include bending moment))
- **(m)**, **(n)**, **(o)** - 0.30 (Plate held in place by screwed ring))
- **(p)** - 0.25 (Bolted flat head with full face gasket)
- **(q)** - 0.75 (Plate screwed into small diameter vessel)

IMPORTANT Consult Paragraph UG-34 before using these values.

NOTES

- For sketches involving *m*, the attachment factor (C) must be calculated and checked per the Code. The software does not calculate *m*.
- For PD 5500, the attachment factor (C) is calculated according to 3.5.5.3 for welded flat ends. The attachment factor for welded flat heads is taken from figure 3.5-36 and is a function of the (cylinder actual thickness)/(cylinder required thickness) ratio and the pressure/allowable stress ratio. Enter the value from the figure 3.5-36 graph.

Non-Circular Small Diameter - Enter a value for the smaller dimension of a non-circular flat head. The large diameter is entered in **Inside Diameter** in the **Element Data** section**.** The software then calculates values such as required thickness using the formulas in the ASME code, Section VIII, Division 1, Paragraph UG-34. Enter a value of **0** if the flat head is circular.

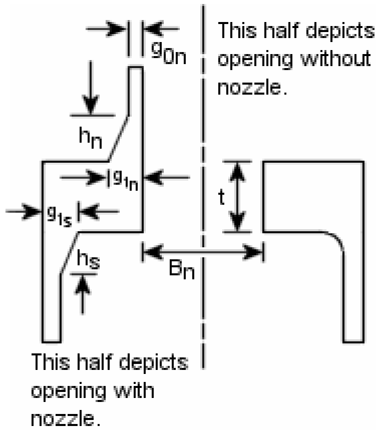
App. 14 Large Central Opening - Select if there is a single, large, centrally-located opening that creates a (hole diameter)/(head diameter) ratio greater than 0.5. This opens the **Integral Flat Head with a Large Centrally Located Opening Dialog Box** (on page 262). This option cannot be used with **Evaluate Uniform Patterned Holes?** and is only available when **Welded Flat** is selected for **Element Type**.

Evaluate Uniform Patterned Holes? - Select if there is a uniform series of holes in the flat head cover. The software then calculates the required thickness due to the ligament created by the hole pattern, according to ASME code Section VIII, Division 1, Paragraph UG-39. When selected, expand **Evaluate Uniform Patterned Holes?** and enter values for **Opening Diameter**, **Ligament Distance to Edge (U3)**, **Opening Diameter (d2)**, **Ligament Distance to Edge (U5)**, and **Pitch Distance (p).**

This option cannot be used when **App. 14 Large Central Opening** is selected. Nozzles are also not compatible and are not analyzed.

Integral Flat Head with a Large Centrally Located Opening Dialog Box

Defines values for calculating stresses and determining the required thickness of a welded flat cover with a large centrally located opening.



Opening Diameter Bn - Enter the diameter of the opening B_n centrally located in the welded flat head, as required by Appendix 14. The diameter should also be greater than 1/2 of the welded flat head outside diameter. If your opening does not meet this criteria, do not use this analysis.

Large End Hub Thk (g1s) - Enter the hub thickness for the large end. The shell side dimensions must be entered in for the calculation to occur. Without proper values of G_{1s}, G_{0s} and h_s, the software does not calculate the required thickness of the head. Like the nozzle, the hub can be straight, and G_{1s} equals G_{0s}. The hub length is entered as stated for nozzles.

Small End Hub Thk (g0s) - Enter the hub thickness for the small end. The shell side dimensions must be entered in for the calculation to occur. Without proper values of G_{1s}, G_{0s} and h_s, the software does not calculate the required thickness of the head. Like the nozzle, the hub can be straight, and G_{1s} equals G_{0s}. The hub length is entered as stated for nozzles.

Hub Length (hs) - Enter the hub length. The shell side dimensions must be entered in for the calculation to occur. Without proper values of G_{1s}, G_{0s} and h_s, the software does not calculate the required thickness of the head. Like the nozzle, the hub can be straight, and G_{1s} equals G_{0s}. The hub length is entered as stated for nozzles.

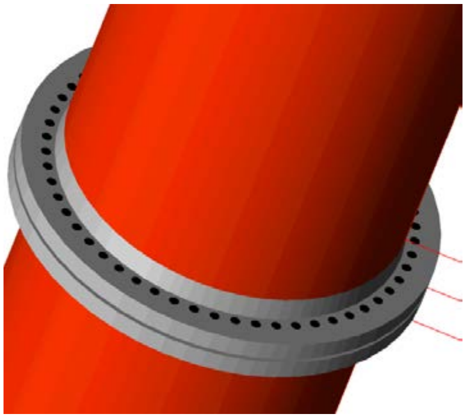
Large End Hub Thk (g1n) - Enter the hub thickness for the large end. The nozzle side may or may not have a nozzle welded to the flat head. If there is no nozzle, the dimensions G_{1n}, G_{0n} and the hub length h_n are zero. If there is a nozzle, you must enter the hub dimensions as shown. For a straight neck, the hub dimensions are the same. If the hub dimensions are the same, the hub length is arbitrary, but cannot be zero. In this case one inch (25mm) is recommended.

Small End Hub Thk (g0n) - Enter the hub thickness for the small end. The nozzle side may or may not have a nozzle welded to the flat head. If there is no nozzle, the dimensions G_{1n}, G_{0n} and the hub length h_n are zero. If there is a nozzle, you must enter the hub dimensions as shown. For a straight neck, the hub dimensions are the same. If the hub dimensions are the same, the hub length is arbitrary, but cannot be zero. In this case one inch (25mm) is recommended.

Hub Length (hn) - Enter the hub length. The nozzle side may or may not have a nozzle welded to the flat head. If there is no nozzle, the dimensions G_{1n}, G_{0n} and the hub length h_n are zero. If there is a nozzle, you must enter the hub dimensions as shown. For a straight neck, the hub dimensions are the same. If the hub dimensions are the same, the hub length is arbitrary, but cannot be zero. In this case one inch (25mm) is recommended.

Body Flange (Additional Element Data)

Defines additional data for flange elements.



Perform Flange Calculation - Select to specify flange parameters and calculate the flange design, such as required thickness, MAWP, and MAPnc, according to the applicable design code. The **Flange Dialog Box** (on page 281) opens. This option is only available when **Body Flange** is selected for **Element Type**.

Flange Weight - Enter the weight of the ANSI/ASME large diameter flange.

This option is only available when **Body Flange** is selected for **Element Type** and **Perform Flange Calculation** is not selected.

Approximate Weight (lbs.) of API 605 Flanges, Class Series B (According to Taylor Forge Cat. #571)

Nominal Size (in.)	**75**	**150**	**300**	**400**	**600**	**900**
26	80	120	400	360	550	1050
28	85	140	450	450	650	1520
30	90	150	550	530	810	1820
32	105	170	685	635	950	2065
34	110	210	750	690	1205	2450
36	145	240	840	855	1340	2520
38	160	290	915	935	1470	3385
40	170	310	990	1090	1630	3620

Nominal Size (in.)	75	150	300	400	600	900
42	185	345	1135	1190	2030	3960
44	230	370	1235	1375	2160	4300
46	245	435	1470	1525	2410	4640
48	270	480	1575	1790	2855	4980
50	290	520	1710	1950	3330	
52	310	550	1840	2125	3560	
54	340	620	1980	2565	3920	
56	400	650	2595	2710	4280	
58	430	780	2770	3230	4640	
60	475	850	2870	3820	5000	

Approximate Weight (lbs.) of MSS SP-44 Flanges, Class Series A (According to Taylor Forge Cat. #571)

Nominal Size (in.)	150	300	400	600	900
26	300	605	650	940	1525
28	345	745	785	1060	1810
30	400	870	905	1210	2120
32	505	1005	1065	1375	2545
34	540	1145	1200	1540	2970
36	640	1275	1340	1705	3395
38	720	695	935	1470	3385
40	775	840	1090	1630	3620
42	890	950	1190	2030	3960
44	990	1055	1375	2160	4300
46	1060	1235	1525	2410	4640
48	1185	1380	1790	2855	4980

Nominal Size (in.)	150	300	400	600	900
50	1270	1530	1950	3330	
52	1410	1660	2125	3560	
54	1585	2050	2565	3920	
56	1760	2155	2710	4280	
58	1915	2270	3230	4640	
60	2045	2470	3820	5000	

ANSI/DIN Class - Select the flange class from the pull down list. The software then finds the allowable pressure on the flange for both the operating and cold conditions of the selected class. This option is only available when **Body Flange** is selected for **Element Type** and **Perform Flange Calculation** is not selected.

ANSI/DIN Grade - Select the nozzle flange material grade. In the ANSI code, grades are assigned into groups.

IMPORTANT There are advisories on the use of certain material grades. Review the cautionary notes in the ANSI B16.5 code.

This option is only available when **Body Flange** is selected for **Element Type** and **Perform Flange Calculation** is not selected.

The following flange grades are available:

ANSI Groups

Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V 3½Ni C-Mn-Si	A 105 A 350 Gr.LF2 A 350 Gr. LF 6 Cl.1 A 350 Gr. LF3	A 216 Gr. WCB A 216 Gr. WCC A 352 Gr. LCC	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si-V 2½Ni 3½Ni C-Si C-Mn-Si	A 350 Gr. LF 6 Cl.2	A 352 Gr. LC2 A 352 Gr. LC3 A 352 Gr. LCB	A 203 Gr. B A 203 Gr. E A 515 Gr. 65 A 516 Gr. 65

Group	Nominal Designation	Forgings	Castings	Plates
1.3	C-½Mo 2½Ni 3½Ni		A 217 Gr. WC1 A 352 Gr. LC1	A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-½Mo ½C-½Mo	A 182 Gr. F1 A 182 Gr. F2		A 204 Gr. A A 204 Gr. B
1.7	Ni-½Cr-½Mo ¾Ni-¾Cr-1Mo		A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1¼Cr-½Mo 1¼Cr-½Mo-Si	A 182 Gr. F22 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	2¼Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.11	Cr-½Mo			A 204 Gr. C
1.13	5Cr-½Mo	A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
1.17	1Cr-½Mo 5Cr-½Mo			
2.1	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304 A 182 Gr. F304H A 182 Gr. F316	A 351 Gr. CF3 A 351 Gr. CF8 A 351 Gr. CF3M	A 240 Gr. 304 A 240 Gr. 304H A 240 Gr. 316
2.2	 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316H A 182 Gr. F317	A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316H A 240 Gr. 317

Group	Nominal Designation	Forgings	Castings	Plates
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti 18Cr-10Ni-Cb	A 182 Gr. F321 A 182 Gr. F321H A 182 Gr. F347		A 240 Gr. 321 A 240 Gr. 321H A 240 Gr. 347
2.5		A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H		A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	23Cr-12Ni			A 240 Gr. 309H
2.7	25Cr-20Ni 20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N	A 182 Gr. F310 A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53	A 351 Gr. CK3MCuN	A 240 Gr. 310H A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750
2.8	24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F55	A 351 Gr. CE8MN A 351 Gr. CD4MCu A 351 Gr. CD3MWCuN	A 240 Gr. S32760
2.9	23Cr-12Ni 25Cr-20Ni			A 240 Gr. 309S A 240 Gr.310S
2.10	25Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	
2.11	18Cr-10Ni-Cb		A 351 Gr. CF8C	
2.12	25Cr-20Ni		A 351 Gr. CK20	
3.1	35Ni-35Fe-20Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200

Group	Nominal Designation	Forgings	Castings	Plates
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe 64Ni-29.5Mo-2Cr-2Fe-Mn-W 54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe	B 462 Gr. N10665 B 462 Gr. N10675 B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001		B 333 Gr. N10665 B 333 Gr. N10675 B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001
3.8	70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu 55Ni-21Cr-13.5Mo 55Ni-23Cr-16Mo01.6Cu	B 573 Gr. N10003 B 574 Gr. 06455 B 564 Gr. N08825 B 462 Gr. N06022 B 462 Gr.N06200		B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. 08825 B 575 Gr. N06022 B 575 Gr. N06200
3.9	47Ni-22Cr-9Mo-I8Fe	B 572 Gr. N06002		B 435 Gr. N06002
3.10	25Ni-46Fe-21Cr-5Mo	B 672 Gr. N08700		B 599 Gr. N08700
3.11	44Fe-25Ni-21Cr-Mo 26Ni-43Fe-22Cr-5Mo	B 649 Gr. N08904 B 621 Gr. N08320		B 625 Gr. N08904 B 620 Gr. N08320

Group	Nominal Designation	Forgings	Castings	Plates
3.12	47Ni-22Cr-20Fe-7Mo 46Fe-24Ni-21Cr-6Mo-Cu-N	B 581 Gr. N06985 B 462 Gr. N08367	A 351 Gr. CN3MN	B 582 Gr. N06985 B 688 Gr. N08367
3.13	47Ni-22Cr-19Fe-6Mo Ni-Fe-Cr-Mo-Cu-Low C	B 581 Gr. N06975 B 462 Gr. N08031		B 582 Gr. N06975 B 625 Gr. N08031
3.14	47Ni-22Cr-19Fe-6Mo 40Ni-29Cr-15Fe-5Mo	B 581 Gr. N06007 B 462 Gr. N06030		B 582 Gr. N06007 B 582 Gr. N06030
3.15	33Ni-42Fe-21Cr	B 564 Gr. N08810		B 409 Gr. N08810
3.16	35Ni-19Cr-1¼Si	B 511 Gr. N08330		B 536 Gr. N08330
3.17	29Ni-20.5Cr-3.5Cu-2.5Mo		A 351 Gr. CN7M	

DIN Grades/Groups - Forgings and Flat Products

Group	Forgings			Flat Products		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
2E0	—	—	—	—	—	—
3E0	—	—	—	P235GH	EN 10028-2	1.0345
3E0	—	—	—	—	—	—
3E0	P245GH	EN 10222-2	1.0352	P265GH	EN 10028-2	1.0425
3E1	P280GH	EN 10222-2	1.0426	P295GH	EN 10028-2	1.0481
4E0	16Mo3	EN 10222-2	1.5415	16Mo3	EN 10028-2	1.5415

Group	Forgings			Flat Products		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
2E0	—	—	—	—	—	—
5E0	13CrMo4-5	EN 10222-2	1.7335	13CrMo4-5	EN 10028-2	1.7335
6E0	11CrMo9-10	EN 10222-2	1.7383	12CrMo9-10	EN 10028-2	1.7375
	-	-	-	10CrMo9-10	EN 10028-2	1.7380
6E1	X16CrMo5-1	EN 10222-2	1.7366	—	—	—
7E0	—	—	—	P275NL1	EN 10028-3	1.0488
	—	—	—	P275NL2	EN 10028-3	1.1104
7E1	—	—	—	P355NL1	EN 10028-3	1.0566
	—	—	—	P355NL2	EN 10028-3	1.1106
7E2	15NiMn6	EN 10222-3	1.6228	15NiMn6	EN 10028-4	1.6228
	—	—	—	11MnNi5-3	EN 10028-4	1.6212
	13MnNi6-3	EN 10222-3	1.6217	13MnNi6-3	EN 10028-4	1.6217
7E3	—	—	—			
	12Ni14	EN 10222-3	1.5637	12Ni14	EN 10028-4	1.5637
	X12Ni5	EN 10222-3	1.5680	X12 Ni 5	EN 10028-4	1.5680
	X8Ni9	EN 10222-3	1.5662	X8Ni9	EN 10028-4	1.5662
8E0	—	—	—	—	—	—
8E2	P285NH	EN 10222-4	1.0477	P275NH	EN 10028-3	1.0487

Group	Forgings			Flat Products		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
2E0	—	—	—	—	—	—
	P285QH	EN 10222-4	1.0478	—	—	—
8E3	P355NH	EN 10222-4	1.0565	P355N	EN 10028-3	1.0562
	P355QH1	EN 10222-4	1.0571	P355NH	EN 10028-3	1.0565
9E0	X20CrMoV11-1	EN 10222-2	1.4922	—	—	1.0565
9E1	X10CrMoVNb9-1	EN 10222-2	1.4903	X10CrMoVNb9-	EN 10028-2	1.4903
10E0	X2CrNi18-9	EN 10222-5	1.4307	X2CrNi18-9	EN 10028-7	1.4307
	—	—	—	X2CrNi19-11	EN 10028-7	1.4306
10E0	—	—	—	X1CrNi25-21	EN 10028-7	1.4335
10E1	X2CrNiN18-10	EN 10222-5	1.4311	X2CrNiN18-10	EN 10028-7	1.4311
11E0	X5CrNi18-10	EN 10222-5	1.4301	X5CrNi18-10	EN 10028-7	1.4301
	X6CrNi18-10	EN 10222-5	1.4948	X6CrNi18-10	EN 10028-7	1.4948
12E0	X6CrNiTi18-10	EN 10222-5	1.4541	X6CrNiTi18-10	EN 10028-7	1.4541
	X6CrNiNb18-10	EN 10222-5	1.4550	X6CrNiNb18-10	EN 10028-7	1.4550
	X6CrNiTiB18-10	EN 10222-5	1.4941	X6CrNiTiB18-10	EN 10028-7	1.4941
13E0	X2CrNiMo17-12-2	EN 10222-5	1.4404	X2CrNiMo17-12-2	EN 10028-7	1.4404

Group	Forgings			Flat Products		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
2E0	—	—	—	—	—	—
	X2CrNiMo17-12-3	EN 10222-5	1.4432	X2CrNiMo17-12-3	EN 10028-7	1.4432
	X2CrNiMo18-14-3	EN 10222-5	1.4435	X2CrNiMo18-14-3	EN 10028-7	1.4435
	X1NiCrMoCu25-20-5	EN 10222-5	Deleted text	X1NiCrMoCu25-20-5	EN 10028-7	1.4539
	—	—	—	X1NiCrMoCu31-27-4	EN 10028-7	1.4563
13E1	X2CrNiMoN17-11-2	EN 10222-5	1.4406	X2CrNiMoN17-11-2	EN 10028-7	1.4406
13E1	X2CrNiMoN17-13-3	EN 10222-5	1.4429	X2CrNiMoN17-13-3	EN 10028-7	1.4429
13E1	—	—	—	X2CrNiMoN17-13-5	EN 10028-7	1.4439
13E1	—	—	—	X1NiCrMoCuN25-20-7	EN 10028-7	1.4529
13E1	—	—	—	X1CrNiMoCuN20-18-7	EN 10028-7	1.4547
14E0	X5CrNiMo17-12-2	EN 10222-5	1.4401	X5CrNiMo17-12-2	EN 10028-7	1.4401
14E0	X3CrNiMo17-13-3	EN 10222-5	1.4436	X3CrNiMo17-13-3	EN 10028-7	1.4436
15E0	X6CrNiMoTi17-12-2	EN 10222-5	1.4571	X6CrNiMoTi17-12-2	EN 10028-7	1.4571
	—	—	—	X6CrNiMoNb17-12-2	EN 10028-7	1.4580
16E0	—	—	—	—	—	—
	—	—	—	X2CrNiN23-4	EN 10028-7	1.4362

Group	Forgings			Flat Products		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
2E0	—	—	—	—	—	—
	X2CrNiMoN22-5-3	EN 10222-5	1.4462	X2CrNiMoN22-5-3	EN 10028-7	1.4462
	X2CrNiMoN25-7-4	EN 10222-5	1.4410	X2CrNiMoN25-7-4	EN 10028-7	1.4410
	—	—	—	—	—	—

DIN Grades/Groups - Castings and Bars

Group	Castings			Bars		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
2E0	GP240GR	EN 10213-2	1.0621	—	—	—
3E0	GP240GH	EN 10213-2	1.0619	P235GH	EN 10273	1.0345
3E0	—	—	—	P250GH	EN 10273	1.0460
3E0	GP280GH	EN 10213-2	1.0625	P265GH	EN 10273	1.0425
3E1	—	—	—	P295GH	EN 10273	1.0481
4E0	G20Mo5	EN 10213-2	1.5419	16Mo3	EN 10273	1.5415
5E0	G17CrMo5-5	EN 10213-2	1.7357	13CrMo4-5	EN 10273	1.7335
6E0	G17CrMo9-10	EN 10213-2	1.7379	11CrMo9-10	EN 10273	1.7383
	—			10CrMo9-10	EN 10273	1.7380
6E1	GX15CrMo5	EN 10213-2	1.7365	—	—	—
7E0	G17Mn5	EN 10213-3	1.1131	—	—	—
	G20Mn5	EN 10213-3	1.6220	—	—	—
7E1	—	—	—	—	—	—

Group	Castings			Bars		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
	—	—	—	—	—	—
7E2	G9Ni10	EN 10213-3	1.5636	—	—	—
	—	—	—	—	—	—
	—	—	—	—	—	—
7E3	—	—	—	—	—	—
	G9Ni14	EN 10213-3	1.5638	—	—	—
	—	—	—	—	—	—
	—	—	—	—	—	—
8E0	—	—	—	—	—	—
8E2	—	—	—	P275NH	EN 10273	1.0487
				—	—	—
8E3	—	—	—	P355NH	EN 10273	1.0565
				P355QH	EN 10273	1.8867
9E0	GX23CrMoV12-1	EN 10213-2	1.4931	—	—	—
9E1	—	—	—	—	—	—
10E0	GX2CrNi19-11	EN 10213-4	1.4309	X2CrNi18-9	EN 10272	1.4307
	—	—	—	X2CrNi19-11	EN 10272	1.4306
10E0	—	—	—	—	—	—
10E1	—	—	—	X2CrNiN18-10	EN 10272	1.4311
11E0	GX5CrNi19-10	EN 10213-4	1.4308	X5CrNi18-10	EN 10272	1.4301
	—	—	—	—	—	—
12E0	—	—	—	X6CrNiTi18-10	EN 10272	1.4541

Group	Castings			Bars		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
	GX5CrNiNb19-11	EN 10213-4	1.4552	X6CrNiNb18-10	EN 10272	1.4550
	—	—	—	—	—	—
13E0	GX2CrNiMo19-11-2	EN 10213-4	1.4409	X2CrNiMo17-12-2	EN 10272	1.4404
	—	—	—	X2CrNiMo17-12-3	EN 10272	1.4432
	—	—	—	X2CrNiMo18-14-3	EN 10272	1.4435
	GX2NiCrMo28-20-2	EN 10213-4	1.4458	X1NiCrMoCu25-20-5	EN 10272	1.4539
	—	—	—	X1NiCrMoCu31-27-4	EN 10272	1.4563
13E1	—	—	—	X2CrNiMoN17-11-2	EN 10028-7	1.4406
13E1	—	—	—	X2CrNiMoN17-13-3	EN 10028-7	1.4429
13E1	—	—	—	X2CrNiMoN17-13-5	EN 10028-7	1.4439
13E1	—	—	—	X1NiCrMoCuN25-20-7	EN 10028-7	1.4529
13E1	—	—	—	X1CrNiMoCuN20-18-7	EN 10272	1.4547
14E0	GX5CrNiMo19-11-2	EN 10213-4	1.4408	X5CrNiMo17-12-2	EN 10272	1.4547
14E0	—	—	—	X3CrNiMo17-13-3	EN 10272	1.4401
15E0	—	—	—	X6CrNiMoTi17-12-2	EN 10272	1.4436
	GX5CrNiMoNb19-11-2	EN 10213-4	1.4581	X6CrNiMoNb17-12-2	EN 10272	1.4571
16E0	GX2CrNiMoCuN25-6-3-3	EN 10213-4	1.4517	—	—	1.4580
	—	—	—	X2CrNiN23-4	EN 10272	—

Group	Castings			Bars		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
	GX2CrNiMoN22-5-3	EN 10213-4	1.4470	X2CrNiMoN22-5-3	EN 10272	1.4362
	—	—	—	X2CrNiMoN25-7-4	EN 10272	1.4462
	GX2CrNiMoN26-7-4	EN 10213-4	1.4469	—	—	1.4410

DIN Grades/Groups - Seamless and Welded Tubes

Group	Seamless Tubes			Welded Tubes		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
2E0	—	—	—	—	—	—
3E0	P195GH	EN 10216-2	1.0348	P195GH	EN 10217-2	1.0348
3E0	P235GH	EN 10216-2	1.0345	P235GH	EN 10217-2	1.0345
3E1	P265GH	EN 10216-2	1.0425	P265GH	EN 10217-2	1.0425
4E1	16Mo3	EN 10216-2	1.5415	16Mo3	EN 10217-2	1.5415
5E0	13CrMo4-5	EN 10216-2	1.7335	—	—	—
6E0	10CrMo9-10	EN 10216-2	1.7380	—	—	—
6E0	11CrMo9-10	EN 10216-2	1.7383	—	—	—
6E1	11CrMo5+NT1	EN 10216-2	1.7362+1	—	—	—
7E0	P275NL1	EN 10216-3	1.0488	P275NL1	EN 10217-3	1.0488
7E0	P275NL2	EN 10216-3	1.1104	P275NL2	EN 10217-3	1.1104
7E1	P355NL1	EN 10216-3	1.0566	P355NL1	EN 10217-3	1.0566
7E1	P355NL2	EN 10216-3	1.1106	P355NL2	EN 10217-3	1.1106
7E2	12Ni14	EN 10216-4	1.5637	—	—	—

Group	Seamless Tubes			Welded Tubes		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
7E2	X10Ni9	EN 10216-4	1.5682	—	—	—
7E3	13MnNi6-3	EN 10216-4	1.6217	—	—	—
8E0	P275NL1	EN 10216-3	1.0488	P275NL1	EN 10217-3	1.0488
8E0	P275NL2	EN 10216-3	1.1104	P275NL2	EN 10217-3	1.1104
8E2	—	—	—	—	—	—
8E3	P355NH	EN 10216-3	1.0565	P355NH	EN 10217-3	1.0565
9E0	X20CrMoV11-1	EN 10216-2	1.4922	—	—	—
9E1	X10CrMoVNb9-1	EN 10216-2	1.4903	—	—	—
10E0	X2CrNi18-9	EN 10216-5	1.4307	X2CrNi18-9	EN 10217-7	1.4307
	X2CrNi19-11	EN 10216-5	1.4306	X2CrNi19-11	EN 10217-7	1.4306
10E0	X1CrNi25-21	EN 10216-5	1.4335	—	—	—
	X2CrNiN18-10	EN 10216-5	1.4311	X2CrNiN18-10	EN 10217-7	1.4311
11E0	X5CrNi18-10	EN 10216-5	1.4301	X5CrNi18-10	EN 10217-7	1.4301
11E0	X6CrNi18-10	EN 10216-5	1.4948	—	—	—
12E0	X6CrNiTi18-10	EN 10216-5	1.4541	X6CrNiTi18-10	EN 10217-7	1.4541
	X6CrNiNb18-10	EN 10216-5	1.4550	X6CrNiNb18-10	EN 10217-7	1.4550
12E0	X7CrNiTi18-10	EN 10216-5	1.4940	—	—	—
12E0	X7CrNiTiB18-10	EN 10216-5	1.4941	—	—	—
12E0	X7CrNiNb18-10	EN 10216-5	1.4912	—	—	—

Group	Seamless Tubes			Welded Tubes		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
	X8CrNiNb16-13	EN 10216-5	1.4961	—	—	—
13E0	X2CrNiMo17-12-2	EN 10216-5	1.4404	X2CrNiMo17-12-2	EN 10217-7	1.4404
	—	—	—	X2CrNiMo17-12-3	EN 10217-7	1.4432
	X2CrNiMo18-14-3	EN 10216-5	1.4435	X2CrNiMo18-14-3	EN 10217-7	1.4435
13E0	X1NiCrMoCu25-20-5	EN 10216-5	1.4539	X1NiCrMoCu25-20-5	EN 10217-7	1.4539
	X1NiCrMoCu31-27-4	EN 10216-5	1.4563	X1NiCrMoCu31-27-4	EN 10217-7	1.4563
	—	—	—	X2CrNiMoN18-15-4	EN 10217-7	1.4438
	X6CrNiMo17-13-2	EN 10216-5	1.4918	—	—	—
13E1	X2CrNiMoN17-13-3	EN 10216-5	1.4429	X2CrNiMoN17-13-3	EN 10217-7	1.4429
	X2CrNiMoN17-13-5	EN 10216-5	1.4439	X2CrNiMoN17-13-5	EN 10217-7	1.4439
	X1CrNiMoN25-22-2	EN 10216-5	1.4466	—	—	—
	X1CrNiMoCuN20-18-7	EN 10216-5	1.4547	X1CrNiMoCuN20-18-7	EN 10217-7	1.4547
	X1NiCrMoCuN25-20-7	EN 10216-5	1.4529	X1NiCrMoCuN25-20-7	EN 10217-7	1.4529
14E0	X5CrNiMo17-12-2	EN 10216-5	1.4401	X5CrNiMo17-12-2	EN 10217-7	1.4401

Group	Seamless Tubes			Welded Tubes		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
	X3CrNiMo17-13-3	EN 10216-5	1.4436	X3CrNiMo17-13-3	EN 10217-7	1.4436
15E0	X6CrNiMoTi17-12-2	EN 10216-5	1.4571	X6CrNiMoTi17-12-2	EN 10217-7	1.4571
	X6CrNiMoNb17-12-2	EN 10216-5	1.4580	—	—	—
16E0	X2CrNiMoS18-5-3	EN 10216-5	1.4424	—	—	—
	X2CrNiMoN22-5-3	EN 10216-5	1.4462	X2CrNiMoN22-5-3	EN 10217-7	1.4462
	X2CrNiN23-4	EN 10216-5	1.4362	X2CrNiN23-4	EN 10217-7	1.4362
	X2CrNiMoN25-7-4	EN 10216-5	1.4410	X2CrNiMoN25-7-4	EN 10217-7	1.4410
	X2CrNiMoCuN25-6-3	EN 10216-5	1.4507	—	—	—
	X2CrNiMoCuWN25-7-4	EN 10216-5	1.4501	X2CrNiMoCuWN25-7-4	EN 10217-7	1.4501
13E1	X2CrNiMoN17-13-3	EN 10216-5	1.4429	X2CrNiMoN17-13-3	EN 10217-7	1.4429
	X2CrNiMoN17-13-5	EN 10216-5	1.4439	X2CrNiMoN17-13-5	EN 10217-7	1.4439
	X1CrNiMoN25-22-2	EN 10216-5	1.4466	—	—	—
	X1CrNiMoCuN20-18-7	EN 10216-5	1.4547	X1CrNiMoCuN20-18-7	EN 10217-7	1.4547
	X1NiCrMoCuN25-20-7	EN 10216-5	1.4529	X1NiCrMoCuN25-20-7	EN 10217-7	1.4529
14E0	X5CrNiMo17-12-2	EN 10216-5	1.4401	X5CrNiMo17-12-2	EN 10217-7	1.4401

Group	Seamless Tubes			Welded Tubes		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
	X3CrNiMo17-13-3	EN 10216-5	1.4436	X3CrNiMo17-13-3	EN 10217-7	1.4436
15E0	X6CrNiMoTi17-12-2	EN 10216-5	1.4571	X6CrNiMoTi17-12-2	EN 10217-7	1.4571
	X6CrNiMoNb17-12-2	EN 10216-5	1.4580	—	—	—
16E0	X2CrNiMoS18-5-3	EN 10216-5	1.4424	—	—	—
	X2CrNiMoN22-5-3	EN 10216-5	1.4462	X2CrNiMoN22-5-3	EN 10217-7	1.4462
	X2CrNiN23-4	EN 10216-5	1.4362	X2CrNiN23-4	EN 10217-7	1.4362
	X2CrNiMoN25-7-4	EN 10216-5	1.4410	X2CrNiMoN25-7-4	EN 10217-7	1.4410
	X2CrNiMoCuN25-6-3	EN 10216-5	1.4507	—	—	—
	X2CrNiMoCuWN25-7-4	EN 10216-5	1.4501	X2CrNiMoCuWN25-7-4	EN 10217-7	1.4501

Flange Type - Select the ANSI flange type. Select **Weld Neck**, **Slip On**, or **Blind**. This option is only available when **Body Flange** is selected for **Element Type**.

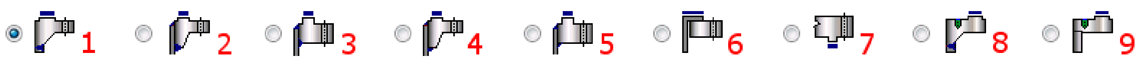
Nominal Size Lookup - Select a nominal flange diameter. The flange diameter, length, and thickness are automatically updated. This value is not saved. This option is only available when **Body Flange** is selected for **Element Type**.

Flange Dialog Box

Defines values for calculating actual and allowable stresses for all types of flanges according to the ASME Code, Section VIII, Division 1, VIII-2; PD 5500; and EN-13445.

Select a Flange Type - Select the type of flange:

- 1 - Integral Weld Neck Flange
- 2 - Integral Slip on Flange
- 3 - Integral Ring Flange
- 4 - Loose Slip on Flange
- 5 - Loose Ring Flange
- 6 - Lap Joint Flange
- 7 - Bolted Blind Flange or TEMA channel Cover
- 8 - Reverse Geometry Weld Neck Flange
- 9 - Loose Reverse Flange

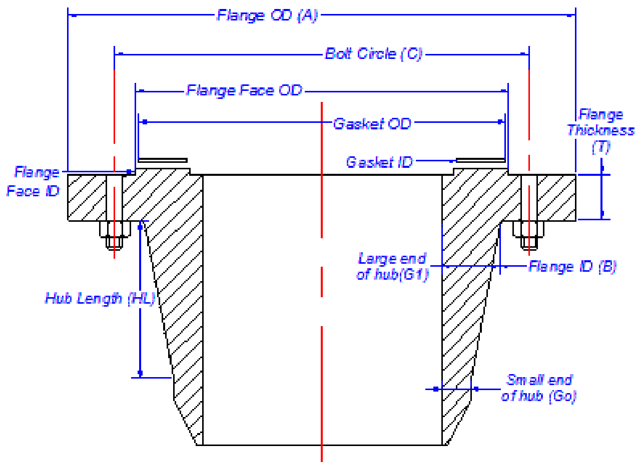


NOTE If the type 7 blind flange is selected, the modulus of elasticity at design temperature is needed. By default the software uses the external pressure charts to retrieve this information. Alternately, you can use the elasticity data supplied in the TEMA standard by typing in the TEMA identifier in **Flange ID**. The TEMA identifier is a number that ranges between 16 and 50, and depends on the composition of the flange material. Listings of the TEMA numbers can be found in *Tubesheet Type and Design Code Tab (Heat Exchanger Tubesheet Input Dialog Box)* (on page 125).

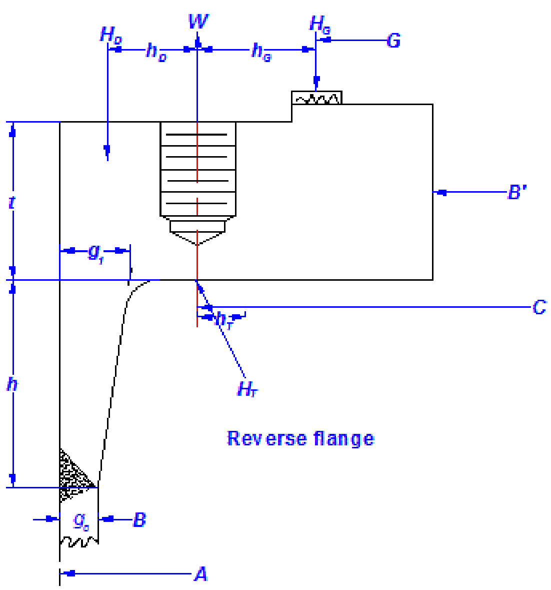
Description - Enter an alphanumeric description for the item.

Flange ID - Enter the inside diameter of the flange. For integral-type flanges, this value is also the inner pipe diameter. This value is referred to as *B* in the ASME code. **Internal Corrosion Allowance** is used to adjust this value (two times the corrosion allowance is added to the uncorroded value of **Flange ID**).

For blind flanges the value is 0.0.



For reverse flanges this is the *B*` dimension as shown in appendix 2 of the ASME Code.



Flange OD - Enter the outer diameter of the flange. This value is referred to as *A* in the ASME code. If the flange is being corroded from the outside, be sure this is a corroded dimension.

Face ID - Enter the inner diameter of the flange face. The software uses the maximum of the **Face ID** and **Gasket ID** to calculate the inner contact point of the gasket.

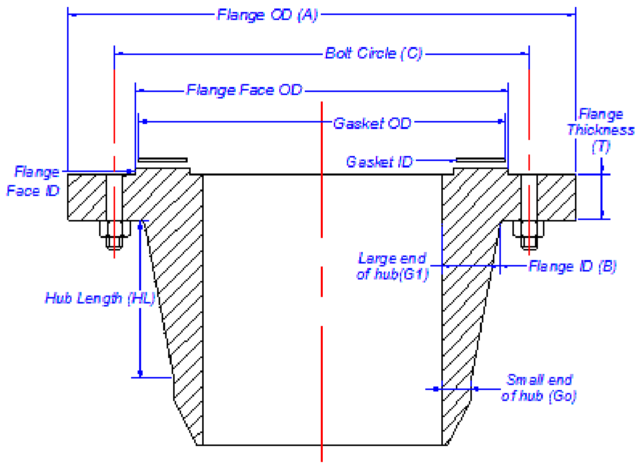
Face OD - Enter the outer diameter of the flange face. The software uses the minimum of **Face OD** and the **Gasket OD** to calculate the outside flange contact point, but uses the maximum in design when selecting the bolt circle. This is done so that the bolts do not interfere with the gasket.

Gasket ID - Enter the inner diameter of the gasket. The software uses the maximum of the **Flange ID** and the **Gasket ID** to calculate the inner contact point of the gasket.

Gasket OD - Enter the outer diameter of the gasket. The software uses the minimum of **Flange OD** and **Gasket OD** to calculate the outside flange contact point, but uses the maximum in design when selecting the bolt circle. This is done so that the bolts do not interfere with the gasket.

Thickness Large - Enter the thickness of the large end of the hub. This value is referred to as *G1* in the ASME code. The corrosion allowance is subtracted from this value. It is permissible for the hub thickness at the large end to equal the hub thickness at the small end.

For flange geometry without hubs, this thickness may be entered as zero.



Thickness Small - Enter the thickness of the small end of the hub. This value is referred to as G_o in the ASME code. The corrosion allowance is subtracted from this value.

For weld neck flange types, this is the thickness of the shell at the end of the flange. For loose slip on flanges without hubs, this is typically the thickness of the attached shell or the hub dimensions can all be zero. For loose ring flanges with a hub, this is the thickness of the hub. For optional loose ring flanges with hubs analyzed as integral, this is the thickness of the attached shell. See ASME general note 2 of Figure 2-4 for more information.

If you are using PD 5500 or EN-13445, please check the code for correct input values.

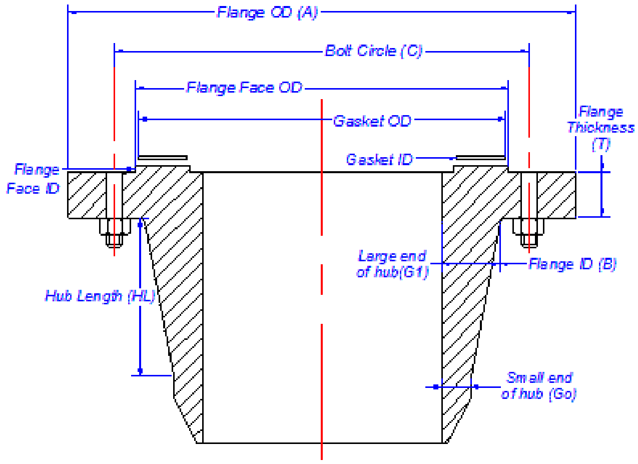
Hub Length - Enter the hub length. This value is referred to as *H* in the ASME code.

For flange geometry without hubs, this length may be entered as zero. When analyzing an optional type flange that is welded at the hub end, the hub length should be the leg of the weld, and the thickness at the large end should include the thickness of the weld.

When you analyze a flange with no hub, such as a ring flange or a lap joint flange, you should enter zero for the hub length, the small end of the hub, and the large end of the hub. However, when you design a loose flange as a ring flange which has a fillet weld at the back, enter the size of a leg of the fillet weld as the large end of the hub. This insures that the software designs the bolt circle far enough away from the back of the flange to get a wrench around the nuts.

Bolt Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Bolt Circle Diameter - Enter the diameter of the bolt circle of the flange.



Thread Series - Select the thread series bolt table. Select **TEMA**, **TEMA Metric**, **UNC**, **BS 3643**, or **SABS 1700 1996.** If you have a bolt that is outside of the bolt table ranges, select **User Root Area** and enter the nominal size in **Nominal Bolt Diameter**.

Nominal Bolt Diameter - Click ... and select the nominal bolt diameter from the bolt table selected for **Type of Threads**. If you have a bolt that is outside of the bolt table ranges, enter the nominal size and select **User Root Area** in **Type of Threads**.

For **TEMA** and **UNC** bolts, the available bolt diameters are:

Bolt Size (inches)	**Bolt Root Area**	
	(cm.^2)	**(in.^2)**
0.500	1.27	0.126
0.625	1.5875	0.202
0.750	1.9050	0.302
0.875	2.22225	0.419
1.000	2.54000	0.551
1.125	2.85750	0.728
1.250	3.17500	0.929
1.375	3.49250	1.155
1.500	3.81000	1.405
1.625	4.12750	1.680

Bolt Size	Bolt Root Area	
1.750	4.44500	1.980
1.875	4.76250	2.304
2.000	5.08000	2.652
2.250	5.71500	3.423
2.500	6.35000	4.292
2.750	6.98500	5.259
3.000	7.62000	6.324
3.250	8.25500	7.487
3.500	8.89000	8.749
3.750	9.52500	10.108
4.000	10.1600	11.566

NOTE This information is adapted from Jawad and Farr, *Structural Analysis and Design of Process Equipment*, pg 425.

Number of Bolts - Enter the number of bolts to be used in the flange analysis.

Root Area - For nonstandard bolts, enter the root cross-sectional area of the bolt.

Gasket Factor m | y - Enter values for the gasket material and contact facing. Enter the factor ratio (**m**) and minimum stress (**y**). Alternatively, click ... to open the **Select a Gasket** dialog box, and select a standard gasket material.

ASME Table 2-5.1 - Gasket List

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
Self-energizing types (O rings, metallic elastomer, and other types considered as self-sealing)	0.00	0.00	II
Elastomers WO/ fabric or high %of asbestos fiber:			
Below 75A Shore Durometer	0.50	0.0	II
75A or higher Shore Durometer	1.00	200.0	II
Asbestos with Suitable Binder			

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
1/8" thick	2.00	1600.0	II
1/16" thick	2.75	3700.0	II
1/32" thick	3.50	6500.0	II
Elastomers W/cotton fabric insert	1.25	400.0	II
Elastomers W/Asbestos fabric insert:			
3 ply	2.25	2200.0	II
2 ply	2.50	2900.0	II
1 ply	2.75	3700.0	II
Vegetable fiber	1.75	1100.0	II
Spiral-wound, asbestos filled:			
Carbon	2.50	10000.0	II
Stainless, Monel, Nickel alloys	3.00	10000.0	II
Corrugated Metal, Asbestos Ins. or corrugated metal, jacketed asbestos:			
soft aluminum	2.50	2900.0	II
soft copper or brass	2.75	3700.0	II
iron or soft steel	3.00	4500.0	II
Monel or 4%-6% chrome	3.25	5500.0	II
SS/nickel base alloys	3.50	6500.0	II
Corrugated Metal:			
soft aluminum	2.75	3700.0	II
soft copper or brass	3.00	4500.0	II
iron or soft steel	3.25	5500.0	II
Monel or 4%-6% chrome	3.50	6500.0	II

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
SS/nickel base alloys	3.75	7600.0	II
Flat metal, jacketed asbestos filled:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel	3.50	8000.0	II
4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	3.75	9000.0	II
Grooved Metal:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel or 4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	4.25	10100.0	II
Solid flat metal:			
soft aluminum	4.00	8800.0	I
soft copper or brass	4.75	13000.0	I
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I
Ring Joint:			
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
SS/nickel base alloys	6.50	26000.0	I

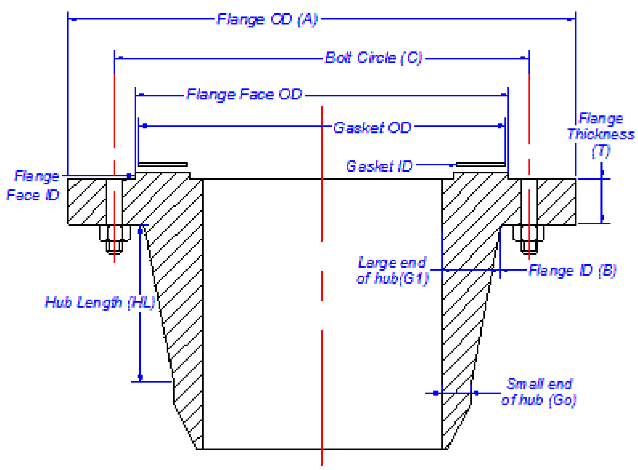
NOTE *SS* refers to stainless steel

Sketch - Select the facing sketch number according to the following and using Table 2-5.2 of the ASME code (see **Gasket Factor m | y**):

Facing Sketch	**Description**
1a	Flat finish faces
1b	serrated finish faces
1c	Raised nubbin-flat finish
1d	Raised nubbin-serrated finish
2	1/64 inch nubbin
3	1/64 inch nubbin both sides
4	Large serration, one side
5	Large serration, both sides
6	Metallic O-ring type gasket

Column - Select the facing column according to **ASME Table 2-5.1 - Gasket List**.

Gasket Thickness - Enter the gasket thickness. This value is only required for facing sketches 1c and 1d (PV Elite equivalents 3 and 4). For more information, see **Sketch**.



Nubbin or RTJ Width - Enter the nubbin width. This value is only required for facing sketches 1c, 1d, 2 and 6 (PV Elite equivalents 3, 4, 5, and 9). For sketch 9, this is not a nubbin width, but the contact width of the metallic ring. For more information, see **Sketch**.

Operating, Wm1 - Enter the bolt load from the mating flange for the operating case.

Seating, Wm2 - Enter the bolt load from the mating flange for the seating condition.

Design, W - Enter the design bolt load for the mating flange.

Axial Force - Enter the magnitude of the external axial force acting on the flange. Because the axial load rotates the flange and causes stress in the hub, this value should always be positive.

Bending Moment - Enter the magnitude of the external bending moment acting on the flange. This is typically the square root of the sum of the squares of moments that contribute to bending (thus. the torsional moment would not be considered). The axial force and resolved moment increase the design pressure on the flange. This is because only pressure and bolt loads cause loads on flange. The ASME code does not consider the effect of bending and external forces on flanges.

Length - Enter the length of the partition gasket. This is the cumulative length of all heat exchanger pass partition gaskets associated with this flange.

NOTE If the flange does have a partition gasket, enter **0** for **Length** and **Width**.

Width - Enter the width of the pass partition gasket. Using **Width**, **Sketch**, **Column, and Gasket Factor m | y,** the software calculates the effective seating width and the gasket loads contributed by the partition gasket.

NOTE If the flange does have a partition gasket, enter **0** for **Length** and **Width**.

Sketch - Select the facing sketch number according to the following and using Table 2-5.2 of the ASME code (see **Gasket Factor m | y**):

Facing Sketch	Description
1a	Flat finish faces
1b	serrated finish faces
1c	Raised nubbin-flat finish
1d	Raised nubbin-serrated finish
2	1/64 inch nubbin
3	1/64 inch nubbin both sides
4	Large serration, one side
5	Large serration, both sides
6	Metallic O-ring type gasket

Column - Select the facing column according the table below:

ASME Table 2-5.1 - Gasket List

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
Self-energizing types (O rings, metallic elastomer, and other types considered as self-sealing)	0.00	0.00	II
Elastomers WO/ fabric or high %of asbestos fiber:			
Below 75A Shore Durometer	0.50	0.0	II
75A or higher Shore Durometer	1.00	200.0	II
Asbestos with Suitable Binder			
1/8" thick	2.00	1600.0	II
1/16" thick	2.75	3700.0	II
1/32" thick	3.50	6500.0	II
Elastomers W/cotton fabric insert	1.25	400.0	II
Elastomers W/Asbestos fabric insert:			
3 ply	2.25	2200.0	II
2 ply	2.50	2900.0	II
1 ply	2.75	3700.0	II
Vegetable fiber	1.75	1100.0	II
Spiral-wound, asbestos filled:			
Carbon	2.50	10000.0	II
Stainless, Monel, Nickel alloys	3.00	10000.0	II
Corrugated Metal, Asbestos Ins. or corrugated metal, jacketed asbestos:			
soft aluminum	2.50	2900.0	II
soft copper or brass	2.75	3700.0	II
iron or soft steel	3.00	4500.0	II
Monel or 4%-6% chrome	3.25	5500.0	II

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
SS/nickel base alloys	3.50	6500.0	II
Corrugated Metal:			
soft aluminum	2.75	3700.0	II
soft copper or brass	3.00	4500.0	II
iron or soft steel	3.25	5500.0	II
Monel or 4%-6% chrome	3.50	6500.0	II
SS/nickel base alloys	3.75	7600.0	II
Flat metal, jacketed asbestos filled:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel	3.50	8000.0	II
4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	3.75	9000.0	II
Grooved Metal:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel or 4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	4.25	10100.0	II
Solid flat metal:			
soft aluminum	4.00	8800.0	I
soft copper or brass	4.75	13000.0	I

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I
Ring Joint:			
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I

NOTE *SS* refers to stainless steel

Gasket Factor m | y - Enter values for the gasket material and contact facing. Enter the factor ratio (**m**) and minimum stress (**y**). Alternatively, click ... to open the **Select a Gasket** dialog box, and select a standard gasket material. See **ASME Table 2-5.1 - Gasket List**, above.

Base Required Thickness on Rigidity (ASME VIII-1)? - Select to calculate the flange rigidity index for both the mating flange **Operating, Wm1 and Seating, Wm2** (as it would normally). If either rigidity index is greater than 1.0, the software iteratively adjusts the flange required thickness until neither index is greater than 1.0. Using this method may prove useful for designing flanges with the new higher allowable stresses (ASME 99 addenda and later), because, as of November 1999, the code has not explicitly addressed the situation of using higher allowables for flange design.

NOTE In some cases, stress considerations govern over rigidity, and the thickness is not influenced by rigidity.

Include Corrosion in Flange Thickness Calculations? - Select to use the corrosion allowance (as is usual) when calculating the final stresses on the flange.

Clear to calculate stresses without subtracting corrosion from the flange thickness, typically producing a thinner flange that is not as highly stressed. The MAWP of the flange is also usually slightly higher. Because this is not directly addressed by any code, agreement is usually between the customer and the manufacturer.

Rigidity Calc. for Seating Case Uses Corroded Thickness? - Select to incorporate the corroded thickness into the rigidity calculation, especially for cases of old pipe already corroded.

Are the Hub and Attached Shell Materials the Same? - Select to indicate that the flange material is same as the attached shell material. The larger of the shell allowable and the flange allowable stress are used to calculate the required small-end hub thickness for the attached integral flange. Clear to consider only the flange allowable.

For some materials with relatively low yield strength (such as stainless steels), the ASME code has established higher stress values. These higher stress values (indicated by the presence of the note G5) can lead to higher deformation. As a result, these material allowables are not used for applications where flange deformation can cause failure or leakage.

When the flange allowables are lower as compared to the attached shell (for the same material), the small-end hub thickness is checked as a cylinder. The allowable stresses of the attached shell should then be used. Otherwise, the required hub thickness may be greater than that of the attached cylinder. If you want the higher shell allowables to be used, select **Are the Hub and Attached Shell Materials the Same?**.

Calculate Cover Deflection (multi-pass unit)? - Select if the flange is a TEMA channel cover (bolted blind flange). A separate thickness, deflection, and MAWP are calculated for channel covers.

Allowable Cover Deflection - For TEMA channel covers, enter the maximum allowed deflection at the center of the cover, as defined in TEMA paragraph RCB-9.21. For TEMA covers with nominal diameters less than or equal to 24 inches, the allowed deflection should be set to **.03** inches. For larger sizes, the deflection should be limited to .125% of the nominal diameter (nominal diameter / 800). If this field is left blank the appropriate allowable deflection will be used. Alternatively, click ... to open the **Compute Allowable Channel Cover Deflection** dialog box, and enter a value for **Nominal Diameter of Attached Shell**. The software then calculates the allowable cover deflection.

Number of Splits in Ring - Enter the number of splits in the ring, if any, for loose-type flanges. This value must be **0**, **1,** or **2**. Typical split flanges are ring-type flanges. A split is used when the flange must be completely removable from the vessel. If the flange is split into two pieces by a single split, the design moment for the flange is multiplied by 2.0. If the flange consists of two separate split rings, each ring is designed as if it were a solid flange (without splits) using 0.75 times the design moment. The pair of rings is assembled so that the splits in one ring are 90º from the splits in the other.

Weld Leg Back of Ring - Enter the length of the weld leg at the back of the ring. This value is added to the inside diameter during the design of ring type flanges to determine the minimum bolt circle.

Attached Shell Thickness - Enter the thickness of the shell attached to the flange.

Lap Joint Contact Point ID - Enter the inner diameter of the flange/joint contact surface.

Lap Joint Contact Point OD - Enter the outer diameter of the flange/joint contact surface.

Diameter (Long Span) - Enter the head characteristic diameter, used to calculate the non-circular flange correction factor *Z* for ASME blind flanges. This factor is defined in paragraph UG-34 of the ASME code. This dimension is perpendicular to the short span dimension.

NOTE For circular blind flanges, the short span and long span dimensions are the same.

Diameter (Short Span) - Enter the head characteristic short diameter, used to calculate the non-circular flange correction factor *Z* for ASME blind flanges. This factor is defined in paragraph UG-34 of the ASME code. This dimension is perpendicular to the long span dimension. It is very important to enter this dimension correctly because it is used in the tangential flange stress computation.

NOTE For circular blind flanges, the short span and long span dimensions are the same.

Perimeter Along Bolt Circle Centerline - Enter the perimeter *L* of the bolted head measured along the centerline of the bolts. This value is needed for both noncircular and circular geometry. For a circular head, enter the value of (π * bolt circle diameter). For non-circular heads this value will have to be manually calculated and entered.

Is this a Standard Flange (No calculation performed)? - Select to define the flange by using ANSI/DIN standards.

Class - Select the ANSI/DIN flange class, which is based on the pressure rating.

Grade - Select the flange material grade (group). Please note that there are certain advisories on the use of certain material grades. Please review those cautionary notes in the ANSI B16.5 code. ASME B16.5-2003 and ASME B16.5-1996 flange grades are available:

Table 1A List of Material Specifications (ASME B16.5-2003)

Material Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V 3½ Ni	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1 A 350 Gr. LF3	A 216 Gr. WCB	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si C-Mn-Si-V 2½Ni 3½Ni	A 350 Gr. LF 6 Cl.2	A 316 Gr. WCC A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	A 203 Gr. B A 203 Gr. E
1.3	C-Si C-Mn-Si 2 ½Ni 3 ½Ni C-½Mo		A 352 Gr. LCB A 217 Gr. WC1 A 352 Gr. LC1	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1		A 204 Gr. A A 204 Gr. B
1.7	½C-½Mo Ni-½Cr-½Mo ¾Ni-¾Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1¼Cr-½Mo 1¼Cr-½Mo-Si	A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	2¼Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.11	Cr-½Mo			A 204 Gr. C
1.13	5Cr-½Mo	A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2

Material Group	Nominal Designation	Forgings	Castings	Plates
1.17	1Cr-½Mo 5Cr-½Mo	A 182 Gr. F12 Cl.2 A 182 Gr. F5		
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H A 182 Gr. F317	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H		A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	23Cr-12Ni			A 240 Gr. 309H
2.7	25Cr-20Ni	A 182 Gr. F310		A 240 Gr. 310H
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
2.9	23Cr-12Ni 25Cr-20Ni			A 240 Gr. 309S A 240 Gr. 310S
2.10	25Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	
2.11	18Cr-10Ni-Cb		A 351 Gr. CF8C	
2.12	25Cr-20Ni		A 351 Gr. CK20	
3.1	35Ni-35Fe-10Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600

Material Group	Nominal Designation	Forgings	Castings	Plates
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe 64Ni-29.5Mo-2Cr-2Fe-Mn-W	B 462 Gr. N10665 B 462 Gr. N10675		B 333 Gr. N10665 B 333 Gr. N10675
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu 55Ni-21Cr-13.5Mo 55Ni-23Cr-16Mo-1.6Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825 B 462 Gr. N06022 B 462 Gr. N06200		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825 B 575 Gr. N06022 B 575 Gr. N06200
3.9	47Ni-22Cr-9Mo-I8Fe	B 572 Gr. N06002		B 435 Gr. N06002
3.10	25Ni-46Fe-21Cr-5Mo	B 672 Gr. N08700		B 599 Gr. N08700
3.11	44Fe-25Ni-21Cr-Mo	B 649 Gr. N08904		B 625 Gr. N08904
3.12	26Ni-43Fe-22Cr-5Mo 47Ni-22Cr-20Fe-7Mo 46Fe-24Ni-21Cr-6Mo-Cu-N	B 621 Gr. N08320 B 581 Gr. N06985 B 462 Gr. N08367	A 351 Gr. CN3MN	B 620 Gr. N08320 B 582 Gr. N06985 B 688 Gr. N08367
3.13	49Ni-25Cr-18Fe-6Mo Ni-Fe-Cr-Mo-Cu-Low C	B 581 Gr. N06975 B 462 Gr. N08031		B 582 Gr. N06975 B 625 Gr. N08031
3.14	47Ni-22Cr-19Fe-6Mo 40Ni-29Cr-15Fe-5Mo	B 581 Gr. N06007 B 462 Gr. N06030		B 582 Gr. N06007 B 582 Gr. N06030
3.15	33Ni-42Fe-21Cr	B 564 Gr. N08810		B 409 Gr. N08810
3.16	35Ni-19Cr-1¼Si	B 511 Gr. N08330		B 536 Gr. N08330
3.17	29Ni-20.5Cr-3.5Cu-2.5Mo		A 351 Gr. CN7M	

Table 1A List of Material Specifications (ASME B16.5-1996)

Material Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1	A 216 Gr. WCB A 216 Gr. WCC	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si C-Mn-Si-V 21/2Ni 31/2Ni	A 350 Gr. LF 6 Cl.2 A 350 Gr. LF3	A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	A 203 Gr. B A 203 Gr. E
1.3	C-Si C-Mn-Si 21/2Ni 31/2Ni		A 352 Gr. LCB	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D

Material Group	Nominal Designation	Forgings	Castings	Plates
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1	A 217 Gr. WC1 A 352 Gr. LC1	A 204 Gr. A A 204 Gr. B A 204 Gr. C
1.7	C-1/2Mo 1/2Cr-1/2Mo Ni-1/2Cr-1/2Mo 3/4Ni-3/4Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1Cr-1/2Mo 11/4Cr-1/2Mo 11/4Cr-1/2Mo-Si	A 182 Gr. F12 Cl.2 A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	21/4Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.13	5Cr-1/2Mo	A 182 Gr. F5 A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H	A 351 Gr. CF8C	A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	25Cr-12Ni 23Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	A 240 Gr. 309S A 240 Gr. 309H
2.7	25Cr-20Ni	A 182 Gr. F310	A 351 Gr. CK20	A 240 Gr. 310S A 240 Gr. 310H

Material Group	Nominal Designation	Forgings	Castings	Plates
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53 A 182 Gr. F55	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
3.1	35Ni-35Fe-20Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe	B 335 Gr. N10665		B 333 Gr. N10665
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825

Nom - Select the nominal value for the flange inside diameter.

ANSI Series - Select the ANSI flange series. Select **ANSI Series A** for general-use flanges. Select **ANSI Series B** for compact flanges.

Get Flange Dimensions Now! - Click to look up flange dimensions from ANSI/DIN tables based on the selections for **Is this a Standard Flange (No calculation performed)?**, **Class**, **Grade**, **Nom**, and **ANSI Series**. The flange thickness table value is shown in **Flange Thickness**.

NOTE Click **Quick Results** to see a report of the flange dimensions.

Flange Thickness - This option is used in the following ways:

- Displays standard flange thickness after **Get Flange Dimensions Now** is clicked.
- Displays designed flange thickness after **Design** is clicked.
- Enter a manual flange thickness.

Use Full Bolt Load in Calc (Sa*Ab)? - Select to allow the full bolt load to be used on just the area of the bolt itself, instead of Am+ Ab, which is the area of the bolt (Ab) plus the required bolt area (Am). You use this option for allowable stress calculations.

Just Like - Select the node of an adjacent element to use the properties of that element. After selecting, click **Copy Now**.

Copy Now - Click to copy the properties of an adjacent element selected in **Just Like**. The **Select Items to Copy** dialog box opens. You then select the properties to copy.

Design - Click to design the flange based on all properties selected in the **Flange** dialog box.

Quick results - Click to see a report of flange results.

Delete - Click to delete all data entered in the **Flange** dialog box, restore default flange values, and close the dialog box.

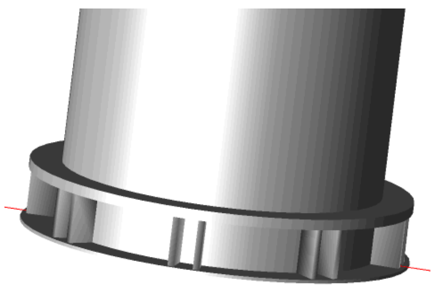
OK - Click to save all data entered in the **Flange** dialog box and close the dialog box.

Cancel - Click to close the **Flange** dialog box without saving the entered data.

Plot - Click to open the **Flange Graphics** dialog box. A cross-section view of the flange design is shown.

Skirt (Additional Element Data)

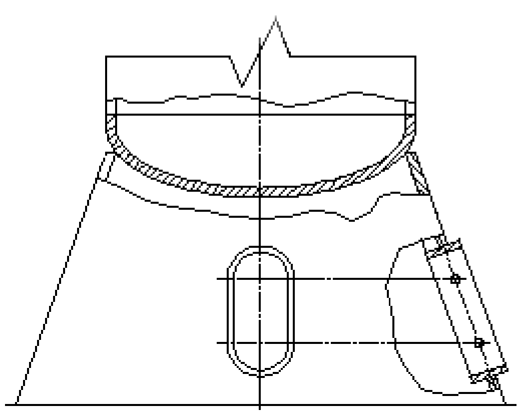
Defines additional data for a skirt support element with basering.



Skirt Diameter at Base - Enter the inside diameter at the bottom of the skirt. This value must be larger than or equal to the inside diameter at the top of the skirt. This option is only available when **Skirt** is selected for **Element Type**.

Perform Basering Analysis - If there is a basering on the skirt, select to specify basering parameters and calculate the basering design. The **Basering Dialog Box** (on page 300) opens. This option is only available when **Skirt** is selected for **Element Type**.

Evaluate Holes in Skirt - Select to specify skirt access opening parameters and analyze the openings. Openings are analyzed when they are of reasonable size, are on tall towers, or are under sizable wind/seismic loads. This option is only available when **Skirt** is selected for **Element Type**.



This option opens the **Skirt Access Openings Dialog Box** (on page 320).

For more information, see, Skirt Opening Analysis Considerations.

Basering Dialog Box

This dialog box defines data for baserings on a skirt element. Basering analysis performs thickness calculations and design for annular plate baserings, top rings, bolting, and gussets found on skirts for vertical vessels. These calculations are performed using industry standard calculation techniques. For more information, see *Basering Analysis Considerations* (on page 300).

The Brownell and Young Method - The Brownell and Young method of design calculates the required thickness of the baseplate, the gussets, and the top plate or top ring. Brownell and Young discuss this method in their book *Process Equipment Design*. Dennis R. Moss also discusses it in the book *Pressure Vessel Design Manual*. This baseplate design method is based on the neutral axis shift method and generally results in a thinner basering design than the method discussed in *Basering Analysis Considerations* (on page 300).

Basering Design Data Tab (Basering Dialog Box) (on page 306)

Tailing Lug Data Tab (Basering Design Data Tab) (on page 313)

Basering Analysis Considerations

Thickness of a Basering under Compression - The equation for the thickness of the basering is the equation for a simple cantilever beam. The beam is assumed to be supported at the skirt, and loaded with a uniform load caused by the compression of the concrete due to the combined weight of the vessel and bending moment on the down-wind / down-earthquake side of the vessel.

Thickness of a cantilever, t:

$$t = l * \sqrt{\frac{3\, f_c}{s_{allow}}}$$

Where

fc = Bearing stress on the concrete

l = Cantilever length of basering

s = Allowable bending stress of basering (typically 1.5 times the code allowable)

There are two commonly accepted methods of determining the stress from the vessel and base-ring acting on the concrete. The simplified method calculates the compressive stress on the concrete assuming that the neutral axis for the vessel is at the centerline.

Stress acting on the concrete, fc:

$$fc = -\left(\frac{W}{A} + \frac{Mc}{I}\right)$$

Where:

W = Weight of the vessel together with the basering

M = Maximum bending moment on vessel

A = Cross-sectional area of basering on foundation

c = Distance from the center of the basering to the outer edge of the basering

I = Moment of inertia of the basering on the foundation

However, when a steel skirt and basering are supported on a concrete foundation, the behavior of the foundation is similar to that of a reinforced concrete beam. If there is a net bending moment on the foundation, then the force upward on the bolts must be balanced by the force downward on the concrete. Because these two materials have different elastic moduli, and because the strain in the concrete cross section must be equal to the strain in the base ring at any specific location, the neutral axis of the combined bolt/concrete cross section will be in the direction of the concrete. Several authors, including Jawad and Farr (*Structural Analysis and Design of Process Equipment,* pg 428 - 433) and Megyesy (*Pressure Vessel Handbook,* pg 70 - 73), have analyzed this phenomenon. The software uses the formulation of Singh and Soler (*Mechanical Design of Heat Exchangers and Pressure Vessel Components*, pg 957 - 959). This formulation seems to be the most readily adaptable to computerization, as there are no tabulated constants. Singh and Soler provide the following description of their method:

In this case, the neutral axis is parallel to the Y-axis. The location of the neutral axis is identified by the angle alpha. The object is to determine the peak concrete pressure (*p*) and the angle alpha.

For narrow base plate rings, an approximate solution may be constructed using numerical iteration. It is assumed that the concrete annulus under the base plate may be treated as a thin ring of mean diameter *c*. Assuming that the foundation is linearly elastic and the base plate is relatively rigid, Brownell and Young have developed an approximate solution which can be cast in a form suitable for numerical solution. Let the total tensile stress area of all foundation bolts be *A*. Within the limits of accuracy sought, it is permissible to replace the bolts with a thin shell of thickness *t* and mean diameter equal to the bolt circle diameter *c*, such that:

Thickness, t:

$$t = \frac{A}{Pl} * c$$

Where:

A = Total cross-sectional area of all foundation bolts

P = Peak concrete pressure

l = Width of basering

c = Thin ring diameter

We assume that the discrete tensile bolt loads, acting around the ring, are replaced by a line load, varying in intensity with the distance from the neutral plane.

Let *n* be the ratio of Young's moduli of the bolt material to that of the concrete; *n* normally varies between 10 and 15. Assuming that the concrete can take only compression (non-adhesive surface) and that the bolts are effective only in tension (untapped holes in the base plate), an analysis, similar to that given above, yields the following results:

$$p = \frac{2W + r_2 tcs}{(t_3 - t) r_1 c}$$

$$s = \frac{2(M - W r_4 c)}{r_2 r_3 t c^2}$$

$$\alpha = \mathrm{acos}\left(\frac{s - np}{s + np}\right)$$

Where:

n = Ratio of elastic modulus of the bolt, Eb, to that of the concrete, Ec:

$$n = \frac{Eb}{Ec}$$

t_3 = Width of the basering, similar to the cantilever length, l, in Jawad and Farr's thickness equation previously mentioned

c = Bolt circle diameter

r_1 - r_4 = Four constants based on the neutral axis angle and defined in Singh and Soler's equations 20.3.12 through 20.3.17, not reproduced here.

These equations give the required seven non-linear equations to solve for seven unknowns, namely *p*, *c*, α, and the *ri* (*i* = 1 - 4) parameters. The iterative solution starts with assumed values of *s* and *p,* s_o and p_o, taken from an approximate analysis performed first. Then α is determined using the above equation. Knowing α the dimensionless parameters r_1, r_2, r_3, and *r4* are computed. This enables computation of corrected values of *p* and *s,* named p_o' and s_o'). The next iteration is started with s_1 and p_1 where we choose:

$$s_1 = 0.5\,(s_o + s_0')$$

$$p_1 = 0.5\,(p_0 + p_0')$$

This process is continued until the errors *ei* and *Ei* at the iteration stage are within specified tolerances --*ei = Ei = 0.005* is a practical value,

Where:

$$e_i = \frac{s_i' - s_i}{s_i}$$

$$E_i = \frac{p_i' - p_i}{p_i}$$

After the new values of bolt stress and bearing pressure are calculated, the thickness of the base ring is calculated again using the same formula given above for the approximate method.

Thickness of Basering under Tension - On the tensile side, if there is no top ring but there are gussets, then there is a discrepancy on how to do the analysis. For example, while Megyesy uses Table F (*Pressure Vessel Handbook,* pg 78) to calculate an equivalent bending moment, Dennis R. Moss uses the same approach but does not give a table (*Pressure Vessel Design Manual,* pg 126-129), and Jawad & Farr use a 'yield-line' theory (*Structural Analysis and Design of Process Equipment,* pg 435-436). Since the Jawad and Farr equation for thickness, t, is both accepted and explicit, the program uses their equation 12.13:

Thickness, t:

$$t = \sqrt{\frac{3.91\ P_t}{s_y * (x + y + z)}}$$

Where:

Bolt Load, Pt:

$$P_t = s_{bolt,allow} * A_{bolt}$$

$$x = 2 * \frac{b}{a}$$

$$y = \frac{a}{2l}$$

$$z = d\ \left(\frac{2}{a} + \frac{1}{2l}\right)$$

s_y = Yield strength of the bolt

a = Distance between gussets

b = Width of base plate that is outside of skirt

l = Distance from skirt to bolt area

d = Diameter of bolt hole

Thickness of Top Ring under Tension - If there is a top ring or plate, its thickness is calculated using a simple beam formula. Taking the plate to be a beam supported between two gussets with a point load in the middle equal to the maximum bolt load, we derive the following equation:

Thickness, t:

$$t = \sqrt{6 * \frac{M}{s * Wt}}$$

Where:

Allowable stress, s:

$$s = 1.5\ s_{plate,allow}$$

Bending moment, M:

$$M = \frac{2 * Pt * Cg}{8}$$

Where:

Cg = Center of gravity, depending on the geometry of the plate

Bolt Load, Pt:

$$P_t = \frac{Allowable\ Stress}{Area}$$

Section Modulus, Z:

$$Z = \frac{W_t * t^2}{6.0}$$

Width of Section, Wt:

$$W_t = \left(\frac{D_o}{2}\right) - \left(\frac{D_s}{2}\right) - db$$

Required Thickness of Gussets in Tension - If there are gussets, they must be analyzed for both tension and compression. The tensile stress, T, is the force divided by the area, where the force is taken to be the allowable bolt stress times the bolt area, and the area of the gusset is the thickness of the gusset, t_{gusset}, times one half the width of the gusset, W_{gusset} (because gussets normally taper):

$$T = \frac{P}{A_{gusset}}$$

Where:

$$P = s_{bolt,allow} * A_{bolt}$$

$$A_{gusset} = t_{gusset} * \frac{1}{2} W_{gusset}$$

Required Thickness of Gussets in Compression - In compression (as a column) we must iteratively calculate the required thickness. Taking the actual thickness as the starting point, we perform the calculation in AISC 1.5.1.3. The radius of gyration for the gusset is taken as 0.289 t per Megyesy's *Pressure Vessel Design Handbook,* page 404. The actual compression is calculated as described above, and then compared to the allowed compression per AISC. The thickness is then modified and another calculation performed until the actual and allowed compressions are within one half of one percent of one another.

Basering Design - When you request a basering design, the software performs the following additional calculations to determine the design geometry:

- **Selection of Number of Bolts**

  This selection is made on the basis of Megyesy's table in *Pressure Vessel Handbook* (Table C, page 67). Above the diameter shown, the selection is made to keep the anchor bolt spacing at about 24 inches.

- **Calculation of Load per Bolt**

  This calculation of load, Pb, per bolt:

$$P_b = -\frac{W}{N} + \frac{2M}{NR}$$

Where:

W = Weight of vessel

N = Number of bolts

R = Radius of bolt area

M = Bending moment

- **Calculation of Required Area for Each Bolt**

  This is the load per bolt divided by the allowable stress:

$$A_r = \frac{P}{s}$$

- **Selection of the Bolt Size**

  The software has a table of bolt areas and selects the smallest bolt with area greater than the area calculated above.

- **Selection of Preliminary Basering Geometry -** The table of bolt areas also contains the required clearances in order to successfully tighten the selected bolt (wrench clearances and edge clearances). The software selects a preliminary basering geometry based on these clearances. Values selected at this point are the bolt circle, base ring outside diameter, and base ring inside diameter.

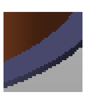
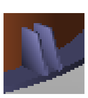
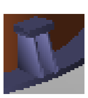
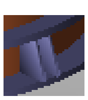
Analysis of Preliminary Basering Geometry - Using the methods described previously for the analysis section, the software determines the approximate compressive stress in the concrete for the preliminary geometry.

Selection of Final Basering Geometry - If the compressive stress calculated above is acceptable then the preliminary geometry becomes the final geometry. If not, then the bolt circle and base ring diameters are scaled up to the point where the compressive stresses are acceptable. These become the final base ring geometry values.

Analysis of Basering Thicknesses - The analysis then continues through the thickness calculation described above, determining required thicknesses for the basering, top ring, and gussets.

Basering Design Data Tab (Basering Dialog Box)

Basering Type - Select the type of basering design:

-  - Simple basering (without gussets)
-  - Basering with gusset plates
-  - Basering with gussets and top plate
-  - Basering with gussets and continuous top ring

Set Default Dimensions - Set default dimensions for all properties in the **Basering** dialog box based on the selected **Basering Type.**

Design Temperature - Enter the design temperature of basering. This value is used to determine the allowable stress values. When you select a value for **Basering Material**, the software uses this temperature to determine the operating allowable stress value for the material.

Basering Design Option - Select a value to either analyze an existing basering or design a new basering. Select **Analyze**, **Design**, **Brn & Young Analyze**, **Brn & Young Design**, or **ADM Analyze**. When a design option is selected, the software may change **Number of Bolts, Nominal Bolt Diameter, Basering ID, and Basering OD.**

Method for Thkness Calc - Select a method for basering thickness calculation:

- **Simplified or Steel on Steel** - Use for baserings located on a steel substructure.
- **Neutral Axis Shifted** - Use to design a thinner basering than the simplified method.

For a traditional basering on concrete, either method can be used.

E for Plates - If a basering with gussets is selected, enter the elastic modulus *E*, used to determine the allowable stress for plates in compression according to AISC. This is a required value. For most common steels, this value is 29 x 10^6 psi.

Sy for Plates - Enter the yield stress for the gusset plates at the design temperature of the base. For tables of yield stress versus temperature, see the ASME Code, Section II, Part D.

Gusset Thickness - Enter the thickness of the gusset plates used for this basering. Any allowances for corrosion should be considered.

Dist. Between Gussets - Enter the distance between the gussets. This dimension is used by the software to calculate the bending moment in the top plate. After the bending moment and bending stress are calculated, the software calculates the required thickness of the top plate.

Bottom Gusset Width - Enter the average width of the bottom gusset plate.

Top Gusset Width - Enter the average width of the top gusset plate.

Height of Gussets - Enter the gusset dimension from the basering to the top of the gusset plate. The forces in the skirt are transmitted to the anchor bolts through the gussets.

Top Plate Thickness - Enter the plate thickness for a basering with a top ring. If a value greater than **0.0** is entered, the software calculates the required thickness of the top plate. If no value is entered, the software does not perform top ring thickness calculations.

Top Plate Width - Enter the plate width for a basering with a top ring. This value is usually equal to the distance between the gusset plates plus two times the gusset plate thickness plus any additional width beyond the gussets.

Radial Width of Top Plate - Enter the radial width if the basering has a top plate. This value is: (top ring OD - top ring ID) / 2. This value must be entered and must be positive.

Bolt Hole Dia in Plate - Enter the diameter of the bolt hole that is in the plate.

Basering Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Basering Thickness - Enter the actual thickness of basering, not including allowances for corrosion or mill tolerance.

NOTE The software calculates the required basering thickness based on the selection in **Method for Thkness Calc**. The value enter here is only used for comparison.

Basering ID - Enter the inside diameter of the basering. The value must be greater than 0 and less than **Bolt Circle Diameter** and **Basering OD**. Enter a good approximation when **Analyze** or **Design** is selected for **Basering Design Option**. If you select **Design** for **Basering Design Option**, the software may change this value.

Basering OD - Enter the outside diameter of the basering. The value must be greater than **Bolt Circle Diameter** and **Basering ID**. Enter a good approximation when **Analyze** or **Design** is selected for **Basering Design Option**. If you select **Design** for **Basering Design Option**, the software may change this value.

Bolt Circle Diameter - Enter the diameter of the bolt circle. This is the diameter passing through the center of each bolt on the basering. The value must be greater than **Basering ID** and less than **Basering OD**. If you select **Design** for **Basering Design Option**, the software may change this value.

Corrosion Allowance - If needed, enter an external corrosion allowance. This allowance is added to **Basering Thickness** in the calculations.

Type of Threads - Select the thread series bolt table. Select **TEMA** (8-thread series), **UNC** (coarse threads, adapted from *Mark's*), **BS 3642**, or **SABS 1700 1996.** If you have a bolt that is outside of the bolt table ranges, select **User Root Area** and enter the nominal size in **Nominal Bolt Diameter**. This information can be obtained from a standard engineering handbook.

Nominal Bolt Diameter - Click ... and select the nominal bolt diameter from the bolt table selected for **Type of Threads**. If you have a bolt that is outside of the bolt table ranges, enter the nominal size and select **User Root Area** in **Type of Threads**.

For **TEMA** and **UNC** bolts, the available bolt diameters are:

Bolt Size (inches)	**Bolt Root Area**	
	(cm.^2)	**(in.^2)**
0.500	1.27	0.126
0.625	1.5875	0.202
0.750	1.9050	0.302
0.875	2.22225	0.419
1.000	2.54000	0.551
1.125	2.85750	0.728
1.250	3.17500	0.929
1.375	3.49250	1.155
1.500	3.81000	1.405

Bolt Size	Bolt Root Area	
1.625	4.12750	1.680
1.750	4.44500	1.980
1.875	4.76250	2.304
2.000	5.08000	2.652
2.250	5.71500	3.423
2.500	6.35000	4.292
2.750	6.98500	5.259
3.000	7.62000	6.324
3.250	8.25500	7.487
3.500	8.89000	8.749
3.750	9.52500	10.108
4.000	10.1600	11.566

NOTE This information is adapted from Jawad and Farr, *Structural Analysis and Design of Process Equipment*, pg 425.

Number of Bolts - Enter the number of bolts required by the basering design. The number of bolts can be between 4 and 120. If you select **Design** for **Basering Design Option**, the software may change this value.

NOTE Bolts sizing is based on the maximum load per bolt in the operating case. The computation of the load per bolt is referenced in Jawad and Farr, equation 12.3, pg 422.

Bolt Material - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (on page 485). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (on page 534). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Bolt C.A. (1/2 total) - Enter the bolt corrosion allowance. The software uses this value to corrode the radius of the root area and calculate a corroded root stress area based on the nominal bolt size and bolt table. This area is then used in the remainder of the bolt load/stress calculations.

The software calculates the required area of the bolt. If the bolt corrosion allowance is greater than zero, the software adds the corrosion allowance and recalculates the diameter based on the new required area:

$$\text{Corroded Bolt Root Diameter} = (4 * \text{New Bolt Area/Pi})^{1/2} - 2 * \text{Bolt Corrosion Allowance}$$

Bolt Root Area - Enter the root area of a single bolt if the basering design requires special bolts.

NOTE This option is mutually exclusive from the selection for **Basering Design Option**. If **Bolt Root Area** is available, the numbers from table 2 (UNC) are used. This value is used without modification by the software, so you should consider any corrosion allowance.

Bolt Shear Allowable - The allowable shear stress acting on the bolt, incorporating the bolt corrosion allowance.

Concrete Strength F'c/Fc <and> Modular Ratio Eplates/Ec - Enter the following for the concrete to which the base is bolted:

- **F'_c** - The nominal ultimate compressive stress of the concrete. This value is F'_c in Jawad and Farr or FPC in Meygesy. A typical entry is 3000 psi.
- **F_c** - The allowable compressive stress of the concrete
- **n** - The steel-to-concrete modulus of elasticity ratio, E_{plates}/E_c.

Average Values of Properties of Concrete Mixes (adapted from Brownell and Young)

Water Content (US Gallons per 94 lb Sack of Cement)	**f'_c 28-day Ultimate Compressive Strength (psi)**	**f_c Allowable Compressive Strength = 0.45*f'c (psi)**	**n Modular Ratio (E_s/E_c)**
7.5	2000	800	15
6.75	2500	1000	12
6	3000	1200	10
5	3750	1400	8

NOTE According to Jawad and Farr, Ec is equal to 57000 multiplied by the square root of f'_c psi. The modulus of elasticity of steel is assumed to be 30 x 10^6.

Lug Start Angle - Reassign the reference angle for positioning the lugs.

Use EIL Spec? - Select to use the EIL standard for basering design. This standard gives guidelines for selecting the number of bolts based on skirt diameter, as shown below:

Skirt Diameter (mm)	**Number of Bolts**
900	4
1200	8
1500	8
1800	12

2100	16
2400	16
2700	20
3000	24
3300	24
3600	28
4000	32

NOTE This option only applies if **Design** is selected for **Basering Design Option**.

Bolt Diameter - Enter a value when **Use EIL Spec?** is selected. This value provides additional tabular results for the standard thickness of basering components, such as the basering itself, gussets, top plate, and gusset height.

% Applied to Bolt Area * Bolt Stress - Enter a value for the percentage of force to apply to the bolt area times the bolt allowable stress in the calculation of the concrete stress. The default value of 100 percent generates the largest possible force. If the load cannot achieve this value, enter a smaller percentage.

Use AISI Design Method? - Select to perform calculations according to AISI Volume 2. The AISI (American Iron and Steel Institute) publishes *Steel Plate Engineering Data* volumes containing numerous useful calculations for steel structures including welding and thickness calculations for anchor bolt chairs.

Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1? - Select to use AISC design manual 9th edition, Eqn. F3-1. The AISC (American Institute of Steel Construction) allows the use of 2/3rds the yield stress as the allowable for determination of the required thickness of the basering parts. A normal bending allowable is 1.5 times the allowable stress for the material at design temperature.

NOTES

- This option and **Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1?** cannot be selected at the same time.
- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1? - Select to use AISC design manual 9th edition, Eqn. F2-1. The AISC (American Institute of Steel Construction) allows the use of 75 percent of the yield stress as the allowable for determination of the required thickness of these parts. A normal bending allowable is 1.5 times the allowable stress for the material at design temperature.

NOTES

- This option and **Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1?** cannot be selected at the same time.

- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use 1/3 Increase per ASIC A5.2? - Select to use AISC design manual 9th edition, Section A5.2. The AISC (American Institute of Steel Construction) allows a 1/3 increase in the allowable stress for parts that are subject to short-term sustained loads such as those due to wind or seismic.

NOTES

- This option is used in addition to the selection of **Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1?** or **Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1?**.
- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use Allowable Weld Stress per AISC Table J2.5? - Select to use AISC design manual 9th edition, Table J2.5. The weld allowable stress is then 0.3 times the minimum of the basering ultimate tensile strength and 58000 psi. If this option is cleared, the allowable weld stress is 0.4 times the minimum of the skirt and basering yield stress at ambient temperature.

NOTES

- This option is used in addition to the selection of **Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1?** or **Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1?**.
- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use the skirt stress to determine the concrete stress for the simplified method? - Select this option to use the maximum skirt stress and the following formula to calculate the concrete stress:

$$s_{concrete} = \frac{s_{skirt} * t_{corroded,skirt}}{W_{baseplate}}$$

During the analysis of the skirt, the software calculates combination stresses due to different load types as outlined in the *Stresses due to combined loads* report. This option is only necessary when **Simplified** is selected for **Method for Thkness Calc.**

NOTES

- When **Use the skirt stress to determine the concrete stress for the simplified method?** is selected, **% Applied to Bolt Area * Bolt Stress** is inactive because this option is nearly identical to a percentage of 0.
- If **Neutral Axis Shifted** is selected for **Method for Thkness Calc**, then this option is not needed because the concrete stress is calculated in iterations. The concrete stress is presented for information.

Determine the Basering design bolt load accounting for Load Case Factors? - Select to calculate the bolt load based on moments and load case factors, depending on the local scalars used. The bolt load is calculated by the highest moment due to the bending stress from the different combination of loads.

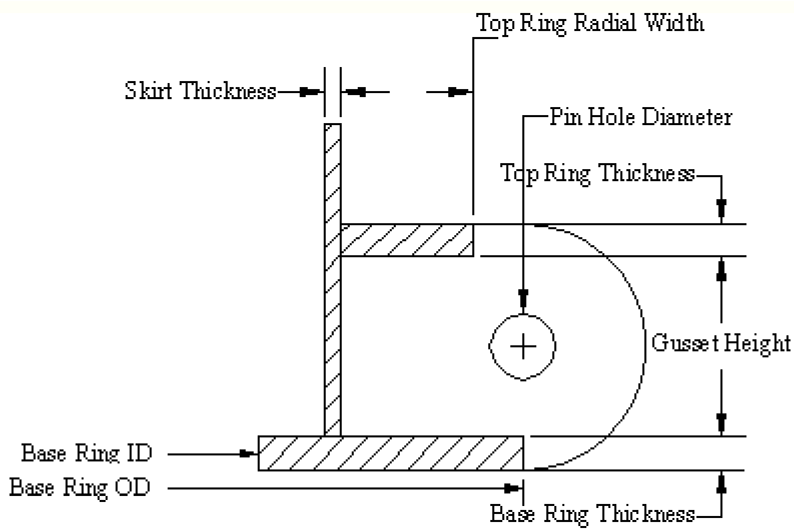
IMPORTANT Be sure to select this option if you have also selected **Use and Allow editing of Local Scalars in the Load Cases** on the **Load Cases** tab.

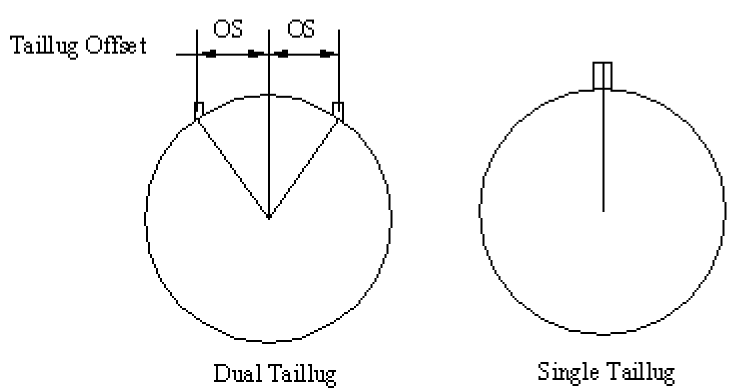
Plot - Click to open the **Basering Graphics** dialog box. A cross-section view of the basering design is shown.

Tailing Lug Data Tab (Basering Design Data Tab)

Tailing lug analysis is included in the basering analysis for a single or dual type design, as shown below. The design is based on a lift position where no bending occurs on the tailing lug. The location of the center of the pinhole is radial at the edge of the outer most of the top ring or the basering, whichever is larger. The tailing lug material is assumed to be the same material as the gusset or basering. The main considerations for the design are section modulus, shear and bearing stress at the pin hole, and weld strength.

NOTE For the software to generate the lifting load on the lug, the rigging data must be entered. To enter the rigging data, click **Installation | Misc. Options** on the **Load Cases** tab of the main window**.**





Perform Tailing Lug Analysis? - Select to enter data for the tailing lug analysis.

Type of Tailing Lug - Select **Single Lug** or **Dual Lug**, as shown above.

Centerline Offset - Enter the offset dimension from centerline *OS*. This entry is only valid for **Dual Lug**.

Lug Thickness - Enter the thickness of the tailing lug.

Pin Hole Diameter - Enter the pinhole diameter. The center of the pinhole is placed radially inline with the larger of the outermost edge of the top ring or the base ring OD.

Weld Size Thickness - Enter the leg size of the weld connecting the lug to the basering and the skirt.

Lug Height - Enter the tailing lug height measured from the top of basering. If you have a top ring, this would usually be the distance to the top ring.

Dist. from Skirt to Hole - Enter the distance from the OD of the skirt to the pin hole.

Analysis Results

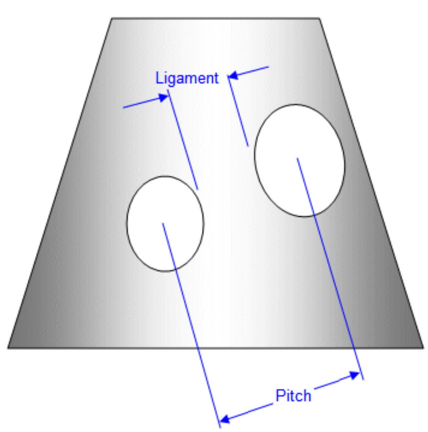
The tail lug design consists of a three-part analysis: the base ring assembly (base ring, skirt and top ring), the strength of the weld, and the tail lug itself. The analysis assumes no bending in the tail lug. In the absent of the top ring, only the base ring and decay length (*e*) are considered for the section modulus calculation.

The following allowable stresses are used to check the design strength.

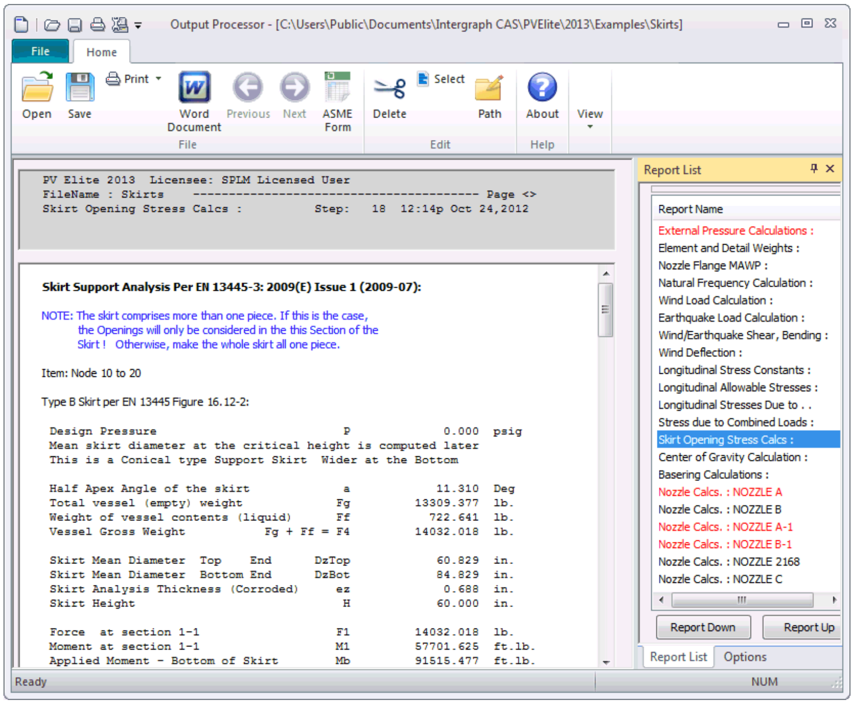
- Shear at the pin hole: 0.4 S_y
- Bearing stress: 0.75 S_y
- Weld stress: 0.49 S_{allow}

Skirt Opening Analysis Considerations

Pitch and Ligament - Pitch is the center-to-center distance between adjacent holes. **Ligament** is the edge-to-edge distance between holes.



If the ligament is too small there is a possibility of collapse and buckling of the skirt. EN 13445, the code chosen as the basis for the analysis, is silent on this issue. This is because EN 13445 only gives consideration to one hole not the multiple holes that may be analyzed by PV Elite. The output processor includes a **Skirt Opening Stress Calcs** report. That report contains the full analysis results for all the openings that were included in the **Skirt Access Openings** dialog box:



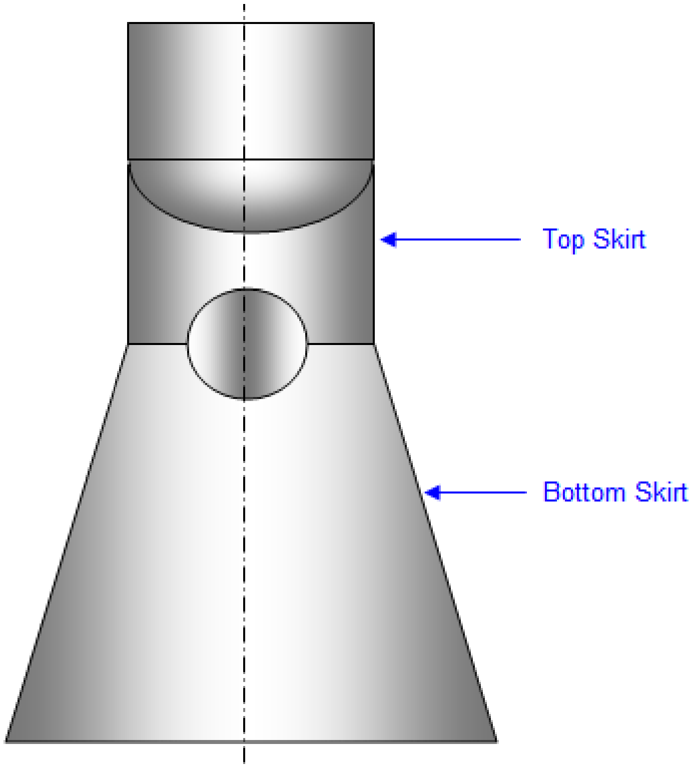
IMPORTANT The results give a recommended minimum ligament distance, but you must evaluate whether this meets project requirements. See the following example:

```
Summary of Stresses:
Stress              Actual          Allowed       Result    Equation
                     psi              psi                   Eqn 16.12
----------------------------------------------------------------
Stot1pi            7808.51         42120.57        Pass       46
Stot1po            2251.14         42120.57        Pass       47
Stot1qi            9645.18         42442.72        Pass       48
Stot1qo           -1219.34         42442.72        Pass       49
Stot2pi           10717.20         42335.75        Pass       50
Stot2po           -1715.91         42335.75        Pass       51
Stot2qi            8063.63         42335.75        Pass       52
Stot2qo             937.67         42335.75        Pass       53
Stot3pi           -1338.29         43186.85        Pass       54
Stot3po            2448.57         43186.85        Pass       55
Stot3qi             727.67         43196.11        Pass       56
Stot3qo           -1331.37         43196.11        Pass       57

Skirt stress at the openings / critical section:
Sm4p                101.72         15000.00        Pass       70
Sm4q               -617.04         15000.00        Pass       71

The stresses are satisfactory

Analysis according to EN 13445 Section 8.7 allowable compressive stress [fc]:
 Note: This is a supplementary calculation not required by the code
 Py    = 2 * fz * ez / R = 2 * 15000.000 * 0.833 / 25.585 = 977.12 psig
 Pm    = 1.21*E*(ez/R)² = 1.21*29 *10^6*(0.833 /25.585 )² = 37225.29 psi
 Pm/Py = 37225.293 / 977.122 = 38.097
```

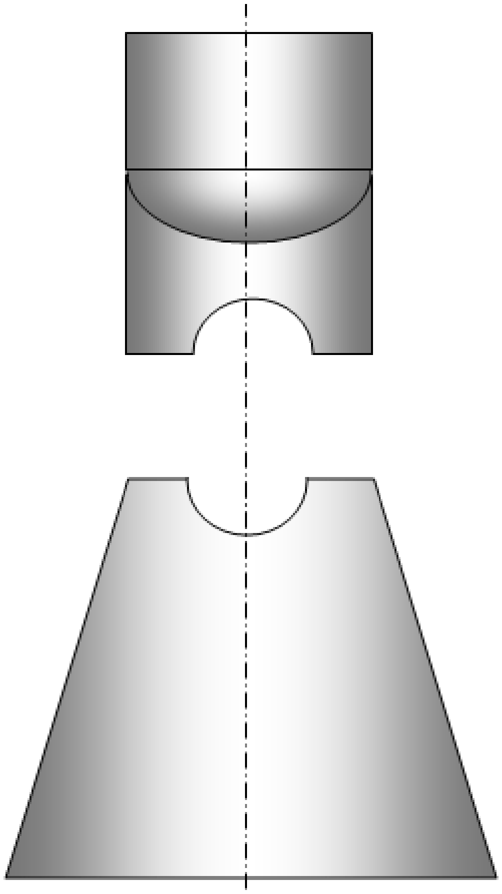
Vessels with Multiple Skirt Elements - Complications arise when:

- A skirt is made from two pieces. That is, two skirts support the vessel.
- The two skirts have different thicknesses.
- The two skirts have different materials.
- The two skirts have different taper (conical) angles.

- An access opening or hole spans the skirt pieces. The opening cannot be considered as a single opening.

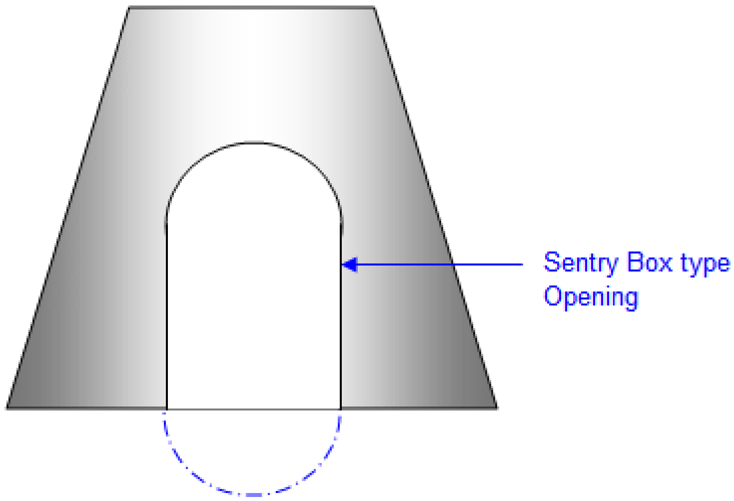


To properly analyze these situations, each skirt is created as a separate element:



1. In the bottom skirt, enter the hole geometry. The height of the center of the hole is located at or near to the top of the skirt and the top half of the hole is outside of the top node of the skirt. The software adjusts the analysis for this, but warns you that the opening is beyond the skirt.
2. In the top skirt, enter the hole geometry. In this case the height of the center of the hole is at or near zero at the bottom of the skirt. Again, the software adjusts the analysis for this, but warns you that the opening is beyond the skirt.

Sentry Box Openings - The software allows entry of oblong holes. With an oblong hole, sentry box geometry is possible:



The bottom of the oblong opening breaks the bottom of the skirt. The software adjusts the analysis for this, but warns you that the opening is beyond the skirt.

Basic Skirt Thickness - The required thickness of the skirt under tension and compression loads is determined using the same formula used for the compressive stress in the concrete, except using the thickness of the skirt rather than the width of the basering:

$$s = -\left(\frac{W}{A} + \frac{Mc}{I}\right)$$

Where:

W = Weight of the vessel during operation or testing

M = Maximum bending moment on the vessel

A = Cross-sectional area of the skirt

c = Distance from the center of the basering to the skirt (radius of the skirt)

I = Moment of inertia of the skirt cross section

In tension, this actual stress is simply compared to the allowable stress, and the required thickness can be calculated directly by solving the formula for t. In compression, the allowable stress must be calculated from the ASME Code, per paragraph UG-23, where the geometry factor is calculated from the skirt thickness and radius, and the materials factor is found in the Code external pressure charts. As with all external pressure chart calculations, this is an iterative procedure. A thickness is selected, the actual stress is calculated, the allowable stress is determined, and the original thickness is adjusted so that the allowable stress approaches the actual stress.

Stress in Skirt Due to Gussets or Top Ring - If there are gussets or gussets and a top ring included in the base plate geometry, there is an additional load in the skirt. Jawad and Farr (Structural Analysis and Design of Process Equipment, pg 434) have analyzed this load and determined that the stress in the skirt, s, due to the bolt load on the base plate is calculated as follows:

$$s = \frac{1.5F\,b}{\pi t^2 h}$$

Where:

F = Bolt load

b = Width of the gusset at its base

t = Thickness of the skirt

h = Height of the gusset

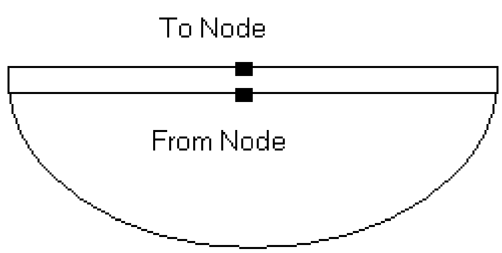
Jawad and Farr note that this stress should be combined with the axial stress due to weight and bending moment and should then be less than three times the allowable stress. They thus categorize this stress as secondary bending. The software performs the calculation of this stress, and then repeats the iterative procedure described above to determine the required thickness of the skirt at the top of the basering.

Skirt Access Openings Dialog Box

Defines data for access openings on a skirt element.

From Node - Displays the software-generated node number describing the starting location of the element. The **From Node** value for this element is also used to define starting locations for details such as nozzles, insulation, and packing that are associated with this element.

The software defines a vertical vessel from bottom to top. If the vertical vessel is on skirt, the first element is the skirt. If it is on legs or lugs, the first element is a head and the legs or lugs are defined as details on the appropriate head or shell elements.

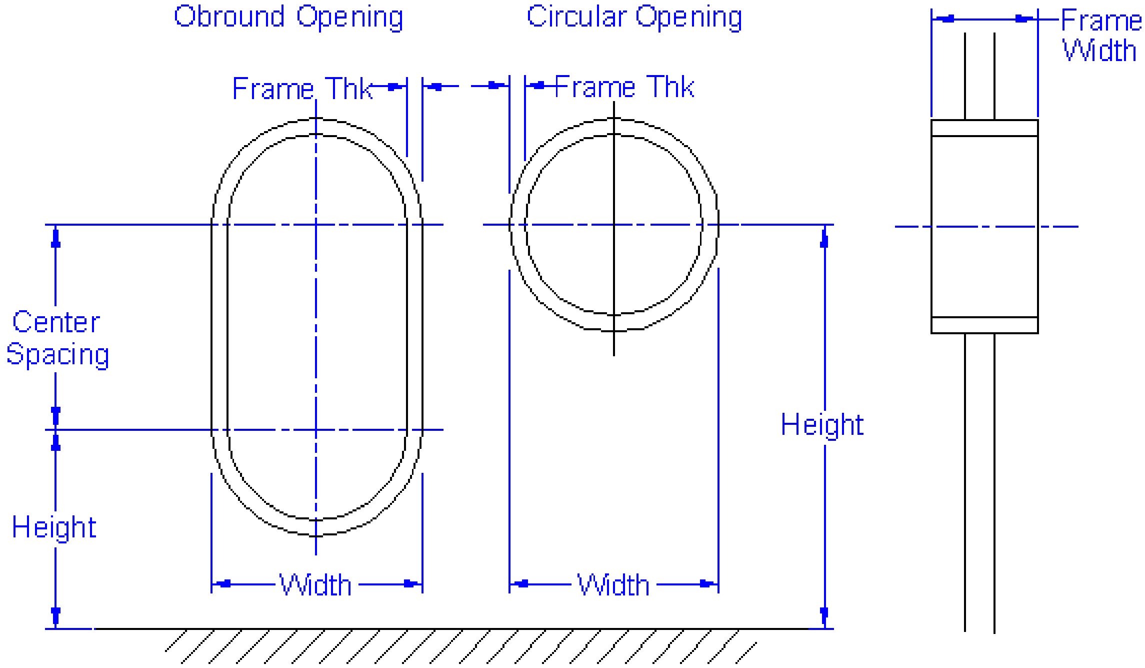


The software defines a horizontal vessel from left to right. The first element in a horizontal vessel is usually a head, and the support saddles are defined as details on the appropriate shell elements.

NOTE This value is not a function of the selected vessel code (such as, PD:5500, EN or ASME).

Number of Skirt Openings - Select the number of holes in the skirt. You can define up to ten holes.

Width - Enter the width of the opening.



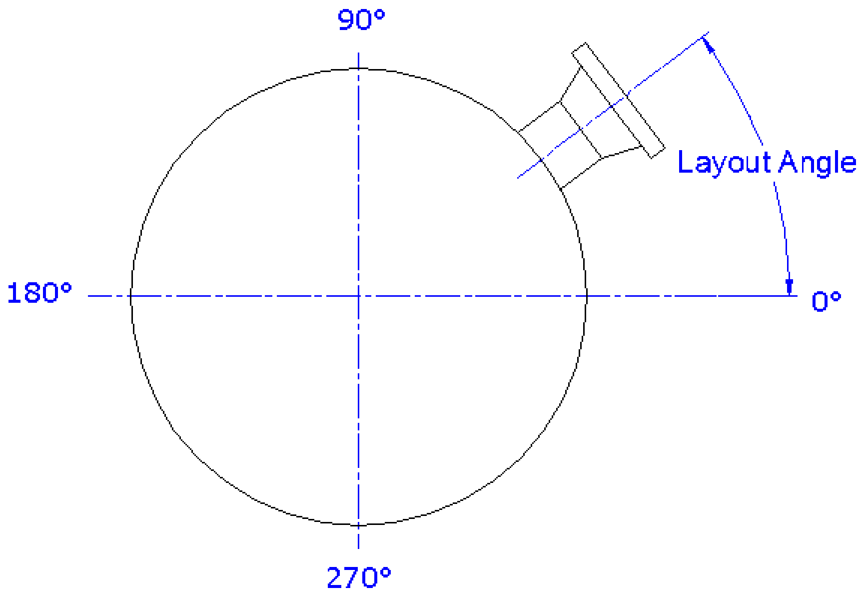
Height - Enter the distance from the base of the skirt to the centerline of the opening.

Center Spacing - Enter the distance between centerlines on a rounded oblong opening.

Frame Thk - Enter the thickness of the opening reinforcement frame.

Frame Width - Enter the depth of the opening reinforcement frame.

Layout Angle - Enter the location of the opening around the skirt, when looking at a plan view of the skirt.



References

1. *Boiler & Pressure Vessel Code: Section VIII.* [S.I.]: ASME Publications, 2010. Print. Div. 1.
2. *Boiler & Pressure Vessel Code: Section VIII.* [S.I.]: ASME Publications, 2010. Print. Div. 2.
3. Brownell, Lloyd Earl, and Edwin H. Young. *Process Equipment Design: Vessel Design*. New York: Wiley, 1959. Print.
4. Farr, James R., and Maan H. Jawad. *Guidebook for the Design of ASME Section VIII Pressure Vessels*. New York: ASME, 1998. Print.
5. Jawad, Mann H., and James R. Farr. *Structural Analysis and Design of Process Equipment.* Second ed. New York: Wiley, 1989. Print.
6. Megyesy, Eugene F. *Pressure Vessel Handbook*. Eighth ed. Tulsa, OK: Pressure Vessel Handbook Pub., 1986. Print.
7. Moss, Dennis R. *Pressure Vessel Design Manual: Illustrated Procedures for Solving Major Pressure Vessel Design Problems*. Amsterdam: Gulf Professional Publ, 2004. Print.
8. Singh, Krishna P., and Alan I. Soler. *Mechanical Design of Heat Exchangers: and Pressure Vessel Components*. Cherry Hill: Arcuturus, 1984. Print.

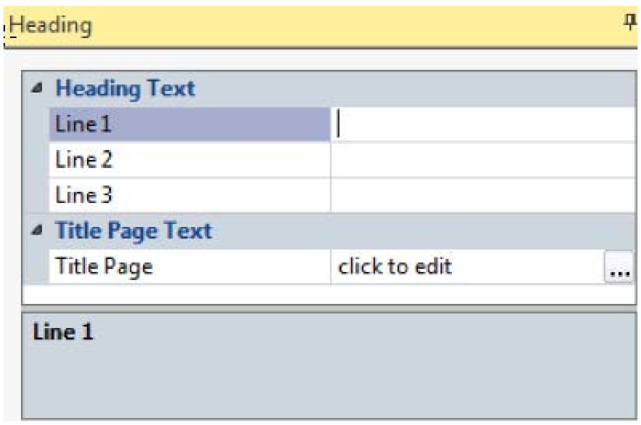
SECTION 12

Report Headings (Heading Tab)

Page heading text and cover sheet text for reports are entered on the **Heading** tab. You can enter up to three lines of heading text, to appear at the top of each page, and the text to appear on a separate cover sheet.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > Heading**.

   *The **Heading** tab appears at the bottom-left of the PV Elite window.*



Heading Text - Enter up to three lines of text to appear at the top of each printed page of the output report.

Title Page Text - Click ... to open the **Title Page Text** dialog box and enter text to appear on the cover page of the output report.

Set Title - Click to insert a default title page in the **Title Page Text** dialog box. The default title page is stored in the file *Title.Hed*, in the software subdirectory. You can edit the file.

SECTION 13

Design Constraints Tab

Enter data such as pressures and temperatures, hydrotest data, and wall thicknesses in the **Design Constraints** tab. If no data is entered, the software uses the system defaults.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > Design Constraints**.

   *The **Design Constraints** tab appears at the bottom-left of the PV Elite window.*

In This Section

Design Data (Design Constraints Tab)

Vessel analysis uses the following set of design parameters.

Design Internal Press - Enter the design internal pressure for the vessel. This value is used as general design data and also with the UG-99b note 34 type hydrotest.

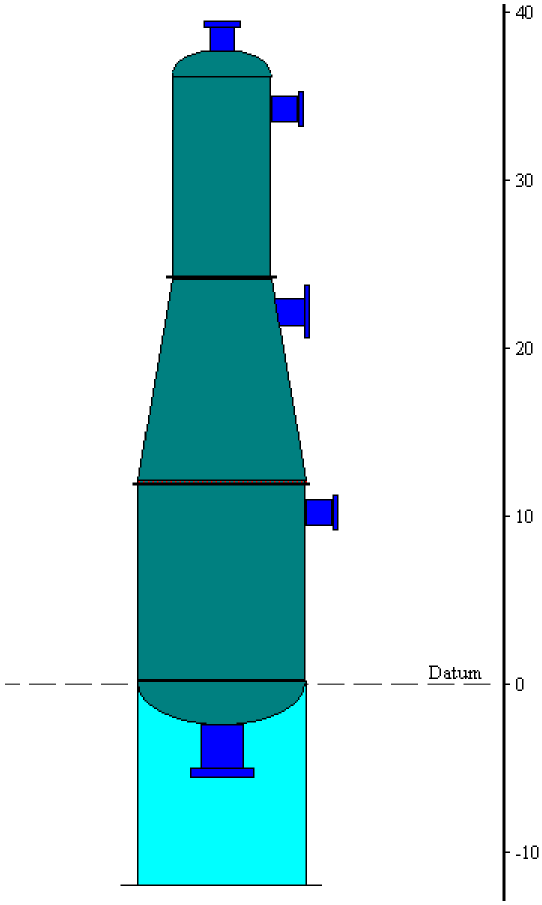
Design External Press - Enter the design external pressure for the vessel if the vessel is required to be rated for vacuum conditions.

Design Internal Temp - Enter the design internal temperature for the vessel. This value is only used by the input echo to help insure the correct design data was entered. It is not used by the analysis portion of the software.

Design External Temp - Enter the design external temperature for the vessel if the vessel is required to be rated for vacuum conditions.

Datum Line Options - Click ... to display the **Datum Line Options** dialog box. For more information, see *Datum Line Options Dialog Box* (on page 332).

Enter the location of the datum line for the **From Node** of the first element. You can then use **List Dialog** on the **Auxiliary** toolbar to enter the distance of nozzles and platforms from the datum line.



Select the type of hydrotest. For Division 1, the software provides three different methods to determine hydrotest pressure. For Division 2, the program provides two hydrotest methods. Select one of the following:

- **UG-99b** - ASME UG-99 (b), Division 1. The hydrotest pressure is 1.3 times (1.5 for pre-99 addenda) the maximum allowable working pressure for the vessel multiplied by the lowest ratio of the stress value S for the test temperature to the stress value S for the design temperature. This type of hydrotest is normally used for non-carbon steel vessels where the allowable stress changes with temperature, starting even at a somewhat low temperature.
- **UG-99c** - ASME UG-99(c), Division 1. The hydrotest pressure is determined by multiplying the minimum MAP by 1.3 (1.5 for pre-99 addenda) and reducing this value by the hydrostatic head on that part. The hydrostatic head is calculated based on the dimensions of the vessel and by values for **Projection from Top, Projection from Bottom, and Projection from Bottom Oper**. In addition, **Hydrotest Position** is used to determine the head pressure.
- **UG-99b(34)** - ASME UG-99(b), footnote 34, Division 1. The hydrotest pressure is 1.3 times (1.5 for pre-99 addenda) the design pressure for the entire vessel, multiplied by the lowest ratio of the stress value Sa for the test temperature to the stress value S for the design temperature.

- **UG-100** - ASME UG-100 pneumatic test. The test pressure is 1.1 times (1.25 for pre-99 addenda) the stated design pressure for the entire vessel, multiplied by the lowest ratio of the stress value Sa for the test temperature to the stress value S for the design temperature.

  NOTE The stress ratio mentioned above includes bolt allowable stresses for flanges that are designed according to Appendix 2. This will usually result in a ratio of 1. See ASME Interpretation VIII-1-83-260 for more information. Click **Tools** > **Configuration** to turn off this option, resulting in a ratio greater than one in cases where the operating and ambient stresses for the vessel parts are not the same.

- **AT-300** - ASME AT-300, Division 2, based on vessel design pressure. The hydrotest pressure is 1.25 times the design pressure to be marked on the vessel, multiplied by the lowest ratio of the stress intensity value Sm for the test temperature to the stress intensity value Sm for the design temperature. This type of hydrotest is normally used for non-carbon steel vessels where the allowable stress changes with temperature starting even at a somewhat low temperature.
- **AT-301** - ASME AT-301, Division 2, based on calculated pressure. A hydrostatic test based on a calculated pressure may be used by agreement between the user and the manufacturer. The hydrostatic test pressure at the top of the vessel is the minimum of the test pressures calculated by multiplying the basis for calculated test pressure for each element by 1.25 and reducing this value by the hydrostatic head on that element.
- **No Hydro** - No hydrotest pressure.
- **User Entered Pressure**
- **1.43 * MAWP (PED)**

Hydrotest Position - Select a hydrotest position. This input is required so that the total static head can be determined and subtracted when **UG-99c** is selected for **Hydrotest Type**. This value is used in conjunction with **Projection from Top**, **Projection from Bottom**, and **Flange Distance to Top** to determine the total static head. Select from the following:

- **Vertical** - The vessel is tested in the upright or vertical position. This is not common.
- **Horizontal** - The vessel is tested in the horizontal position. This is common for most vessels. The vessel is on its side (in the case of a vertical vessel) or in its normal position (for a horizontal vessel).

Projection from Top - Enter the projected distance of the nozzle from the outer surface of the vessel *in its test position* to the face of the highest flange. This distance is added to the height (for vertical test positions) or to the maximum diameter of the vessel (for horizontal test positions) to determine the static head when **UG-99c** is selected for **Hydrotest Type**.

Projection from Bottom - Enter the projected distance of the nozzle from the outer surface of the vessel *in its test position* to the face of the lowest flange. This distance is added to the height (for vertical test positions) or to the maximum vessel diameter (for horizontal test positions) to determine the static head when **UG-99c** is selected for **Hydrotest Type**. This distance is essential when a nozzle flange projecting from the bottom of the vessel is governing the analysis.

Projection from Bottom Ope - Enter the projected distance of the nozzle from the outer surface of the vessel to the face of the flange. This distance is used to calculate the MAWP of an ANSI flange when it governs the MAWP. If the ANSI flange governs, then the static operating liquid head is subtracted from the basic rating of the flange.

Min. Des Metal Temp - Enter the specified minimum design metal temperature for the vessel. The software does not design any components based on this value. It is merely to help document the calculations and to be used for reference. This value is listed in the Internal Pressure Calculations report for comparison with the calculated UCS-66 minimum temperature.

No UG-20(f) Exemptions - Select this option if the vessel cannot take exemptions according to UG-20(f), for ASME VIII-1.

Flange Distance to Top - Enter the distance from the centerline/face of flange to the top of the vessel. This value is used when the flanges govern the MAP of the vessel. If the vessel is in the vertical position and **UG-99c** is selected for **Hydrotest Type**, this value is used in conjunction with **Projection from Top** to determine the total static head to subtract for the c type hydrostatic test.

Construction Type - Select the type of construction to be included on the name plate. Select **Welded**, **Press. Welded** (Pressure welded), **Brazed**, or **Resist. Welded** (Resistance welded).

Service Type - Select a type of special service in which the vessel is used. Select **None**, **Lethal** (Lethal service), **Unfired Steam** (Unfired steam boiler), **Direct Firing**, **Nonstationary**, **Air/Water/Steam**, **Sour**, **Severe Sour**, or **Amine.** This value is for information only; it is reported in the input echo.

Degree of Radiography - Select the symbolic representation of the degree of radiography. Select one of the following:

- **RT 1** - The complete vessel satisfies the full radiography requirements of UW-11(a) and the spot radiography provisions of UW-11(a)(5)(b) have not been applied.
- **RT 2** - The complete vessel satisfies the full radiography requirements of UW-11(a)(5) and the spot radiography provisions of UW-11(a)(5)(b) have been applied.
- **RT 3** - The complete vessel satisfies the spot radiography requirements of UW-11(b).
- **RT 4** - Only part of the vessel has met the other category requirements, or none of the other requirements are applied.
- **None**

This value is for information only; it is reported in the input echo.

NOTES

- **PD-5500 Non-destructive Testing Category 1**: 100% non-destructive testing (NDT) per section is required according to 5.6.4.1. There are no restrictions for vessels built under this category. The NDT is either (generally) radiography, or ultrasonic testing. The code must be consulted in all matters pertaining to this category. This is the strictest category.
- **PD-5500 Non-destructive Testing Category 2**: Limited random spot NDT according to section 5.6.4.2 is required. Generally about 10% of the welds must be examined by NDT (radiography or ultrasonic). There are restrictions when employing this category. Only certain groups of materials may be used (for example, groups 1, 2 and 4). The maximum thickness of the main components must not exceed 40 mm or 30 mm depending on the material group. It is essential that Table 3.4-1 be consulted when employing category 2. The code must be consulted in all matters pertaining to this category.
- **PD-5500 Non-destructive Testing Category 3**: Only visual inspection of the weld is required. However, steels are restricted to C and CMn types and materials in group 8. The maximum thickness is restricted to 13 mm for C and CMn steels, and 25 mm for materials in group 8. The design stress is much lower (see 3.4.2.2 for details). This category is restricted to temperatures no lower than 0°C. The code must be consulted in all matters pertaining to this category.

Miscellaneous Weight % - Enter a percentage value to include additional weight to account for vessel attachments and internal items not otherwise included in the vessel. Typical values are **3.0** or **5.0**. The software multiplies the total weight of the vessel by 1.0 plus this value converted to a decimal (such as 1.03 or 1.05). Enter **0** if no additional weight is needed.

Design Code - Displays the material design code for the project. This value is selected when you create a new project with **File** > **New** and can be changed by selecting a new value for **Design/Code** on the **Units/Code** toolbar. Available codes are **ASME SectionVIII-Divison 1**, **ASME SectionVIII-Divison 2**, **British Standard PD:5500**, or **EN-13445**.

User Defined MAWP - Enter a manually-defined value for maximum allowable working pressure to override the software-generated value. The software-generated MAWP is based on pressure ratings for the elements and ANSI flanges. If this value is zero, the software-generated MAWP is used. This is the default behavior.

User Defined MAPnc - Enter a manually-defined value for MAPnc to override the software-generated value. The software-generated MAPnc is based on pressure ratings for the elements and ANSI flanges. If this value is zero, the software-generated MAPnc is used. This is the default behavior.

User Defined Hydro. Press - Enter a manually-defined value for hydrostatic test pressure to override the software-generated value. The value is then used to calculate the stresses on elements subjected to this pressure. If this value is greater than 0, the software uses this pressure plus the applicable hydrostatic head that is computed based on the hydrotest position. If this value is zero, the software-generated hydrostatic test pressure is used. This is the default behavior.

Additional Ope. Static Press - Enter the additional static pressure at the top of the vessel.

Use Higher Long. Stress - Select to use higher allowable stresses for longitudinal stress calculations for wind and earthquake loadings. Loads are increased by an occasional load factor of 1.2.

ASME Section VIII, Division 1, Paragraph UG-23(d) permits the allowable stress for the combination of earthquake loading, or wind loading with other loadings to be increased by a factor of 1.2.

ASME Section VIII Division 2 A08 does not explicitly allow for an increase in allowable stresses, but this is subject to change.

Some wind and earthquake codes such as ASCE-7 98 and IBC 2000 and later have explicitly defined load combinations embedded in the standard. This set of codes essentially uses the same methodology to define earthquake loads. The load combinations either divide the earthquake load by a factor of 1.4 or multiply by 0.7 to convert from Limit State to Allowable Stress design. ASCE-7 2005, Chapter 13, paragraph 13.1.7 states that when a reference document "ASME Section VIII Division 1 in this case" provides a basis for earthquake design (UG-22), the reference document is used. This paragraph goes on to state that the loads shall be multiplied by 0.7. Please note that the IBC codes point directly to ASCE for wind and seismic load calculations.

NOTES

- In PV Elite versions prior to 2009, the allowables used for longitudinal stress calculations did not allow the 1.2 increase in tensile and compressive allowables for earthquake load cases, even if this option was selected. This restriction was removed in the 2009 version.
- For PD 5500 and EN-13445 the occasional load factor is not applied.

Consider Vortex Shedding - Select to perform vortex shedding calculations on tall, slender vertical vessels susceptible to wind-induced oscillations. This method is documented in the National Building Code of Canada and in texts on wind engineering. The software calculates fatigue stresses based on loads generated by wind vibration and the number of hours of safe operation remaining under vibration conditions. The software calculates the likelihood of this occurrence based on the research of Zorilla and Mahajan. If there is no possibility of wind vibration and you have this option selected, the software warns you that unrealistically high stresses will result and gives you the option to turn this calculation off.

Is This a Heat Exchanger? - If the Dimensional Solutions 3D file interface button is selected, also select this option to write geometry and loading information for this vessel design to the *<jobname>*.ini file created in the current working directory. See *Dimensional Solutions http://www.Dimsoln.com* for more information about the Dimensional Solutions product line. This entry is optional.

NOTE To completely define an exchanger it is necessary to enter in the required information regarding the tubes, tubesheets and the floating head (if any). With the exchanger data, PV Elite can then compute the weights and required thicknesses of the exchanger components. For more information, see **Tubesheet** (on page 124).

Corroded Hydrotest - Select to use the corroded wall thickness when calculating stresses on the elements during the hydrotest, when it is necessary to hydrotest the vessel after it has corroded. If cleared, the software uses the uncorroded wall thickness

NOTE Longitudinal stresses due to hydrostatic test pressure are also calculated using the corroded wall thickness when this option is selected.

Hyd. Allowable is 90% Yield - Select to use 90 percent of the ambient yield stress as the hydrotest allowable stress. Clear to use the ASME Division 1 value, which is 1.3 times the ambient allowable stress *Sa* for the material. When the vessel is tested, the largest circumferential stress should not exceed this value. The software recalculates the hydrotest allowable each time this option is selected or cleared.

ASME Steel Stack - Select to perform an ASME steel stack analysis, based on the ASME recommended guidelines for Steel Stacks STS-2000 with addenda. This analysis is for circular stacks that meet the design requirements in the steel stack guidelines. The results are shown in the *ASME STS Stack Calculations* report. If **Design Code** is not set to **Division 1** (ASME VIII-1), the stack analysis is not performed.

Also select this option if you are analyzing a steel stack and want to check it against ANSI/ASME STS-2000/STS-1a-2003. After the software completes the calculation, the program generates the *Stress Due to Combined Loads* report with a listing of the stack calculations. Compressive allowables in the report are calculated based on Section 4.4.

When selected, expand **ASME Steel Stack** and enter values for **ASCE Wind Exposure**, **Factor of Safety**, **Mean Hourly Wind Speed**, **Is the Stack Lined?**, and **Importance Factor**.

IMPORTANT Read and understand the ASME stack guidelines. This is not a code like ASME Division 1 or 2, but a set of design guidelines for designers and engineers.

The following paragraphs from the stack guidelines are addressed:

- 4.4 Allowable Stresses
- 4.4.1 Longitudinal Compression, equations 4.7,4.8 and 4.9
- 4.4.2 Longitudinal Compression and Bending
- 4.4.3 Circumferential Stresses

- 4.4.4 Combined Longitudinal and Circumferential Compressive Stresses
- 4.4.5 Circumferential Compression in Stiffeners, equations 4.14, 4.15, 4.16
- 4.4.7 Minimum Structural Plate Thickness
- 5.2.2 Wind Responses, equations 5.3, 5.4 and (1),(2) and (3), (b) equations 5.5, 5.6 and 5.7

ASCE Wind Exposure - Select a value for wind exposure, taken from the ASCE #7 Wind Design Code. It is permissible to use another wind code or pressure versus elevation. However, in order to determine the Wind Responses, the Exposure Category (A,B,C or D) based on ASCE must be entered:

Exposure	**Exposure Description**
A	Large City Centers
B	Urban and Suburban Areas
C	Open Terrain with scattered obstructions
D	Flat, unobstructed areas exposed to wind flowing over open water for a distance of at least 1 mile.

The default value is **C**.

Factor of Safety - Enter a value for the factor of safety. Factors of safety from ASME STS, Table 4.1 are shown below:

F.S.	**Load Combination**
1.50	Dead + Live + Other + Thermal + Along or Cross Wind
1.50	Dead + Live + Other + Thermal + Seismic
1.33	Dead + Live + Other + Abnormal Thermal + Along Wind/4
1.33	Construction

The default value is **1.5**.

Mean Hourly Wind Speed - Enter the mean hourly wind speed at the stack location.

Is the Stack Lined? - Select this option to indicate that the stack lined with at least a 2 inch thick 100 pcf liner.

Importance Factor - Enter the ASCE-7/STS importance factor from Table I-2.

Datum Line Options Dialog Box

Specifies options for datum lines.

Datum is taken from bottom/left tangent or bottom of vessel – Select this option and type any offset from that point if the datum is to be taken from the bottom of the vessel or the left tangent.

For Vertical Vessels – Specifies where the datum line is to be positioned if the datum is not taken from the bottom/left tangent or bottom of the vessel. The available options include:

- Top Shell Seam or Top of Vessel
- Bottom Shell Seam or Bottom of the Vessel
- Base of Skirt

Offset (+ or -) distance from seam – Modifies the position of the datum line vertically from the seam based on the value in the **For Vertical Vessels** box.

For Horizontal Vessels – Specifies where the datum line is to be positioned if the datum is not taken from the bottom/left tangent or bottom of the vessel. The available options include:

- Left Shell Seam or Left End of Vessel
- Right Shell Seam or Right End of Vessel

Offset (+ or -) distance from seam – Modifies the position of the datum line horizontally from the seam based on the value in the **For Horizontal Vessels** box.

Design Modification (Design Constraints Tab)

Select Wall Thickness for Internal Pressure - Select **Yes** to automatically set the wall thickness for internal pressure. If the required element thickness for internal pressure exceeds the value of **Finished Thickness** defined for **Element Data** on the **General Input** tab, the software increases **Finished Thickness** to meet or exceed the thickness required for internal pressure. The software exceeds the required thickness only if you select **Round Thickness to Nearest Nominal Size** on the **Configuration** (see "**Job Specific Setup Parameters Tab (Configuration Dialog)**" on page 200) dialog box (**Tools** tab > **Set Configuration Parameters**). The software automatically calculates this wall thickness as model data is entered, so select this option before any part of the vessel is modeled. If the original value for **Finished Thickness** is greater than the required thickness, then no change is made.

NOTE During data entry, the software does not check the required thickness for flanges. That check is performed during analysis.

Select Wall Thickness for External Pressure - Select **Yes** to automatically set the wall thickness for external pressure. The **Equipment Installation and Miscellaneous Options Dialog Box** (on page 339) opens. If the required element thickness for external pressure exceeds the value of **Finished Thickness** defined for **Element Data** on the **General Input** tab, the software calculates the required thickness of each element (or group of elements) and increases the value of **Finished Thickness**. The software notifies you when the thickness is changed.

NOTE **Select Stiffening Rings for External Pressure** is set to **No** when this value is **Yes**.

Select Stiffening Rings for External Pressure - Select **Yes** to automatically set the wall thickness for external pressure. The **Equipment Installation and Miscellaneous Options Dialog Box** (on page 339) opens. If the required element thickness for external pressure exceeds the value of **Finished Thickness** defined for **Element Data** on the **General Input** tab, the software calculates the required thickness of each element (or group of elements) and increases the value of **Finished Thickness**. The software notifies you when the thickness is changed.

NOTE **Select Stiffening Rings for External Pressure** is set to **No** when this value is **Yes**.

Select Wall Thickness for Axial Stress - Select **Yes** to automatically increase the thickness for axial stress in vertical vessels. The software calculates the axial stress and required thickness of each element (or group of elements) for longitudinal loadings (such as wind, earthquake, and weight of vertical vessels) and increases the value of **Finished Thickness**. The software exceeds the required thickness only if you select **Round Thickness to Nearest Nominal Size** on the **Configuration** (see "**Job Specific Setup Parameters Tab (Configuration Dialog)**" on page 200) dialog box.

SECTION 14

Load Cases Tab

Enter stress combination and nozzle pressure load cases. If no data is entered, the software uses the system defaults.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > Stress Combination Load Cases**.

   *The **Load Cases** tab appears at the bottom-left of the PV Elite window.*

Reset Cases - Click to reset **Stress Combination Load Cases** to the default set of cases provided with the software.

Global Scalar for WI Loads - Enter a value for the wind scalar multiplier. After wind loads are generated, they are multiplied by this value. A value that is greater than one increases the loads. A value less than one decreases the loads. Enter zero if there are no wind loads. The > button to the right sets the value back to the default value.

In many building codes, load cases for wind and seismic are discussed. When the code uses allowable stress design, wind and seismic loads are usually scaled by a factor. For example, in the IBC Code, earthquake loads are divided by a factor of 1.4. Other codes multiply by the earthquake load by 0.7.

Global Scalar for EQ Loads - Enter a value for the seismic scalar multiplier. After seismic loads are generated, they are multiplied by this value. A value that is greater than one increases the loads. A value less than one decreases the loads. Enter zero if there are no seismic loads. The > button to the right sets the value back to the default value.

In many building codes, load cases for wind and seismic are discussed. When the code uses allowable stress design, wind and seismic loads are usually scaled by a factor. For example, in the IBC Code, earthquake loads are divided by a factor of 1.4. Other codes multiply by the earthquake load by 0.7.

Use and Allow editing of Local Scalars in the Load Cases - Select to use load case scalars in place of global scalars. Also select this option if you selected **Determine the Basering design bolt load accounting for Load Case Factors?** in the **Basering Dialog Box** (on page 300) on the **General Input** tab. If this option is not selected, then values of 1.0 are used for scalar multipliers.

Set Load Cases to show WI and EQ Scalars - Click to show the scalar multipliers for **Global Scalar for WI Loads** and **Global Scalar for EQ Loads** in the load combination equations. These values change internally when the wind or seismic design code is changed. When version 2005 or earlier files are used, this value is automatically calculated.

In This Section

Stress Combination Load Cases (Load Cases Tab)

Enter the loads to combine for each load case. The software performs calculations for various combinations of internal pressure, external pressure, hydrotest pressure, wind load, and seismic load. You can define up to twenty combinations of these loadings for evaluation. Load cases are defined by a string of abbreviations showing the loads to be added. For example, "IP+OW+WI" is the sum of internal pressure plus operating weight plus wind. The software provides the following set of default load cases:

Load Case	**Load Combinations**
1	NP+EW+WI+FW+BW
2	NP+EW+EQ+FS+BS
3	NP+OW+WI+FW+BW
4	NP+OW+EQ+FS+BS
5	NP+HW+HI
6	NP+HW+HE
7	IP+OW+WI+FW+BW
8	IP+OW+EQ+FS+BS
9	EP+OW+WI+FW+BW
10	EP+OW+EQ+FS+BS
11	HP+HW+HI
12	HP+HW+HE
13	IP+WE+EW
14	IP+WF+CW
15	IP+VO+OW
16	IP+VE+EW
17	NP+VO+OW
18	FS+BS+IP+OW
19	FS+BS+EP+OW
20	

The following abbreviations are used for loads:

NP No Pressure

IP Internal Pressure

EP External Pressure

HP Hydrotest Pressure

EW Empty Weight

OW Operating Weight

HW Hydrotest Weight

WI Wind Load

EQ Earthquake Load

HE Hydrotest Earthquake

HI Hydrotest Wind

BW Bending Stress due to Lat. Forces for the Wind Case, Corroded

BS Bending Stress due to Lat. Forces for the Seismic Case, Corroded

BN Bending Stress due to Lat. Forces for the Wind Case, Uncorroded

BU Bending Stress due to Lat. Forces for the Seismic Case, Uncorroded

CW Axial Weight Stress, New and Cold (no corrosion allowance, CA)

WE Wind Bending Moment, New and Cold (Empty) (no CA)

WF Wind Bending Moment, New and Cold (Filled) (no CA)

FS Axial Stress due to Applied Axial Forces in Seismic Cases

FW Axial Stress due to applied forces in Wind Cases

Live loads (wind and earthquake) are calculated for the operating and hydrotest conditions. In both cases, the basic loads calculated are identical but the hydrotest live loads are usually a fraction of the operating live load. These hydrostatic fractions (percents) are entered in the live load definitions.

If **Consider Vortex Shedding** is selected on the **Design Constraints Tab (on page 325),** the following loads may also be used:

VO Bending Stress due to Vortex Shedding Loads (Ope)

VE Bending Stress due to Vortex Shedding Loads (Emp)

VF Bending Stress due to Vortex Shedding Loads (Test, no CA.)

PV Elite allows individual load case descriptors to have their own scale factors. These factors scale the stresses produced by the corresponding load case component. For example 1.25EQ would produce an earthquake stress 1.25 times higher than the design earthquake stress. An example of a complete load case would be:

```
IP+OW+0.7143EQ+FS+BS
```

This facility allows designers to comply with a variety of loading scenarios. Another application of this may be that fractions of wind and seismic loads can be added together in the same load case. ASME states that doing this is not required; however, some design institutions mandate this practice. Here is another example:

```
0.7EQ+0.25WI+OW
```

Notice that there is no need to put a star (*) in front of each descriptor. If this box is not checked then values of 1.0 will be used for scalar multipliers. However, if there is a global scalar specified for wind or seismic, that value will be used. Please note that this is for vertical vessels only. During the stress calculations, the maximum stress is saved at the location of the support (skirt base, lug,or leg). Knowing the section properties, the moment needed to create that stress can be computed and used in the skirt, lug or leg calculation as required.

Any load case component can have a specified scalar. It is not meaningful to have a value in front of the NP component. It is important to specify NP for any case that does not have pressure.

For vertical vessels, the maximum stress is saved at the location of the support (skirt base, lug, and leg). Using the section properties, the moment needed to create that stress can be calculated and used in the skirt, lug or leg calculation as required.

NOTE It is often stated that the required thickness of the skirt is needed. It is not valid to directly calculate this value based on bending stress and axial stress because the section modulus is needed and the element OD or ID is still unknown. While it is possible to make an assumption, this will not generate an accurate mathematical result. A small change in the thickness can change the allowable compressive stress (factor A and factor B) in a very non-linear fashion. For more information, see British Code PD 5500, Annex B, paragraph B.1.5.

Vary Compressive Allowable for Internal/External Cases - Select to use the external design temperature to calculate the stress factor "B" for load combination cases that involve external pressure and dead weight. The software uses the design internal temperature to calculate the allowable compressive stress from the External Pressure Chart. By default, the software uses the maximum of the internal and external design temperatures to calculate the allowable compressive stress for operating-type cases. This is also true for cases involving internal pressure.

For example, a load case of "IP + OW + WI" uses the design temperature for internal pressure to calculate the allowable compressive stress. The load case "EP + OW + WI" uses the external design temperature to calculate the allowable compressive stress.

CAUTION When using this option, a disruption in process may leave the column at design internal temperature and a vacuum. If the design external temperature was much lower, this could lead to non-conservative results.

Corrode Case Components WE, WF and CW - Select to use loads **WE** (Wind Bending Moment Empty, no CA), **WF** (Wind Bending Moment Filled, no CA), and **CW** (Axial Weight Stress, no CA) in the corroded condition.

Installation | Misc. Options - Click ... to open the **Equipment Installation and Miscellaneous Options Dialog Box** (on page 339) and specify options as to where items such as platforms, packing, and insulation are to be installed. These options are used to compute the center of gravity of the vessel in both the shop and field positions. The options are also used to compute weights such as operating weight and field test weight.

Fatigue Analysis - Click ... to open the **Fatigue Pressure/Cycle and UTS-Yield Data Dialog Box** and set options used to perform fatigue analysis on nozzles. In the dialog box, you enter values for **Number of Fatigue Cases to Process**, and then values for **Low Pressure**, **High Pressure**, and **# of Cycles** for each case. You must also select **Fatigue Calc?** in the **Nozzle Input Analysis** dialog box for each nozzle.

Equipment Installation and Miscellaneous Options Dialog Box

Define data for stiffening rings, saddles, legs, and lugs.

Platform Wind Area Calculation Method - Select the method for determining the surface area that a platform provides for wind load calculations. The selection is used to compute the wind area for all platforms specified in this project. Information on the force coefficient is also provided. Select one of the following:

- **Height * Width * Force Coefficient [Cf]**
- **1/2 Floor Plate Area * Force Coefficient [Cf]**
- **1/3 Height * Width * Force Coefficient [Cf]**
- **1/3 Projected Area * Force Coefficient [Cf]**

The software uses the area of the platforms to calculate forces applied to the vessel during wind loading analysis. Unfortunately, there is no standard method for computing the amount of area that a platform provides for wind load calculations.

NOTE If the selection is changed after one or more platforms are installed, click **List Dialog** to open the **Detail Listing Dialog Box** and then click **Platform Wind Area** to update the areas.

List Dialog - Click to open the **Detail Listing Dialog Box** and add details for platforms, nozzles, weights, packing, forces/moments, and pressure rings.

Adjust detail elevations by - Enter a value to shift the position of all details by the specified elevation distance. A negative value will move details down or left. A positive value moves the details up or right. This option is useful when all of the details such as rings, nozzles, and trays need to be adjusted by a specified amount. This may happen if an element is inserted into the model after it has been completed and the detail elevations need to be kept constant.

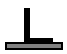
CAUTION If the adjustment moves a detail (such as a tray) into an element (such as a body flange), the software does not allow this, and the detail is lost and cannot be recovered.

But only for "From" nodes > or = - Enter the **From Node** number where you would like the change in position to start. All details on this element and the following elements are affected. A value of zero affects all elements.

External Pressure Ring Requirement Data

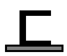
Structural Database - Select the structural database to use for external pressure ring stiffeners. Select **AISC**, **Korean/Japanese**, **German 1991**, **UK**, **Australian**, **South African**, or **India ISI**.

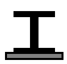
Stiffener Type to Meet Inertia Requirements - Select the stiffener type to use for external pressure rings on the vessel. For ASME VIII-1 and VIII-2, the software determines maximum stiffener spacing and adds rings to the vessel. Select one of the following:

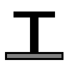

Equal Angle (flange in either direction)

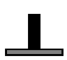

Unequal Angle (flange in either direction, "hard way" shown)


Double Large Angle
Double Small Angle
(with long or short sides back-to-back)

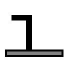

Channel


I - Beam

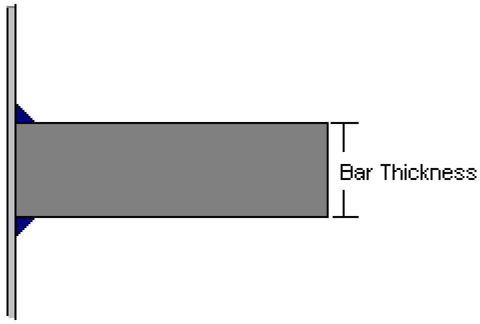

WT Section
MT Section
ST Section


Bar
The software designs a ring with an aspect ratio of 10 to 1.00. The height of the ring is 10 times its thickness. The minimum starting ring width is 0.5 inches (12mm).

For angle sections, are they rolled the "Hard Way"? - Select if the stiffener selected in **Stiffener Type to Meet Inertia Requirements** is rolled the "hard way" to have the strong axis of the ring perpendicular to the vessel wall, with the flange away from the vessel wall.



Bar thickness to use when designing new rings - Enter a thickness when **Bar** is selected for **Stiffener Type to Meet Inertia Requirements.** The software uses the thickness to calculate the bar depth for the pressure ring. The software creates a ring with a 10 to 1 aspect ratio. In other words, the width of the ring is ten times the thickness. If no value is entered, the software uses a default thickness of 0.375 inches. (9.5 mm).



Select From Standard Bar Ring List - When **Bar** is selected for **Stiffener Type to Meet Inertia Requirements**, select this option to calculate the minimum bar pressure ring size that meets moment of inertia requirements of UG-29(a) (or Appendix 1-5 or 1-8 for a cone-cylinder junction ring). The bar ring is selected from the list below:

Ring Thickness (in.)	**Ring Widths (in.)**				
1/4	1.5	1.75	2.00	2.50	---
3/16	2.5	---	---	---	---
3/8	2.0	2.5	3.0	3.5	4.0
1/2	3.5	4.0	4.5	5.0	5.5
5/8	5.0	6.0	---	---	---
3/4	5.0	5.5	6.0	8.0	---
7/8	6.0	8.0	---	---	---
1	6.0	8.0	10.0	11.0	---
1 1/4	8.0	10.0	12.0	---	---
1 1/2	8.0	10.0	12.0	---	---
2	12.0	18.0	---	---	---
4	30.0	---	---	---	---

Rigging Data

Impact Factor (<1 NoCalc) - Enter a rigging impact factor. Typical values range from 1.5 to 2.0, but a maximum value of 3.0 is possible. The software performs a rigging analysis (combined shear plus bending stress) to account for sudden loads when the vessel is lifted from the ground. The impact factor effectively increases the overall weight of the vessel, and accounts for a skirt support and **Lug Distances from Base**. If the value is **0**, no rigging analysis is performed.

Lug Distances from Base - Enter the lifting lug elevation for the vessel. Enter an elevation distance, one in each text box, to account for two lifting lugs. The order of the elevations in the text boxes does not matter. These distances are measured from the bottom of the vertical vessel or from the left end of the horizontal vessel. If both values are **0**, no rigging analysis is performed.

Saddle Calc Options

Calc per which method - Select a saddle calculation method: **ASME Div 2** or **PD-5500**.

Use New Metal Weight for Saddle Calcs - Select to use the uncorroded new metal weight to determine saddle loads for the operating condition. By default, the software uses the corroded metal weight.

Number of intermediate supports to be used during the hydrotest - Enter a value from **0** to **20**. When some larger vessels are hydrotested after construction, a number of intermediate supports may be placed under the vessel to keep the saddle stresses below their allowables.

Installation Location - The location of installation of various components, the platform, packing tray insulation or lining, can be specified here, whether in the shop or the field.

Combine Wind and Earthquake Loads - This option allows for wind and earthquake loads to be combined for saddles, legs, and lugs.

Nozzle Design Options (Load Cases Tab)

Define additional load case parameters for nozzle design.

Nozzle/Clip Design Pressure Options - Select one of the following design pressure methods for calculating the pressure at the nozzle:

- **1. MAWP + Static Head to Element Bottom -** Calculates the internal pressure on the nozzle on the bottom of the element where the nozzle is located. This pressure is the maximum allowable working pressure (MAWP) of the vessel plus the static liquid pressure head to the bottom of that element. Thus, the design pressure can vary for nozzles located on different elements. This option is appropriate if you are certain that your nozzle locations will not vary during the design process. If you use this option and a nozzle is lowered in the vessel and under additional pressure due to liquid head, you must rerun the analysis in order to determine if the nozzle geometry is satisfactory.
- **2. Design P + Static Head** - Calculates the exact internal pressure at the nozzle location. The pressure is the design internal pressure plus the additional static liquid pressure at the nozzle location. This option is appropriate for re-rating vessels or for the design of new vessels where there are no MAWP considerations. If the overall MAWP of the vessel is to be determined, it is strongly recommended that the model be rerun with the computed MAWP to be sure that all components pass at this higher pressure, which will be stamped on the nameplate. The pressure for all elements can be changed at **Design Internal Press** on the **Design Constraints Tab** (on page 325).

- **3. Overall MAWP + Static Head (governing element)** - Calculates one design internal pressure for all of the nozzles located on the vessel, based on the static liquid pressure to the bottom of the element that is governing the MAWP. If the nozzle location on a vessel changes due to a client request, there is no need to rerun nozzle calculations because the pressure used in the calculations does not change. This method is ideal for designing new vessels and is the most conservative option.
- **4. MAWP + Static Head to Nozzle** - Calculates the MAWP of the vessel and then adds the static liquid pressure from the liquid surface to the nozzle location. For nozzles at different elevations, the design pressure will vary.

NOTE If the resulting nozzle reinforcement MAWP does not need to govern the MAWP of the vessel, options 1, 3 or 4 should be used. This is a common requirement for vessels that are used in the chemical and petro-chemical industries.

Consider MAPnc - Select to require that nozzle reinforcement calculations are performed for the MAP new and cold condition. The software checks to see if the nozzle is reinforced adequately using the MAPnc generated during internal pressure calculations. When the area-of-replacement calculations are made for this case, cold allowable stresses are used and the corrosion allowance is set to 0. Designing nozzles for this case helps the vessel to comply with UG99 or appropriate (hydrotest) requirements. Check your design requirements to see if this case is required by your client.

Consider External Loads for Nozzle Tr - Select to calculate the nozzle area of replacement requirements using the required thickness of the shell. This value, *tr*, is critical in the ASME code. The software determines the maximum thickness based on the highest stress ratio and uses that value if it governs over the required thickness based on internal or external pressure. There are cases where pressure requirements do not govern the value of tr. This can occur when a nozzle is located near the bottom of a tall vertical vessel. If there is a high wind load or seismic load on the structure, bending stress can govern the required thickness of the shell section. If this is the case, then the value of tr (per UG-22 Div. 1) should be based on the controlling factor.

NOTE Optionally, if tr needs to be specified for a specific nozzle, the value can be entered directly to **User Tr** on the **Nozzle Input/Analysis Dialog Box** of **Nozzle Input (see "Nozzle" on page 58)**.

Use Appendix 1-9 (Div. 1) - Select to use ASME Code Case 2168. On February 14, 1994 ASME approved case 2168, providing an alternative method for reinforcing radial nozzles in cylindrical shells. The nozzle must be connected to the cylindrical shell by a full penetration groove weld.

Design Pads to Reinforce Openings - Select when pad-defined geometries are inadequately reinforced. The software calculates the diameter and thickness of the pad required to reinforce the opening. If the software changes the pad data during analysis, it will prompt you to reload the file so that you can view the new changes.

NOTE This option is restricted to ASME Section VIII analysis.

Nozzle Sort Options - Select a sort order of nozzles for the Nozzle Schedule report. Select **By Name Ascending**, **By Name Descending**, **By Diameter, Ascending**, **By Diameter, Descending**, or **No Sorting**.

ASME Large Nozzle Calc Options - Select the load cases to use for evaluation of large openings. Select **Use 1-7** or **Use 1-10**.

SECTION 15

Wind Loads (Wind Data Tab)

Click **Input** > **Wind Loads** to enter wind code data. The **Wind Data** tab appears on the left. Select a wind design code and enter data required by that code. If no data is entered, the software uses the system defaults.

Wind Design Code - Select the design code to use for wind calculations:

Code	**Description**
As/Nz 1170:2002 **An/Nz 1170.2:2011**	Design Wind Code of Australia and New Zealand, 2002 and 2011 editions For more information, see *As/Nz 1170:2002 and As/Nz 1170.2:2012 Wind Data* (see "*As/Nz 1170:2002 & As/Nz 1170.2:2011 Wind Data*" on page 347).
ASCE-93 **ASCE-95** **ASCE-98/02/05/IBC-03** **ASCE-2010**	American Society of Civil Engineers Standard 7 (formerly ANSI A58.1). For more information, see: *ASCE-93 Wind Data* (on page 353) *ASCE-95 and ASCE-98/02/05/IBC-03 Wind Data* (on page 354) *ASCE-2010 Wind Data* (on page 356)
BS6399-97	Britain's Wind Design Code, 1997 Edition (Replaces CP-3) For more information, see *BS6399-97 Wind Data* (on page 361).
Brazil NBR 6123	Design Wind Code for Brazil For more information, see *Brazil NBR 6123 Wind Data* (on page 357).
China GB 50009	Design Wind Code for China For more information, see *China GB 50009 Wind Data* (on page 363).
EN-2005 **EN-2010**	Design Wind Codes for several European Countries with a 2005 and 2010 editions For more information, see *EN-2005/EN-2010 Wind Data* (see "*EN-2005 and EN-2010 Wind Data*" on page 366).
Euro Code	Design Wind Code for several European countries, including France For more information, see *Euro Code Wind Data* (on page 367).
IBC 2006 **IBC 2009** **IBC 2012**	International Building Codes For more information, see *IBC 2006, IBC 2009, and IBC 2012 Wind Data* (on page 370)

IS-875	Wind Design Code of India, 1987 edition Amd. 1 and 2 (2003) For more information, see *IS-875 Wind Data* (on page 371).
JPI-7R-35-2004	Japanese Wind Code For more information, see *JPI-7R-35-2004 Wind Data* (on page 373).
Mexico	Official Design Wind Code of Mexico, 1993 and 2008 For more information, see *Mexico 1993 Wind Data* (on page 374).
NBC-95 **NBC-2005** **NBC-2010**	National Building Code of Canada For more information, see: *NBC-95 and NBC-2005 Wind Data* (on page 384) *NBC-2010 Wind Data* (on page 386)
SANS 10160-3:2010	South African National Standard 10160, Section 3, 2010 edition. For more information, see *SANS 10160-3:2010 Wind Data* (on page 388).
UBC	Uniform Building Code For more information, see *UBC Wind Data* (on page 389).
User Defined	Enter your own wind pressure versus elevation in the Wind Profile Table.
No Wind Loads	

Wind for Hydrotest - Enter the percentage of the wind load (not wind speed) that is applied during the hydrotest. This is typically as low as 33 percent of the design wind load, because it can be assumed that the vessel will not be hydrotested during a hurricane or severe storm.

Beta: Operating/Empty/Full - **Beta: Operating** - In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, pages 246, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty - In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full - In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

In This Section

As/Nz 1170:2002 & As/Nz 1170.2:2011 Wind Data

The principles of the Australian/New Zealand code are basically the same as other wind codes. A moving air mass causes forces to be exerted on anything that obstructs its path. The starting point of any wind action analysis starts with the Bernoulli equation that converts the kinetic energy of the wind into a static pressure. This pressure simply attempts to move anything in its path. In the case of a vertical tower, the wind deflects the tower in the direction of the flowing wind. Because of the friction, the wind (which is a fluid) passes around the vessel, creating small eddie currents called vortices on either side of the vessel. The spinning air loses some of it pressure as its angular kinetic energy increases. This is again an effect predicted by the Bernoulli equation. This spinning eddie, or vortex, causes a lowering of the pressure on one side of the vessel. As these vortices occur alternatively on the sides of the vessel, the tower tends to sway from side to side perpendicular to the direction of the wind, vibrating like an upside-down pendulum.

The wind pressure causes a static deflection in the direction of the wind stream, and the vortices cause the vessel to vibrate at right angles to the wind stream (vector) as mentioned above.

Static deflection and dynamic vibration cause the vessel to experience longitudinal, or axial bending stresses. Both the stresses and deflection must be limited to provide for good design.

As stated above, the wind kinetic energy is converted to pressure, but there are other factors influencing the final wind pressure. Among these factors are (but not limited to) the following:

- The height, or altitude, of the tower or vessel in the wind stream.
- The elevation of the site on which the vessel is installed.
- The base elevation of the vessel if is supported on an elevated platform or structure.
- The effect of the surround geography, such as hills and escarpments.
- The effect of upstream buildings.

To perform a reasonable analysis, you must enter sufficiently accurate information for the software to apply the correct factors and special calculations for specific cases.

Design Wind Speed - Enter the design wind speed V_r. This is the regional wind speed, as described in section 3.2, table 3.1 of the code:

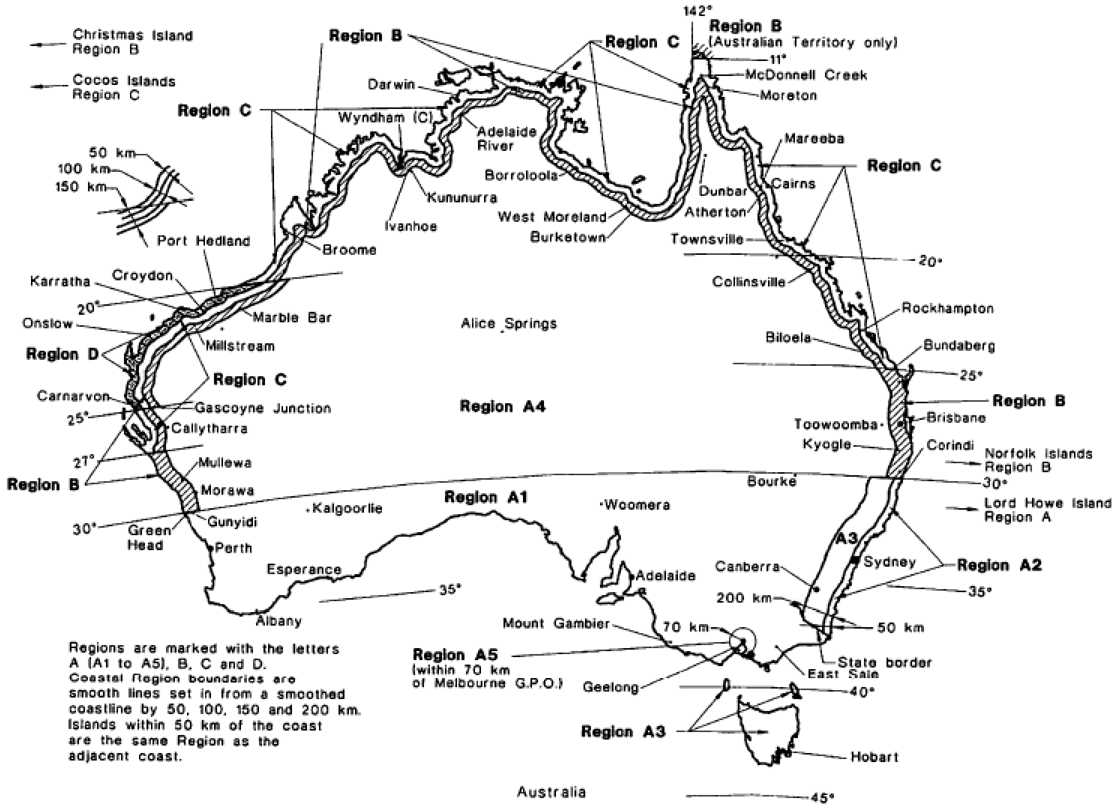
Regional wind speed (m/s)	Region				
	Non-cyclonic			Cyclonic	
	A (1 to 7)	W	B	C	D
V_5	32	39	28	F_C33	F_D35
V_{10}	34	41	33	F_C39	F_D43
V_{20}	37	43	38	F_C45	F_D51
V_{25}	37	43	39	F_C47	F_D53
V_{50}	39	45	44	F_C2	F_D60
V_{100}	41	47	48	F_C56	F_D66
V_{200}	43	49	52	F_C61	F_D72
V_{500}	45	51	57	F_C66	F_D80
V_{1000}	46	53	60	F_C70	F_D58
V_{2000}	48	54	63	F_C73	F_D90
V_R (see NOTE)	$67-41R^{-0.1}$	$104-70R^{-0.045}$	$106-92R^{-0.1}$	F_C x ($122-104R^{-0.1}$)	F_D x ($156-142R^{-0.1}$)

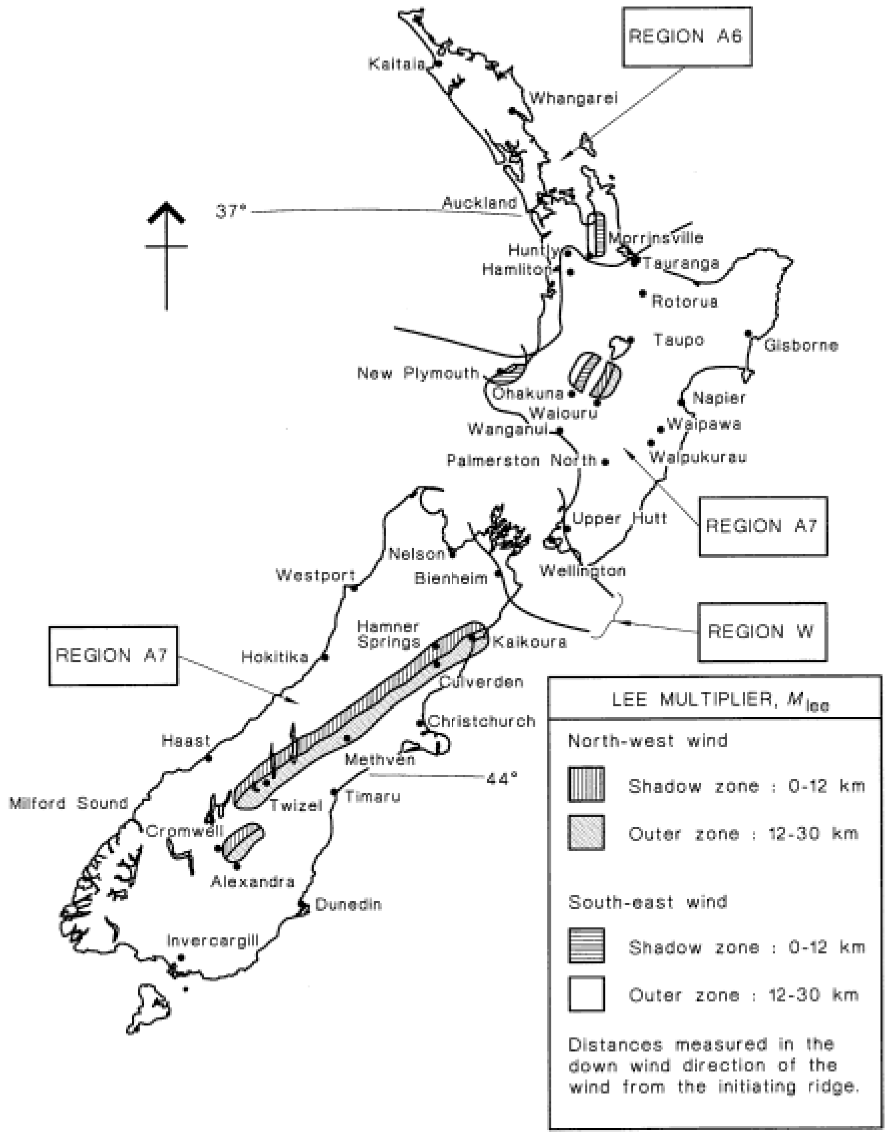
NOTE Round the calculated value to the nearest 1 m/s.

If you are entering the wind speed in other units, remember to convert the above wind speeds from m/s to your specific units:

Convert m/s to	Multiply by
kph	3.600
mph	2.237
ft/s	3.281

Wind Region - Select the wind region. The wind region is determined from the geographic locations for Australia and New Zealand. The maps of these locations are found in Figure 3.1 of the code:





New Zealand

Terrain Category - Select the terrain category, as defined in section 4.2.1 of the code:

Category 1 - Exposed open terrain, with few or no obstructions and water surfaces at serviceable wind speeds.

Category 2 - Water surfaces, open terrain, grassland with few, well-scattered obstructions having heights generally from 1.5 m to 10 m.

Category 3 - Terrain with numerous closely spaced obstructions 3 m to 5 m high, such as areas of suburban housing.

Category 4 - Terrain with numerous large, high (10 m to 30 m high) and closely spaced obstructions, such as large city centres and well-developed industrial complexes.

NOTE Select the terrain category with due regard to the permanence of the obstructions that constitute the surface roughness. In particular, vegetation in tropical cyclonic regions shall not be relied upon to maintain surface roughness during wind events.

Lee Effect Multiplier (Mlee) - Enter the lee effect multiplier. The default value is **1.0**.

Paragraph 4.4.3 discusses the issue of the lee effect multiplier. In the case of New Zealand, reference is made to the New Zealand site map shown in **Wind Region**. For all other sites, it shall be taken as 1.0. No lee zones have been defined in Australia.

The full paragraph from the code reads as follows:

4.4.3: The lee (effect) multiplier (Mlee) shall be evaluated from New Zealand sites in the lee zones as shown in Figure 3.1(b). For all other sites, the lee multiplier shall be 1.0. Within the lee zones, the lee multiplier shall apply only to wind from the cardinal directions nominated in Figure 3.1(b). Each lee zone shall be 30 km in width, measured from the leeward crest of the initiating range, downwind in the direction of the wind nominated. The lee zone comprises a 'shadow lee zone', which extends 12 km from the upwind boundary of the lee zone (crest of the initiating range), and an 'outer lee zone' over the remaining 18 km. The lee multiplier shall be 1.35 for sites within the shadow lee zone (i.e., within 12 km of the crest of the range). Within the outer lee zone, the lee multiplier shall be determined by linear interpolation with the horizontal distance, from the shadow/outer zone boundary (where Mlee - 1.35), to the downwind lee zone boundary (where Mlee = 1.0).

Hill Shape Factor (Mh) - Enter the hill shape factor M_h, taken from Table 4.4:

Upwind Slope (H/2Lu)	$\mathbf{M_h}$
< 0.05	1.0
0.05	1.08
0.10	1.16
0.20	1.32
0.30	1.48
>= 0.45	1.71

Paragraph 4.4.2 gives precise details for the derivation of the hill shape factor.

Wind Direction Multiplier - Enter the wind direction multiplier.   The default value is 1.0.

The wind direction multiplier is detailed in paragraph 3.4 of the code, specifically Table 3.2. As the wind multiplier is determined from the cardinal wind directions (N, NE, E, SE, S SW, W and NW), the value for any direction is specified in the table as **1.0**. This value is recommended for all cases.

Convert to Permissible Stress Gust Wind Speed - Select to convert the given wind speed to a permissible stress basis. this lowers the wind loads on the vessel. AS/NZS 1170.2 Supp 1:2002, Section C3 discusses the division of the wind speed given in the standard by the square root of 1.5.

Surface Roughness Height hr - Enter the surface roughness value h_r in mm. This value is used to calculate the ratio h_r/d which is then used to calculate the drag force coefficient (C_d) for rounded cylindrical shapes, according to Table E3. For pressure vessels, typical values range from **0.003** mm for painted metal surfaces to **15** mm for heavily rusted surfaces. Light rust has a value of **2.5** mm, while galvanized steel has a value of **0.15** mm.

Site Elevation (E) - Enter the height of the site above mean sea level *E*.

Base Elevation from Site (Eb) - Enter the height of the base of the vessel above the site level E_b. This is relevant in cases where the vessel is supported on a structure or an elevated foundation base.

Avg Spacing of Shielding Bldgs - Enter the average spacing of the buildings that shield the vessel. This is discussed in paragraph 4.3.3 of the code:

The shielding parameter (s) in Table 4.3 shall be determined as follows:

$$s := \frac{l_s}{\sqrt{h_s \cdot b_s}}$$

where

l_s = average spacing of shielding buildings, given by:

$$= \quad h \cdot \left(\frac{10}{n_s} + 5 \right)$$

h_s = average roof height of shielding buildings

b_s = average breadth of shielding buildings, normal to the wind stream

h = average roof height, above ground, of the structure being shielded

n_s = number of upwind shielding buildings within a 45 deg. sector of radius 20h and with $h_s \geq z$

Avg Breadth of Shielding Bldgs - Enter the average breadth of the buildings that shield the vessel. For more information, see **Avg Spacing of Shielding Bldgs.**

Avg Height of Shielded Bldgs (hs) - Enter the average height of the buildings that shield the vessel. For more information, see **Avg Spacing of Shielding Bldgs.**

# of Upwind Bldgs @ 45 degrees - Enter the number of upwind buildings within a 45° arc. The upwind buildings are the ones shielding the vessel. For more information, see **Avg Spacing of Shielding Bldgs.**

ASCE-93 Wind Data

Design Wind Speed - Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant - Select an ASCE-7 exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor - Enter a value for the importance factor. ASCE-7 generally requires a value between 0.95 and 1.11. The software uses this value directly without modification. Values are taken from table 5 of the ASCE standard:

	Importance Factor		
Category	**100 mi from Hurricane Oceanline**	**At Oceanline**	**Classification**
I	**1.00**	**1.05**	Buildings and structures not listed below. NOTE Most petrochemical structures use this category.
II	**1.07**	**1.11**	Buildings and structures where more than 300 people congregate in one area.
III	**1.07**	**1.11**	Buildings designed as essential facilities, hospitals, and so on.
IV	**0.95**	**1.00**	Buildings and structures that represent a low hazard in the event of a failure.

Roughness Factor - Enter the roughness factor (from ASCE-7, Table 12):

- **1** - Round, moderately smooth

- **2** - Round, rough (D'/D = 0.02)
- **3** - Round, very rough (D'/D = 0.08)

NOTES

- Most petrochemical sites use a value of 1 (moderately smooth). Some designers use a value of 3 (very rough) to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, click **Tools** tab > **Set Configuration Parameters** to enter it in directly on the *Job Specific Setup Parameters* (see "*Job Specific Setup Parameters Tab (Configuration Dialog)*" on page 200) tab.

ASCE-95 and ASCE-98/02/05/IBC-03 Wind Data

Basic Wind Speed [V] - Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Terrain Surface Roughness - Select an ASCE-7 exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor [I] - Enter a value for the importance factor. ASCE-7 generally requires a value between 0.77 and 1.15. The software uses this value directly without modification. Values are taken from table 6-2 of the ASCE 95 standard or table 6-1 from the ASCE 98 standard:

Category	Factor	Classification
I	**0.87**	Buildings and other structures that represent a low hazard to human life in the event of failure. NOTE Most petrochemical structures use this category.
II	**1.00**	Buildings and structures except those listed in categories I, III and IV.

III	**1.15**	Buildings and structures that represent a substantial hazard in the event of a failure.
IV	**1.15**	Buildings designed as essential facilities, such as hospitals.

NOTE In the 98 standard, the importance factor can be **0.77** for category I when wind speeds are greater than 100 miles per hour.

Type of Surface - Enter the roughness factor (from ASCE-7, Table 12):

- **Smooth**
- **Rough** (D'/D = 0.02)
- **Very Rough** (D'/D = 0.08)

NOTES

- Most petrochemical sites use smooth. Some designers use very rough to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab, click **Set Configuration Parameters** to enter it in directly on the **Job Specific Setup Parameters** tab.

Hill Height [H] - Enter the height of a hill or escarpment relative to the upwind terrain. See ASCE 7-95/98, Fig. 6-2 for details.

Distance to Site [x] - Enter the distance (upwind or downwind) from the crest of the hill to the building site. See ASCE 7-95/98 Fig. 6-2 for details.

Crest Distance [Lh] - Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. See ASCE 7-95/98, Fig. 6-2 for more details.

Type of Hill - Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. See ASCE 7-95/98, Fig. 6-2 for details.

ASCE-2010 Wind Data

Basic Wind Speed [V] - Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Terrain Surface Roughness - Select an exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor [I] - This value is no longer used.

Type of Surface - Enter the roughness factor (from ASCE 7-10, Figure 29.5-1):

- **Smooth** (Moderately smooth)
- **Rough** (D'/D = 0.02)
- **Very Rough** (D'/D = 0.08)

NOTES

- Most petrochemical sites use **Smooth**.
- You can use very rough to account for items such as platforms, piping, and ladders, instead of entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab and click **Set Configuration Parameters** to enter it directly on the **Job Specific Setup Parameters** tab.

Hill Height [H] - Enter the height of a hill or escarpment relative to the upwind terrain. For more information, see ASCE 7-10, Figure 29.8-1.

Distance to Site [x] - Enter the distance (upwind or downwind) from the crest of the hill to the building site. For more information, see ASCE 7-10, Figure 29.8-1.

Crest Distance [Lh] - Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. For more information, see ASCE 7-10, Figure 29.8-1.

Type of Hill - Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. For more information, see ASCE 7-10, Figure 29.8-1.

Beta: Operating - In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, pages 246, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty - In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full - In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Brazil NBR 6123 Wind Data

NBR 6123 provides two types of wind analysis: static and dynamic.

Static wind analysis is performed when the fundamental period of vibration is less than one second. The goal of the analysis is to determine the wind pressure at each elevation of interest (z). The basic equation is:

$$q(z) = V_k^2/1.63$$

Where:

- $V_k = V_o*S_1*S_2*S_3$
- V_o, S_1 and S_3 are constants
- S_2 changes as a function of the height
- Wind velocities are specified in m/s and pressures are specified in N/mm^2

After the pressure is calculated, the force on the element is the pressure times the element area times the shape factor:

Surface Condition	Shape Factor for Height / Diameter of:						
	0.5	1.0	2.0	5.0	10	20	∞
Rough	0.7	0.7	0.7	0.8	0.9	1.0	1.2
Smooth	0.5					0.6	0.6

Dynamic wind analysis is performed when the fundamental period of vibration is greater than 1 second. You must first calculate the project wind velocity V_p:

$$V_p = 0.69 * V_o * S_1 * S_3$$

Next, calculate d/H and V_p * T_1/1800. Using these two terms and knowing the height of the vessel, the value of Xi can be determined from figures 14-18 in NBR 6123.

b and p, functions of **Roughness Category**, can now be determined:

Value	**Ground Category**				
	I	**II**	**III**	**IV**	**V**
p	0.095	0.150	0.185	0.230	0.310
b	1.23	1.00	0.86	0.71	0.50

Finally, calculate the wind pressure:

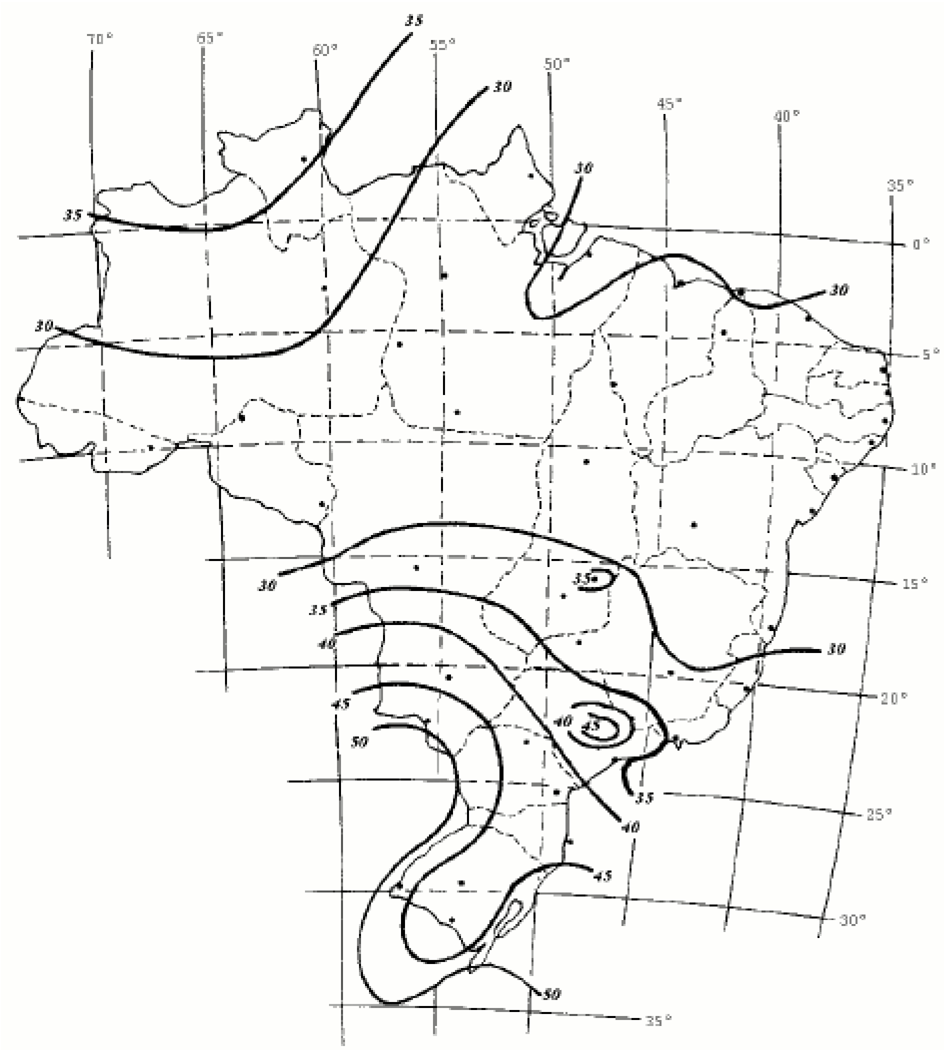
$$q(z) = 0.613 * b_2 * V_p^2\{(z/10)^{2p} + [(H/10)^p * (z/H)^{1.7} * (4.4/(2.7 + p) * Xi]\}$$

After the pressure at the needed elevation is known, the force is determined as stated above.

NOTE If you have a shape factor value that you want to enter yourself, go to the **Tools** tab, click **Set Configuration Parameters**, and then enter the value on **Job Specific Setup Parameters** tab in the **Wind Shape Factor** box.

Basic Wind Velocity (Vo) - Enter the wind velocity V_o from a three second gust, exceeded only once in 50 years. The velocity is measured at 10 meters over smooth open ground and depends on the plant location. As a general rule, the wind can blow in any horizontal direction. This velocity is taken from Figure 1 and item 8 showing the iso-velocities over Brazil.

The following are found in Petrobras document BPE-500-P4-19i and the Brazilian Wind Code NBR 6123:



Refinery Name	Wind Velocity (in m/s)
LUBNOR	30.0
RECAP	40.0
REDUC	35.0
REFAP	45.0
REGAP	30.0
REPAR	40.0

REPLAN	45.0
REMAN	30.0
REVAP	40.0
RPBC	50.0
RLAM	30.0

Topographic Factor (S1) - Enter the topographic factor S_1. This factor accounts for the variations and profile of the land. For plain, or slightly uneven ground, use a value of **1**. The larger this value is, the greater the final calculated wind pressure will be. If the vessel is on a hill top, this value should be calculated according to section 5.2 of NBR 6123.

Roughness Category (S2) - Select the roughness category S_2:

Category	Description
I	Applies to plain ground with large dimensions (more than 5 km of extension)
II	Applies to plain (or slightly uneven) ground with few, and separated, obstacles
III	Applies to plain or uneven ground obstructed by obstacles (walls or separated low buildings)
IV	Applies to ground with many grouped obstacles in industrial or urban areas
V	Applies to ground with many grouped and tall obstacles (such as developed industrial areas)

NOTE The *lower* the category, the *higher* the wind load. For example, Category I produces a higher wind load than Category II.

Dimension Class - Select the Dimension class. This parameter accounts for the greatest horizontal or vertical dimension of the vessel.

Class	Description
A	The greatest dimension is less than or equal to 20 meters
B	The greatest dimension is greater than 20m and less than 50 meters
C	The greatest dimension is greater than or equal to 50 meters

Statistical Factor (S3) - Enter the statistical factor S_3.This factor accounts for security and the expected life of the equipment. For industrial plants, use **1.0**.

Base Elevation - Enter the distance the base of the equipment is from grade.

Vessel Surface Condition - Select the vessel surface condition, **Smooth** or **Rough**. **Rough** results in an increased value of the shape factor and generates a higher wind load on the vessel because there is more drag. The shape factor is based on the height to diameter ratio of the vessel:

Surface Condition	**Shape Factor for Height / Diameter of:**						
	0.5	1.0	2.0	5.0	10	20	∞
Rough	0.7	0.7	0.7	0.8	0.9	1.0	1.2
Smooth	0.5					0.6	0.6

BS6399-97 Wind Data

BS 6399-97 - The British Wind Code - Loadings for buildings - Part 2: Code of practice for wind loads. The year of issuance of this code is 1997, replacing CP3.

Design Wind Speed - Enter the design value of the wind speed V_b. This varies according to geographical location and according to company or vendor standards. Typical wind speeds are shown in Figure 6 of BS 6399 and vary from 20 to 31 meters per second (44.7 to 69.3 miles per hour). The wind speeds are only relevant to the United Kingdom.

NOTE Enter the lowest value reasonably allowed by the standards you are following, because the wind design pressure, and thus force, increases as the square of the speed.

Site Elevation (Delta s) - Enter the site altitude above mean sea level (paragraph 2.2.2.2 of the code). This value plus **Base Elevation** is used to calculate the height of each point in the vessel above mean sea level. For example, if the vessel is installed on a site that is 100 m (328 ft) above sea level, it is exposed to a higher wind pressure *P* than if installed on the beach (at mean sea level).

Upwind Building Height Ho - Enter the average height H_o of buildings upwind of the vessel, when **Town** is selected for **Vessel Location**. The buildings tend to shield the vessel from the wind. Conservatively, this value can be zero, so that the vessel takes the full force of the wind. H_o is used to modify the effective vessel wind height H_e for any vessel element. See paragraph 1.7.3.3 of BS6399.

NOTE The corrected final wind pressure V_e acting on any element of the vessel is determined by **Vessel Location**, **Distance to Coastline**, and the effective height H_e (modified by **Upwind Building Height Ho)**. Factors in table 4 of BS6399 modify the wind velocity. This table derives S_b, which is calculated internally by the software.

Upwind Building Spacing X - Enter the average spacing *X* of buildings upwind of the vessel, when **Town** is selected for **Vessel Location**. The buildings tend to shield the vessel from the wind. If the buildings are closer together, they provide greater protection from the wind. See paragraph 1.7.3.3 of BS6399.

Base Elevation - Enter the elevation at the base of the vessel. This value plus the value of **Site Elevation (delta s)** is used to calculate the height of each point in the vessel above mean sea level.

Vessel Location - Select the type of location where the vessel is installed, **Town**, or **Country**.

NOTE The corrected final wind pressure V_e acting on any element of the vessel is determined by **Vessel Location**, **Distance to Coastline**, and the effective height H_e (modified by **Upwind Building Height Ho)**. Factors in table 4 of BS6399 modify the wind velocity. This table derives S_b, which is calculated internally by the software.

Distance to Coast Line - Enter the distance the vessel is located from the coast.

NOTE The corrected final wind pressure V_e acting on any element of the vessel is determined by **Vessel Location**, **Distance to Coastline**, and the effective height H_e (modified by **Upwind Building Height Ho)**. Factors in table 4 of BS6399 modify the wind velocity. This table derives S_b, which is calculated internally by the software.

Size Effect Factor Ca - Enter the size effect factor C_a. This factor generally ranges from 0.53 to a maximum value of 1.0. The size effect factor is a function of the diagonal dimension *a*, the effective height H_e,**Vessel Location**, and **Distance to Coastline**. This value is taken from figure 4 of BS-6399-2.

Factor Kb from Table 1 - Select the building-type factor K_b , taken from Table 1 of BS6399. The default value is 2. Select 8, 4, 2, 1, or 0.5 based on the following vessel height limitations:

K_b	Building Type	Maximum Total Height of Vessel
8	Welded steel unclad frames.	23 m (75.4 ft)
4	Bolted steel and reinforced concrete unclad frames.	75m (246 ft)
2	Portal sheds and similar light structures with few internal walls.	240m (787 ft)
1	Framed buildings with structural walls around lifts and stairs only (such as, office buildings of open plan or with partitioning).	300m (984 ft)
0.5	Framed buildings with structural walls around lifts and stairs with additional masonry subdivision walls (such as apartment buildings), building of masonry construction and timber-framed housing.	300m (984 ft)

NOTE Towers over 75 meters in height are not likely, and require other design considerations in addition to wind loading.

Annual Probability Factor Q - Enter the probability factor *Q*, used to calculate the final probability factor S_p associated with the likelihood of high velocity gusts occurring over specified periods. The default value is 0.02, set by the code as a standard value for a mean recurrence of 50 years. See Annex D of BS6399 for more information.

Q	Explanation
0.632	NOTE 1: The annual mode, corresponding to the most likely annual maximum value. (S_p = 0.749)

0.227	NOTE 2: For the serviceability limit, assuming the partial factor for loads for the ultimate limit is ?f = 1.4 and for the serviceability limit is ?f = 1.0, giving S_p = Sqrt(1 / 1.4) = 0.845. (S_p = 0.845)
0.02	NOTE 3: The standard design value, corresponding to a mean recurrence interval of 50 years. (S_p = 1.000)
0.0083	NOTE 4: The design risk for bridges, corresponding to a mean recurrence interval of 50 years. (S_p = 1.048)
0.00574	NOTE 5: The annual risk corresponding to the standard partial factor for loads, corresponding to a mean recurrence interval 1,754 years. Back-calculated assuming the partial factor load for the ultimate limit is ?f = 1.4 and all risk is ascribed to the recurrence of wind. (S_p = Sqrt(1.4))
0.001	NOTE 6: The design risk for nuclear installations, corresponding to a mean recurrence interval of 10,000 (ten thousand) years. (S_p = 1.263)

Seasonal Factor Ss - Enter a factor S_s for exposure to the weather. The default value is 1.0. BS6399 in paragraph 2.2.2.4 states: "For permanent buildings and buildings exposed for continuous periods of more than six months a value of 1.0 should be used for S_s." A value of less than **1.0** should only be used after solid research.

Directional Factor Sd - Enter a factor S_d for directionality of the tower. The default value is **1.0** because a tower is typically symmetrical about its central axis. A value of less than **1.0** should only be used under exceptional circumstances. For other values, see Table 3. The values in that table range between 0.73 and 1.00.

Vessel Surface Type - Select the surface type of the vessel, **Rough** or **Smooth**.

China GB 50009 Wind Data

The Chinese Wind Code analysis in the software is taken from Chinese specification GB 50009 - 2001, 2002. The wind loading calculation guidelines begin on page 24 of the code.

The basic formulation for determining the wind pressure at an arbitrary elevation is based on equation 7.1.1-1. This equation is for Main Wind Force Resisting Systems. This is the printed equation: *wk* = *ßzμsμzwo*. From the tables in the code, values of *μs*, *μz* and the other values are determined.

This formula includes the shape coefficient. The generated wind pressure is not dependent on the type of structure. However, when the final force is calculated, it is necessary to include the shape factor, such as for a cylinder taken from page 39 for a tower or chimney.

Ref. Wind Pressure (wo) - Enter the reference wind pressure W_o , from table D.4 of the Chinese Wind Design Code. The value should be no less than 0.3 kN/m^2.

Terrain Roughness Category - Enter the terrain roughness category:

Value	**Description**
A	Flat, unobstructed open terrain (most conservative)
B	Village, hill and less populated and less congested sites
C	Populated sites with low buildings and shorter structures
D	Densely populated areas with many tall structures that provide shielding (least conservative)

The category is used to find elevation μ_z, from Table 7.2.1:

Height (m)	**Terrain Roughness Category**			
	A	**B**	**C**	**D**
5	1.17	1.00	0.74	0.62
10	1.38	1.00	0.74	0.62
15	1.52	1.14	0.74	0.62
20	1.63	1.25	0.84	0.62
30	1.80	1.42	1.00	0.62
40	1.92	1.56	1.13	0.73
50	2.03	1.67	1.25	0.84
60	2.12	1.77	1.35	0.93
70	2.20	1.86	1.45	1.02
80	2.27	1.95	1.54	1.11
90	2.34	2.02	1.62	1.19
100	2.40	2.09	1.70	1.27
150	2.64	2.38	2.03	1.61
200	2.83	2.61	2.30	1.92
250	2.99	2.80	2.54	2.19
300	3.12	2.97	2.75	2.45

350	3.12	3.12	2.94	2.68
400	3.12	3.12	3.12	2.91
450	3.12	3.12	3.12	3.12

Vessel Surface Condition - Select the vessel surface condition, taken from page 39 of the Chinese Wind Code. Select **Smooth**, **Rough**, or **Very Rough**. You can override this option in **Tools** > **Configuration**.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Is the Vessel Building Supported? - Select to use the factors for β_{gz} from Table 7.5.1:

Height (m)	**Terrain Roughness**			
	A	**B**	**C**	**D**
5	1.69	1.88	2.30	3.21
10	1.63	1.78	2.10	2.76
15	1.60	1.72	1.99	2.54
20	1.58	1.69	1.92	2.39
30	1.54	1.64	1.83	2.21
40	1.52	1.60	1.77	2.09
50	1.51	1.58	1.73	2.01
60	1.49	1.56	1.69	1.94
70	1.48	1.54	1.66	1.89
80	1.47	1.53	1.64	1.85
90	1.47	1.52	1.62	1.81
100	1.46	1.51	1.60	1.78
150	1.43	1.47	1.54	1.67
200	1.42	1.44	1.50	1.60
250	1.40	1.42	1.46	1.55
300	1.39	1.41	1.44	1.51

The value of β_{gz} decreases with elevation so that the wind pressure decreases as the elevation increases. The software uses equation 7.1.1-2 in the code to determine the wind pressure at the needed elevation.

EN-2005 and EN-2010 Wind Data

Ref. Wind Velocity Vb,0 - Enter the basic wind velocity $V_{b,0}$ of the area where the equipment is situated. $V_{b,0}$ is used along with **C Dir** and **C Season** to compute V_b.

Terrain Category - Select the terrain category:

Terrain Category	Description
0	Sea or coastal area exposed to the open sea.
I	Lakes or flat and horizontal areas with negligible vegetation and without obstacles.
II	Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights.
III	Area with regular cover of vegetation or buildings, or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, permanent forest).
IV	Area in which at least 15% of the surface is covered with buildings and their average height exceeds 15 m.

NOTE The *lower* the category, the *higher* the wind load. Category 0 produces the highest wind loads, while Category 5 produces the lowest.

C Dir - Enter the directional factor, C_{dir}. Values are found in the National Annex. The recommended value is **1.0**.

C Season - Enter the season factor, C_{season}. Values are found in the National Annex. The recommended value is **1.0**.

CsCd - Enter the structural factor C_sC_d, used to determine the force on the vessel. This value is defined in the EN 1991-1-4:2005(E) wind load specification, Annex D. This value normally ranges between **0.90** and **1.10**. The greater the value of the structural factor, the higher the element load.

Force Coefficient Cf - Enter the force coefficient C_f. This value accounts for the fact that the vessel is circular in cross section and is used to modify the area of the vessel that the wind is blowing against. This value is quite often specified in the design specifications or can be calculated based on the methodology given in Section 7.9 for circular cylinders. A typical value for C_f is between **0.7** and **0.8**.

Base Height - Enter in the distance from the bottom of the vessel to base (ground) elevation, in cases where vessels are not fixed to the ground, but are attached to other structures.

Euro Code Wind Data

In common with other wind codes, the principles of the European code are basically the same. A moving air mass causes forces to be exerted on anything that obstructs its path. The starting point of any wind action analysis starts with the Bernoulli equation that converts the kinetic energy of the wind into a static pressure. This pressure simply attempts to move anything in its path. In the case of a vertical tower, the wind deflects the tower in the direction of the flowing wind. Because of the friction, the wind (which is a fluid) passes around the vessel, creating small eddie currents called vortices on either side of the vessel. The spinning air loses some of it pressure as its angular kinetic energy increases. This is again an effect predicted by the Bernoulli equation. This spinning eddie, or vortex, causes a lowering of the pressure on one side of the vessel. As these vortices occur alternatively on the sides of the vessel, the tower tends to sway from side to side perpendicular to the direction of the wind, vibrating like an upside-down pendulum.

The wind pressure causes a static deflection in the direction of the wind stream, and the vortices cause the vessel to vibrate at right angles to the wind stream (vector) as mentioned above.

Static deflection and dynamic vibration cause the vessel to experience longitudinal, or axial bending stresses. Both the stresses and deflection must be limited to provide for good design.

As stated above, the wind kinetic energy is converted to pressure, but there are other factors influencing the final wind pressure. Among these factors are (but not limited to) the following:

- The height, or altitude, of the tower or vessel in the wind stream.
- The elevation of the site on which the vessel is installed.
- The base elevation of the vessel if is supported on an elevated platform or structure.
- The effect of the surround geography, such as hills and escarpments.
- The effect of upstream buildings.

To perform a reasonable analysis, you must enter sufficiently accurate information for the software to apply the correct factors and special calculations for specific cases.

Ref. Wind Velocity Vref,0 - Enter the reference wind velocity V_{ref}, taken from section 7.2 of the code. Annex A provides the values:

Country	**Suggested Wind Velocities (SI Metric Unit System)**
Austria	Special consideration must be given, depending on the district.
Belgium	26.2 m/s
Denmark	27.0 m/s
Finland	23.0 m/s
France	24.0 m/s to 34.0 m/s, depending on the district (Departements et Cantons).
Germany	24.3 m/s to 31.5 m/s, depending on the particular Zone.
Greece	30.0 m/s to 36.0 m/s, depending whether coastal or country.

Iceland	The wind speed must be calculated - See Annex A.8.
Ireland	27.0 m/s to 30 m/s, depending on the region and exposure to the Atlantic Ocean.
Italy	25.0 m/s to 31 m/s, depending on the region.
Luxembourg	26.0 m/s
Netherlands	22.5 m/s to 30.0 m/s, depending on return period and area.
Norway	25.0 m/s to 65 m/s. Please refer to Figure A7 in the code.

Terrain Category - Select the terrain category, taken from Section 8.3 of the code:

Terrain Category	**k_T**	**Z_0 [m]**	**Z_{min} [m]**	**ε**	
I - Flat, Unobstructed	Rough open sea, lakes with at least 5 km fetch upwind and smooth flat country without obstacles	0.17	0.01	2	[0.13]
II - Farmland Areas	Farmland with boundary hedges, occasional small farm structures, houses or trees	0.19	0.05	4	[0.26]
III - Suburban/ Industrial	Suburban or industrial areas and permanent forests	0.22	0.3	8	[0.37]
IV - Urban Areas	Urban areas in which at least 15% of the surface is covered with buildings and their average height exceeds 15 m	0.24	1	16	[0.43]

C Dir - Enter the direction factor C_{Dir}. The direction factor is a function of the country where the vessel is installed. From Annex A:

Country	**C_{Dir}**
Austria	1.0
Denmark	1.0
Finland	1.0
France	1.0
Germany	1.0
Greece	1.0
Iceland	Not Specified

Ireland	1.0
Italy	Not Specified

C Tem - Enter the temperature factor C_{tmp}, which affects the air density. From Annex A:

Country	C_{tmp}
Austria	Not Specified
Denmark	1.0
Finland	1.0
France	1.0
Germany	Please refer to Table shown in A.6. Values range from .30 to 0.65
Greece	1.0
Iceland	Not Specified
Ireland	1.0
Italy	Not Specified

C Alt - Enter the altitude factor C_{Alt}. From Annex A:

Country	C_{Alt}
Austria	Not Specified
Denmark	1.0
Finland	1.0
France	1.0
Germany	Calculated from the formula in section A.6. 1,0 seems to be a safe value.
Greece	1.0
Iceland	Not Specified
Ireland	1.0
Italy	Not Specified

G Peak - Enter the peak factor. In section 8.5 the value is given as 3.5.

Force Coefficient Cf - Enter the force coefficient $C_{f,0}$. For cylindrical surfaces the force coefficient is 0.70. This assumes the Reynolds number of $2.10^5 < Re < 1,2.10^6$. For more information, see Table 10.7.1 in the code. No other values are suggested, so please refer to the code for any relevant information. Section 10.8.1 goes further into the subject where the Reynolds number can be computed. Figure 10.8.2 can also be consulted, but you will need to calculate the Reynolds number.

Base Height - Enter the base height.   This is the height of the base of the vessel above the site (or ground).

Cd (used if > 0.94) - Enter the direction factor C_{dir}. A value of **0** is recommended in the absence of any other information. The software calculates this value, but if a value greater than 0.94 is entered, that value will be used. For more information, see Annex A of the code.

IBC 2006, IBC 2009, and IBC 2012 Wind Data

Basic Wind Speed [V] - Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Terrain Surface Roughness - Select an ASCE-7 exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor [I] - Enter a value for the importance factor. ASCE-7 generally requires a value between 0.77 and 1.15. The software uses this value directly without modification. Values are taken from table 6-2 of the ASCE 95 standard or table 6-1 from the ASCE 98 standard:

Category	**Factor**	**Classification**
I	**0.87**	Buildings and other structures that represent a low hazard to human life in the event of failure. NOTE Most petrochemical structures use this category.

II	**1.00**	Buildings and structures except those listed in categories I, III and IV.
III	**1.15**	Buildings and structures that represent a substantial hazard in the event of a failure.
IV	**1.15**	Buildings designed as essential facilities, such as hospitals.

NOTE In the 98 standard, the importance factor can be **0.77** for category I when wind speeds are greater than 100 miles per hour.

Type of Surface - Enter the roughness factor (from ASCE-7, Table 12):

- **Smooth**
- **Rough** (D'/D = 0.02)
- **Very Rough** (D'/D = 0.08)

NOTES

- Most petrochemical sites use smooth. Some designers use very rough to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab, click **Set Configuration Parameters** to enter it in directly on the **Job Specific Setup Parameters** tab.

Hill Height [H] - Enter the height of a hill or escarpment relative to the upwind terrain. See ASCE 7-95/98, Fig. 6-2 for details.

Distance to Site [x] - Enter the distance (upwind or downwind) from the crest of the hill to the building site. See ASCE 7-95/98 Fig. 6-2 for details.

Crest Distance [Lh] - Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. See ASCE 7-95/98, Fig. 6-2 for more details.

Type of Hill - Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. See ASCE 7-95/98, Fig. 6-2 for details.

IS-875 Wind Data

Basic Wind Speed - Enter a value for basic wind speed. Basic wind speed is based on peak gust velocity averaged over a short time interval of about three seconds and corresponds to mean heights above ground level in an open terrain. This varies according to geographical location and according to company or vendor standards. Typical wind speeds range from 85 to 120 miles per hour. Enter the lowest value reasonably allowed by the standards that you are following. This is because the wind design pressure, and thus force, increases as the square of the speed. This entry is optional, and overrides the basic wind speed determined from the zone entered for **Wind Zone Number**.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Wind Zone Number - Enter a value for the wind zone. India is divided into six wind zones. The software provides the basic wind speed based on the zone. Basic wind speed is applicable to a 10 meter height above mean ground level for different zones India as determined from Figure 1 in IS-875.

Zone	**Basic Wind Speed**
1	33 m/sec 73.82 miles/hour
2	39 m/sec 87.25 miles/hour
3	44 m/sec 98.43 miles/hour
4	47 m/sec 105.15 miles/hour
5	50 m/sec 111.86 miles/hour
6	55 m/sec 123.04 miles/hour

NOTE Optionally, you can enter a value for **Basic Wind Speed**. This value overrides **Wind Zone Number**.

Risk Factor - Enter a value for the risk coefficient *K1*. The value varies based on the zone selected for Wind Zone Number, and assumes 100 years as the mean probable design life of a tower.

Wind Zone	**Risk Factor (K1)**
1	**1.05**
2	**1.06**
3	**1.07**
4	**1.07**
5	**1.08**
6	**1.08**

Terrain Category - Select the category for the terrain in which the vessel is located:

- **Category 1** - Exposed open terrain with few or no obstructions. The average height of any object surrounding the equipment is less than 1.5 m. This category includes open seacoasts and flat treeless plains.
- **Category 2** - Open terrain with well scattered obstructions having height generally between 1.5 to 10 m. This includes airfields, open parklands and undeveloped, sparsely built-up outskirts of towns and suburbs. This category is commonly used for design purposes.
- **Category 3** - Terrain with numerous closely-spaced obstructions with buildings and structures up to 10 m in height. This includes well wooded areas, towns and industrial areas that are fully or partially developed.
- **Category 4** - Terrain with numerous large-height, closely-spaced obstructions. This includes large city centers, generally with obstructions above 25 m, and well developed industrial complexes.

Equipment Class - Select the equipment/component classification:

- **Class A** - Equipment and/or components having a maximum dimension (horizontal or vertical dimension) of less than 20 m.
- **Class B** - Equipment and/or components having a maximum dimension (horizontal or vertical dimension) between 20 and 50 m.
- **Class C** - Equipment and/or components having a maximum dimension (horizontal or vertical dimension) greater than 50 m.

Topography Factor - Enter a topography factor *K3*. This factor takes care of local topographic features such as hills, valleys, cliffs, or ridges which can significantly affect wind speed in their vicinity. The effect of topography is to accelerate wind near summits of hills or crests of cliffs and decelerate the wind in valleys or near the foot of cliffs. Topography effect is significant if upwind slope is greater than about three degrees. Below three degrees, the value of *K3* can be taken as 1.0. For slopes above three degrees, the value of *K3* ranges between 1.0 and 1.36.

Use the Gust Response Factor? - Select to calculate the gust response factor according to IS-875, and use it in the appropriate equations. Select only if the design specifications and the customer explicitly requires the gust response factor. This factor increases the wind load three to six times and may lead to a very conservative wind design.

Use Cf from Table 23? - Select to calculate the drag coefficient C_f from Table 23 of IS-875.

JPI-7R-35-2004 Wind Data

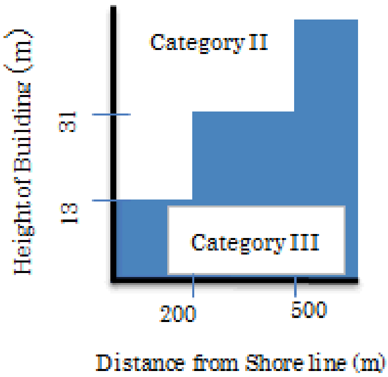
Design Wind Speed - Enter the design value of the wind speed.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than if it were mounted at grade.

Exposure Constant - Select an exposure factor:

<table>
<tr><th colspan="2">Exposure Category</th><th>Z_b (m)</th><th>Z_G (m)</th><th>α</th></tr>
<tr><td>I</td><td>Outside of the town planning area and flat, unobstructed terrain</td><td>5</td><td>250</td><td>0.10</td></tr>
<tr><td>II</td><td>Outside of the town planning area other than Exposure Category I, except for building height is 13 m or less.

Inside of the town planning area other than Exposure Category IV,


</td><td>5</td><td>350</td><td>0.15</td></tr>
<tr><td>III</td><td>Other than Category I, II, and IV</td><td>5</td><td>450</td><td>0.20</td></tr>
<tr><td>IV</td><td>Inside of the town planning area and the center of city</td><td>10</td><td>550</td><td>0.27</td></tr>
</table>

Mexico 1993 Wind Data

Mexico 1993 En Español (on page 379)

Design Wind Speed - Enter the velocity of the wind, V_R, the maximum mean velocity likely to occur within a defined recurrence period in a determined zone or region of the country, as defined by Paragraph 4.6.2 (Isotach Maps regional Velocity).

The isotach maps that are included in this clause, with the different periods of return, such velocities, refer to homogeneous conditions that correspond to a height of 10 meters over the surface of the floor in the flat terrain (Category 2 per table I.1). The maps do not consider the local terrain roughness characteristics or the specific topography of the site. Therefore, such velocity is associated with three-second wind gusts and takes into account the possibility that there might be hurricane winds present in the coastal zones.

The regional velocity, V_R, is determined by taking into account the geographic location of the site of the uproot of the building and its destination.

In figures I.1 through I.4, the isotach regional maps are shown, corresponding to the periods of recurrence for 200, 50 and 10 years.

The importance of the structures (Para. 4.3) dictates the periods of recurrence which should be considered for the wind design. From this, the groups A, B and C in **Structure Class** associate themselves with the periods of return of 200, 50 and 10 years, respectively. The uproot site is located in the map with the recurrence period which corresponds to the group to which the building belongs to, in order to obtain the regional velocity. In the Tomo III from Ayudas de diseño, a table is shown with the main cities in the country and their corresponding regional velocities for the different periods of return.

Site Elev Above Sea Level - Enter the site elevation above mean sea level h_m.

Base Elevation (hb) - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Structure Class - Select a value for the structure class, from Paragraph 4.6.1, Table I.2:

- **A** - Horizontal or vertical structures measuring less than 20 meters of length. Every remote structural element is exposed directly to the wind action.
- **B** - Horizontal or vertical structures measuring between 20 and 50 meters of length
- **C** - Horizontal or vertical structures measuring more than 50 meters of length

Factors Affected by Structure Class

Paragraph 4.6.3.1 size factor F_C, α, and δ. The size factor, F_C, takes into consideration the time in which the gust of wind works in an effective manner over a building of given dimensions. Considering the classification of the structures size (table I.2), this factor can be determined according to table I.3.

Structure Class	F_C
A	1.0
B	0.95
C	0.90

The coefficients α and δ perform with the **Terrain Category**. Table I.4 below assigns the values for these coefficients. In Figure III.1 from *Tomo de Ayudas de diseño*, it is shown the factor's variation Fα with the height, with the terrain category and the structure class.

Terrain Category	α			δ
	Structure Class			
	A	B	C	
1	0.099	0.101	0.105	245
2	0.128	0.131	0.138	315
3	0.156	0.160	0.171	390
4	0.170	0.177	0.193	455

δ - The average height from the level of the land of uprooted above, by which the variation of the velocity of the wind is not important and can be supposed constant. This height is known as the gradient height; δ and Z are given in meters.

α - The exponent that determines the variation form of the wind velocity with the height. This value is unitless.

Terrain Category - Select the category for the terrain in which the vessel is located, from Paragraph 4.6.1, Table I.1:

Cat	**Description**	**Examples**	**Limitations**
1	Open terrain, practically smooth and without obstructions.	Coastal flat stripes, swamp zones, aerial fields, pasture fields and crop lands with no hedges or fences. Flat snow-covered surfaces.	The minimum length for this type of terrain in the direction of the wind must be of 2000 m or 10 times the height of the structure to be designed.
2	Flat terrain or undulated, with few obstructions.	Crop lands or farms with few obstructions around such as hedges of fences, trees and scattered buildings.	The obstructions have a height of 1.5 to 10 m, in a minimum length of 1500 m.
3	Terrain covered by many obstructions narrowly spaced out.	Urban, suburban areas and forests, or any other terrain with many obstructions widely scattered. The sizes of the buildings are like the houses and living spaces.	The obstructions have a 3 to 5 m height. The minimum length for this type of terrain in the direction of the wind must be 500 m or 10 times the height of the structure.
4	Terrain with many big, tall, and narrowly spaced-out obstructions.	Cities with downtown areas and well developed industrial complex areas.	At least 50% of the buildings have a height of more than 20 m. The obstructions measure up from 10 to 30 m in height. The minimum length for this type of land in the direction of the wind should be the biggest between 400 m and 10 times the height of the construction.

Factors Affected by Terrain Category

The factors κ', η, and δ, are related to the terrain category, as defined in Paragraph 4.9.3.2, Table I.29. κ' η are unitless, while δ is the height gradient in m.

	Terrain Category			
Factor	**1**	**2**	**3**	**4**
κ'	1.224	1.288	1.369	1.457
η	-0.032	-0.054	-0.096	-0.151
δ	245	315	390	455

The factors α' and k_r are also related to the terrain category, k_r is a factor related to the roughness of the land.

	Terrain Category			
Factor	**1**	**2**	**3**	**4**
k_r	0.06	0.08	0.10	0.14
α'	0.13	0.18	0.245	0.31

Year of Analysis - Select the analysis year to use.

Topographic Factor (Ft) - Enter a value for the topography local factor F_t, from Paragraph 4.5.4, Table I.5. This factor takes into account the local topographic effect from the place in which the structure will uproot. For example, for buildings found on hillsides, on top of hills, or on mountains at important heights with respect to the general level of the surrounding terrain, it is probable that wind accelerations will generate and the local velocity should be increased. Experts in local wind velocities should justify and validate the results of any of these procedures.

Places	**Topography**	F_t
Protected	Promontories base and skirts of mountain ranges from the leeward side.	**0.8**
	Closed valleys.	**0.9**
Normal	Flat terrain, Open field, absence of important topographical changes, with smaller slopes than 5%.	**1.0**
Exposed	Lands inclined with slopes between 5 and 10%, flat coastal and open valleys.	**1.1**
	Top of promontories, hills or mountains, terrains with greater slopes than 10%, glens closed and valleys that form a funnel or islands.	**1.2**

Damping Factor (Zeta) - Enter a value for the damping factor ζ. Enter **0.01** for steel framework construction. Enter **0.02** for concrete framework construction.

Drag Coefficent (Ca) - Enter a value for the drag coefficient C_a, from Paragraph 4.8.2.12 (Chimneys and Towers), Table I.28:

Cross Section	Type of Surface	Drag Coefficient C_a for H/b of:			
		1	7	25	≥ 40
Circular ($bV_D \geq 6\ m^2/s$)	Smooth or little rough (d'/b ≈ 0.0)	**0.5**	**0.6**	**0.7**	**0.7**
	Rough (d'/b ≈ 0.02)	**0.7**	**0.8**	**0.9**	**1.2**
	Very rough (d'/b ≈ 0.08)	**0.8**	**1.0**	**1.2**	**1.2**
Circular ($bV_D \geq 6\ m^2/s$)	Any	**0.7**	**0.8**	**1.2**	**1.2**
Hexagonal or octagonal	Any	**1.0**	**1.2**	**1.4**	**1.4**
Square (wind normal to a face)	Any	**1.3**	**1.4**	**2.0**	**2.2**
Square (wind on a corner)	Any	**1.0**	**1.1**	**1.5**	**1.6**

NOTES

- **b** - The diameter or the horizontal dimension of the structure, including the roughness of the wall. To determine the product bV_D, this diameter is located at two-thirds of the total height from the level of the land, in m.
- **d'** - The dimension that extends from the roughness, such as ribs or "spoilers", in m.
- **V_D** - The velocity of the wind of design (4.6), in m/s at two thirds of the total height
- For intermediate values of *H/b* and *d'/b*, lineal interpolation is permitted.

Strouhal Number (St) - Enter a value for the Strouhal number S_t. Enter **0.2** for circular sections or **0.14** for rectangular sections.

Barometric Height (Omega) - Enter the barometric pressure Ω. This value is related to **Site Elev Above Sea Level**, as described in Paragraph 4.6.5, Table I.7:

Height (meters above sea level)	Barometric Pressure (mm Hg)
0	760
500	720
1000	675

1500	635
2000	600
2500	565
3000	530
3500	495

NOTE Interpolate for intermediate values.

Ambient Temperature (T) - Enter a value for the ambient temperature *T*.

Mexico 1993 En Español

See *Mexico 1993 Wind Data* (on page 374) for the English version of this text.

Párrafo 4.6.1

Tabla I.1 CATEGORIA DEL TERRENO SEGUN SU RUGOSIDAD

Cat.	Descripción	Ejemplos	Limitaciones
1	Terreno abierto, prácticamente plano y sin obstrucciones	Franjas costeras planas, zonas de pantanos, campos aéreos, pastizales y tierras de cultivo sin setos o bardas alrededor. Superficies nevadas planas	La longitud mínima de este tipo de terreno en la dirección del viento debe ser de 2000 m o 10 veces la altura de la construcción por diseñar, la que sea mayor.
2	Terreno plano u ondulado con pocas obstrucciones	Campos de cultivo o granjas con pocas obstrucciones tales como setos o bardas alrededor, árboles y construcciones dispersas	Las obstrucciones tienen alturas de 1.5 a 10 m, en una longitud mínima de 1500 m
3	Terreno cubierto por numerosas obstrucciones estrechamente espaciadas	Áreas urbanas, suburbanas y de bosques, o cualquier terreno con numerosas obstrucciones estrechamente espaciadas. El tamaño de las construcciones corresponde al de las casas y viviendas	Las obstrucciones presentan alturas de 3 a 5 m. La longitud mínima de este tipo de terreno en la dirección del viento debe ser de 500 m o 10 veces la altura de la construcción, la que sea mayor
4	Terreno con numerosas obstrucciones largas, altas y estrechamente espaciadas	Centros de grandes ciudades y complejos industriales bien desarrollados.	Por lo menos el 50% de los edificios tiene una altura mayor que 20 m. Las obstrucciones miden de 10 a 30 m de altura. La longitud mínima de este tipo de terreno en la dirección del viento debe ser la mayor entre 400 m y 10 veces la altura de la construcción.

Tabla I.2 CLASE DE ESTRUCTURA

Clase	Descripcion
A	Todo elemento estructural aislado, expuesto directamente a la accion del viento; Construcciones horizontales o verticales cuya mayor dimension, sea menor que 20 metros.
B	Construcciones horizontales o verticales cuya mayor dimension, varie entre 20 y 50 metros.
C	Construcciones horizontales o verticales cuya mayor dimension, sea mayor que 50 metros.

Párrafo 4.6.2 MAPAS DE ISOTACAS. VELOCIDAD REGIONAL, V_R

La velocidad regional del viento, V_R, es la máxima velocidad media probable de presentarse con un cierto periodo de recurrencia en una zona o región determinada del país.

En los mapas de isotacas que se incluyen en este inciso con diferentes periodos de retorno, dicha velocidad se refiere a condiciones homogéneas que corresponden a una altura de 10 metros sobre la superficie del suelo en terreno plano (Categoría 2 según la tabla I.1); es decir, no considera las características de rugosidad locales del terreno ni la topografía específica del sitio. Asimismo, dicha velocidad se asocia con ráfagas de 3 segundos y toma en cuenta la posibilidad de que se presenten vientos debidos a huracanes en las zonas costeras.

La velocidad regional, V_R, se determina tomando en consideración tanto la localización geográfica del sitio de desplante de la estructura como su destino.

En las figures I.1 a I.4 se muestran los mapas de isotacas regionales correspondientes a periodos de recurrencia de 200, 50 y 10 anos, respectivamente.

La importancia de las estructuras (véase el inciso 4.3) dictamina los periodos de recurrencia que deberán considerarse para el diseño por viento; de esta manera, los Grupos A, B y C se asocian con los periodos de retorno de 200, 50 y 10 años, respectivamente. El sitio de desplante se localizará en el mapa con el periodo de recurrencia que corresponde al grupo al que pertenece la estructura a fin de obtener la velocidad regional. En el Tomo III de Ayudas de diseño se presenta una tabla con las principales ciudades del país y sus correspondientes velocidades regionales para diferentes periodos de retorno.

Párrafo 4.6.3 FACTOR DE EXPOSICION, Fα

El coeficiente Fα refleja la variación de la velocidad del viento con respecto a la altura Z. Asimismo, considera el tamaño de la construcción o de los elementos de recubrimiento y las características de exposición.

El factor de exposición se calcula con la siguiente expresión:

$$F\alpha = F_C \, F_{RZ}$$

en donde:

F_C : es el factor que determina la influencia del tamaño de la construcción, adimensional, y

F_{RZ} : el factor que establece la variación de la velocidad del viento con la altura Z en función de la rugosidad del terreno de los alrededores, adimensional.

Los coeficientes F_C y F_{RZ} se definen en los incisos 4.6.3.1 y 4.6.3.2, respectivamente.

Párrafo 4.6.3.1 FACTOR DE TAMAÑO, F_C **α y δ**

El factor de tamaño, F_C, es el que toma en cuenta el tiempo en el que la ráfaga del viento actúa de manera efectiva sobre una construcción de dimensiones dadas. Considerando la clasificación de las estructuras según su tamaño (véase la tabla I.2), este factor puede determinarse de acuerdo con la tabla I.3.

Tabla I.3 FACTOR DE TAMAÑO, F_C

Clase de estructura	F_C
A	1.0
B	0.95
C	0.90

δ : es la altura, media a partir del nivel del terreno de desplante, por encima de la cual la variación de la velocidad del viento no es importante y se puede suponer constante; a esta altura se le conoce como altura gradiente; δ y Z están dadas en metros, y

α : el exponente que determina la forma del la variación de la velocidad del viento con la altura y es adimensional.

Los coeficientes α y δ están en función de la rugosidad del terreno (tabla I.1). En la tabla I.4 se consignan los valores que se aconsejan para estos coeficientes. En la figura III.1 del Tomo de Ayudas de diseño se muestra la variación del factor Fα con la altura, con la categoría del terreno y con la clase de estructura.

Tabla I.4 VALORES DE α y δ

Categoría de terreno	α			δ
	Clase de estructura			
	A	B	C	
1	0.099	0.101	0.105	245
2	0.128	0.131	0.138	315
3	0.156	0.160	0.171	390
4	0.170	0.177	0.193	455

Párrafo 4.5.4 FACTOR DE TOPOGRAFIA, F_T

Este factor toma en cuenta el efecto topográfico local del sitio en donde se desplantara la estructura. Así, por ejemplo, si la construcción se localiza en las laderas o cimas de colinas o montañas de altura importante con respecto al nivel general del terreno de los alrededores, es muy probable que se generen aceleraciones del flujo del viento y, por consiguiente, deberá incrementarse la velocidad regional.

Tabla I.5 FACTOR DE TOPOGRAFIA LOCAL, F_T

Sitios	**Topografía**	**F_T**
Protegidos	Base de promontorios y faldas de serranías del lado de sotavento.	0.8
	Valles cerrados.	0.9
Normales	Terreno prácticamente plano, campo abierto, ausencia de cambios topográficos importantes, con pendientes menores que 5%.	1.0
Expuestos	Terrenos inclinados con pendientes entre 5 y 10%, valles abiertos y litorales planos.	1.1
	Cimas de promontorios, colinas o montañas, terrenos con pendientes mayores que 10%, cañadas cerradas y valles que formen un embudo o cañon, islas.	1.2

Expertos en la materia deberán justificar y validar ampliamente los resultados de cualquiera de estos procedimientos.

Párrafo 4.6.5 VALORES DE LA ALTITUD, h_m

Tabla I.7 RELACION ENTRE LOS VALORES DE LA ALTITUD, h_m

Altitud (msnm)	**Presión barométrica (mm de H_g)**
0	760
500	720
1000	675
1500	635
2000	600
2500	65
3000	530
3500	495

Nota: Puede interpolarse para valores intermedios de la altitud, h_m

Párrafo 4.8.2.12 CHIMENEAS Y TORRES

Tabla I.28 COEFICIENTE DE ARRASTRE, C_a

Sección transversal	Tipo de superficie	Relación H/b			
		1	**7**	**25**	**≥ 40**
Circular ($bV_D \geq 6$ m^2/s)	Lisa o poco rugosa (d'/b ≈ 0.0)	0.5	0.6	0.7	0.7
	Rugosa (d'/b ≈ 0.02)	0.7	0.8	0.9	1.2
	Muy rugosa (d'/b ≈ 0.08)	0.8	1.0	1.2	1.2
Circular ($bV_D \geq 6$ m^2/s)	Cualquiera	0.7	0.8	1.2	1.2
Hexagonal u octagonal	Cualquiera	1.0	1.2	1.4	1.4
Cuadrada (viento normal a una cara)	Cualquiera	1.3	1.4	2.0	2.2
Cuadrada (viento sobre una esquina)	Cualquiera	1.0	1.1	1.5	1.6

b : es el diámetro o la dimensión horizontal de la estructura, incluyendo la rugosidad de la pared; para determinar el producto **bV_D**, este diámetro será el que se localiza a dos tercios de la altura total, a partir del nivel del terreno, en m.

d' : es la dimensión que sobresale de las rugosidades, tales como costillas o "spoilers", en m.

V_D : es la velocidad del viento de diseño (inciso 4.6), convertida a m/s, y valuada para los dos tercios de la altura total.

Para valores intermedios de **H/b** y **d'/b** se permite la interpolación lineal.

Párrafo 4.9.3.2 LAS VARIABLES κ' η δ:

Tabla I.29 FACTORES κ' η δ

Categoría	1	2	3	4
κ'	1.224	1.288	1.369	1.457
η	-0.032	-0.054	-0.096	-0.151
δ	245	315	390	455

Las variables κ' η δ, adimensionales, dependen de la rugosidad del sitio de desplante, y δ es la altura gradiente en m. Estas variables se definen en la tabla I.29.

k_r : es un factor relacionado con la rugosidad del terreno:

ζ : es el coeficiente de amortiguamiento critico:
para construcciones formadas por marcos de acero = 0.01
para construcciones formadas por marcos de concreto = 0.02

VALORES DE α' k_r

Categoría de terreno	α'	k_r
1	0.13	0.06
2	0.18	0.08
3	0.245	0.10
4	0.31	0.14

VALORES DE ζ

Nota:	ζ
Para construcciones formadas por marcos de acero	0.01
Para aquellas formadas por marcos de concreto	0.02

NUMERO DE STROUHAL

S_t : es el numero de Strouhal, adimensional; 0.2 para secciones circulares y 0.14 para las rectangulares.

NBC-95 and NBC-2005 Wind Data

Reference Wind Pressure - Enter the reference value of the wind pressure, q. This varies according to geographical location and according to company or vendor standards.

Design Wind Speed - Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are **85.0**, **100.0**, **110.0**, and **120.0**.

NOTE Use the lowest value reasonably allowed because the wind design pressure and force increase as the square of the speed.

Importance Factor (Iw) - Enter a value for the NBC wind importance factor, I_w. The software uses this value directly without modification. Values are taken from Table 4.1.7.1 of Division B.

Category	Importance Factor, I_w	
	ULS	**SLS**
Low	**0.8**	**0.75**
Normal	**1**	**0.75**
High	**1.15**	**0.75**
Post Disaster	**1.25**	**0.75**

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant - Select an NBC exposure factor, as defined in UBC-91, Section 2312:

- **Not Used**
- **Exposure A** - **Open/Standard** exposure.

  NOTE Most petrochemical sites use exposure A.
- **Exposure B** - **Rough** areas similar to urban and suburban areas.
- **Exposure C** - **Very Rough** area similar to centers of large cities.

NOTE These exposure factors are in reverse order from the exposure factors of ASCE-7 or UBC.

Roughness Factor - Select the roughness factor:

- **Moderately smooth**
- **Rough surface** (D'/D = 0.02)
- **Very rough surface** (D'/D = 0.08)

NOTES

- Most petrochemical sites use **Moderately smooth**.
- You can use **Very rough surface** to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab and click **Set Configuration Parameters**. On the **Job Specific Setup Parameters** tab, enter that value in the **Wind Shape Factor** dialog box.

Hill Height [Hh] - Enter the height of a hill or escarpment relative to the upwind terrain. See ASCE 7-95/98, Fig. 6-2 for details.

Distance to site [x] - Enter the distance (upwind or downwind) from the crest of the hill to the building site. See ASCE 7-95/98 Fig. 6-2 for details.**Crest Distance [Lh]** - Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. See ASCE 7-95/98, Fig. 6-2 for more details.

Type of Hill - Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. See ASCE 7-95/98, Fig. 6-2 for details.

Beta: Operating - In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, pages 246, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty - In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full - In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

NBC-2010 Wind Data

Reference Velocity Pressure (q) - Enter the reference value of the wind pressure, q. This varies according to geographical location and according to company or vendor standards.

Design Wind Speed - Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are **85.0**, **100.0**, **110.0**, and **120.0**.

NOTE Use the lowest value reasonably allowed because the wind design pressure and force increase as the square of the speed.

Importance Factor (Iw) - Enter a value for the NBC wind importance factor, I_w. The software uses this value directly without modification. Values are taken from Table 4.1.7.1 of Division B.

Category	Importance Factor, I_w	
	ULS	SLS
Low	**0.8**	**0.75**
Normal	**1**	**0.75**
High	**1.15**	**0.75**
Post Disaster	**1.25**	**0.75**

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant - Select an NBC exposure factor, as defined in UBC-91, Section 2312:

- **Not Used**
- **Exposure A** - **Open/Standard** exposure.

NOTE Most petrochemical sites use exposure A.

- **Exposure B** - **Rough** areas similar to urban and suburban areas.
- **Exposure C** - **Very Rough** area similar to centers of large cities.

NOTE These exposure factors are in reverse order from the exposure factors of ASCE-7 or UBC.

Surface Roughness - Select the roughness factor:

- **Moderately smooth**
- **Rough surface** (D'/D = 0.02)
- **Very rough surface** (D'/D = 0.08)

NOTES

- Most petrochemical sites use **Moderately smooth**.
- You can use **Very rough surface** to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab and click **Set Configuration Parameters**. On the **Job Specific Setup Parameters** tab, enter that value in the **Wind Shape Factor** dialog box.

Hill Height [Hh] - Enter the height of a hill or escarpment relative to the upwind terrain, H_h. For more information, see ASCE 7-10, Figure 29.8-1.

Distance to Site [x] - Enter the distance (upwind or downwind) from the crest of the hill to the building site. For more information, see ASCE 7-10, Figure 29.8-1.

Crest Distance [Lh] - Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment, L_h. For more information, see ASCE 7-10, Figure 29.8-1.

Type of Hill - Select the type of hill: **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. For more information, see ASCE 7-10, Figure 29.8-1.

Beta: Operating - In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, pages 246, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty - In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full - In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

SANS 10160-3:2010 Wind Data

South African National Standard (SANS) wind data code guidelines detail standards for determining natural wind actions for the structural design of buildings and industrial structures. The standard focuses on predicting the characteristics of wind actions on land-based structures with an overall height of 100 m or less, elements of buildings and structures having a natural frequencies greater than 5 Hz, and chimneys with circular cross-sections with heights of less than 60 m and a height-to-diameter ratio of less than 6, 5.

NOTE This part of SANS 10160 does not apply to structures and buildings higher than 100 m, dynamic effects and design of dynamically-sensitive structures (such as slender chimneys), or off-shore structures.

Percent Wind for Hydrotest - Enter the percentage of the wind load (not wind speed) that is applied during the hydrotest. This is typically as low as 33 percent of the design wind load, because it can be assumed that the vessel will not be hydrotested during a hurricane or severe storm.

Basic Wind Speed - Enter the basic wind speed for the vessel location, which you can find from the wind speed map of South Africa in SANS 10160-3 or as needed by end user requirements.

Terrain Category - Enter the terrain roughness category:

Category	**Description**
A	Flat, horizontal terrain with negligible vegetation and without any obstacles (for example, coastal areas exposed to open sea or large lakes).
B	Area with low vegetation, such as grass and isolated obstacles (for example, trees and buildings) with separations of at least 20 obstacle heights.
C	Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, and permanent forest).
D	Area in which at least 15 percent of the surface is covered with buildings and their average height exceeds 15 m.

For more information on the roughness factors and height parameters that apply to the various terrain categories, see Section 7.3.2, *Terrain roughness,* in SANS 10160-3:2010, Edition 1.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than if it were mounted at grade.

Site Altitude Above Sea Level - Enter the site altitude above mean sea level. Refer to Table 4, *Air density as a function of site altitude*, in Section 7.4 of SANS 10160-3:2010 for the recommended values of the air density as a function of the altitude above sea level.

Topography Factor (Co) - Enter the topography factor (Co), which determines the largest increase in the window speed near the top of the slope. The topography factor accounts for the increase of mean wind speed over isolated hills and escarpments only, and is also determined by the wind speed at the base of the hill or escarpment. For more information on when to consider the effects of topography, see SANS 10160-3:2010, *Section A.3 Numerical Calculations of Topography Coefficients*.

Force Coefficient (Cf) - Enter the force coefficient value (Cf), which is the overall effect of the wind on a structure, structural element, or component as a whole. The force coefficient includes friction, if not specifically excluded.

Beta: Operating - In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, pages 246, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty - In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full - In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

UBC Wind Data

Design Wind Speed - Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation - Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant - Select an UBC exposure factor, as defined in UBC-91 Section 2312:

- **Exposure B** - Terrain with building, forest, or surface irregularities 20 feet or more in height, covering at least 20 percent or the area extending one mile or more from the site.
- **Exposure C** - Terrain that is flat and generally open, extending one-half mile or more from the site in any full quadrant.

  NOTE Most petrochemical sites use this value. It is used to set the gust factor coefficient (Ce) found in Table 23-G.
- **Exposure D** - The most severe exposure with basic wind speeds of 80 miles per hour or higher. Terrain is flat and unobstructed, facing large bodies of water over one mile or more in

width relative to any quadrant of the building site. This exposure extends inland from the shoreline one quarter mile or zero times the building (vessel) height, whichever is greater.

- **Not Used**

Importance Factor - Enter a value for the UBC importance factor. The software uses this value directly without modification. Values are taken from Table 23-L of the UBC standard:

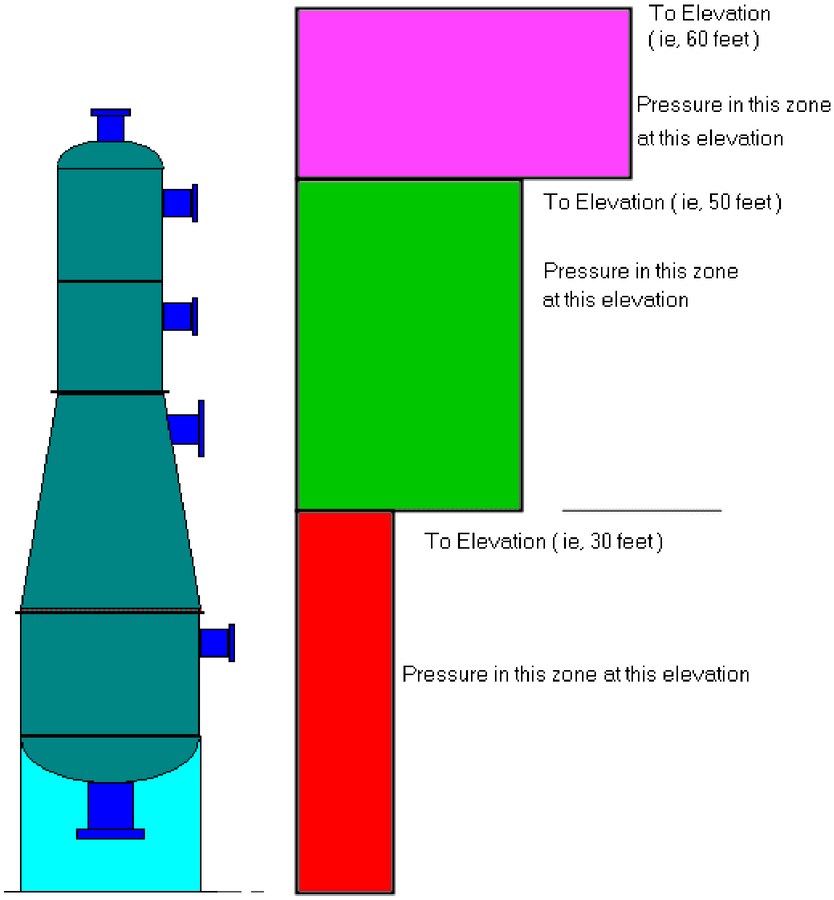
Category	Factor	Classification
I	**1.15**	Essential facilities
II	**1.15**	Hazardous facilities
III	**1.0**	Special occupancy structures
IV	**1.0**	Standard occupancy structures

User-Defined Wind Data

When **User Defined** is selected for **Wind Design Code**, enter values for wind pressure versus elevation in the table. The software uses only the values from the table, forgoing all code calculations. Enter **"To" Height**, the height above grade, in the left text boxes, and **Pressure**, the wind pressure at that height, in the right text boxes. If you have more rows available than you need to describe the wind profile, enter zeros in the remaining cells.

NOTE Multiply the wind pressure at each elevation by the shape factor you wish to use. If you do not your wind loads will be higher (and more conservative) than needed.

The first **"To" Height** value cannot be zero. If it is zero the software will not calculate the wind loads on the following elements. Values should follow the convention below



SECTION 16

Seismic Loads (Seismic Data Tab)

Click **Input** > **Seismic Loads** to enter seismic data. The **Seismic Data** tab appears on the left. Select a seismic design code and enter data required by that code. If no data is entered, the software uses the system defaults.

Seismic Design Code - Select the design code to use for seismic calculations:

Code	**Description**
As/Nz 1170.4	Australian Standard Part 4: Earthquake Loads, 1993 edition For more information, see *AS/NZ 1170.4 Seismic Data* (on page 395).
ASCE-88 **ASCE-93** **ASCE-95** **ASCE-7-98** **ASCE 7-02/05** **ASCE-2010**	American Society of Civil Engineers Standard 7 (formerly ANSI A58.1). For more information, see *ASCE-88 Seismic Data* (on page 397). For more information, see *ASCE-93 Seismic Data* (on page 399). For more information, see *ASCE-95 Seismic Data* (on page 401). For more information, see *ASCE 7-98 Seismic Data* (on page 403). For more information, see *ASCE 7-02/05 Seismic Data* (on page 406). For more information, see *ASCE-2010 Seismic Data* (on page 409).
Chile NCh2369	NCh2369 - Chilean Code for Seismic Analysis of Vessels - 2003 For more information, see *Chile NCh2369 Seismic Data* (on page 412).
China GB 50011	Chinese Seismic Code For more information, see *China GB 50011 Seismic Data* (on page 422).
Costa Rica 2002	Costa Rica Seismic Code 2002 For more information, see *Costa Rica 2002 Seismic Data* (on page 422).
G Loading	Enter an appropriate G loading. The software calculates the mass of the element and multiplies by the G loading in the appropriate direction. For vertical vessels, the maximum of the X and Z acceleration values are chosen for the analysis. For horizontal vessels, all three values are used. For more information, see *G Loading Seismic Data* (on page 425).
IBC 2000 **IBC 2003** **IBC 2006** **IBC 2009**	International Building Code 2000, 2003, 2006, and 2009 editions For more information, see *IBC 2000 Seismic Data* (on page 425). For more information, see *IBC 2003 Seismic Data* (on page 428). For more information, see *IBC 2006 Seismic Data* (on page 431). For more information, see *IBC 2009 Seismic Data* (on page 434).
IS-1893 SCM **IS-1893 RSM**	India's seismic code: Seismic Coefficient Method (SCM), 1984 edition and Response Spectrum Method (RSM), 1984 and 2002 editions. For more information, see *IS-1893 RSM Seismic Data* (on page 437). For more information, see *IS-1893 SCM Seismic Data* (on page 439).

Mexico Sismo	Mexican Seismic Code For more information, see *Mexico Sismo Seismic Data* (on page 440).
NBC-1995 **NBC-2005** **NBC-2010**	National Building Code of Canada, 1995, 2005, and 2010 editions For more information, see *NBC 1995 Seismic Data* (on page 443). For more information, see *NBC 2005 Seismic Data* (on page 444). For more information, see *NBC 2010 Seismic Data* (on page 446).
PDVSA	Venezuelan Code for Seismic Analysis of Vessels - 1999 For more information, see *PDVSA Seismic Data* (on page 447).
Res. Spectrum	Response Spectrum Analysis allows the use of modal time history analysis. The general design guidelines for this analysis are taken from the ASCE 7-98 or IBC 2000 codes. Other predefined spectra are built into the program, such as the 1940 Earthquake El Centro and various spectra form the US NRC guide 1.60. If the spectrum analysis type is user defined, the table of points that define the response spectra must be entered in the table. For tall structures, this analysis gives much more accurate results than the typical static equivalent method. Usually the calculated loads are lower in magnitude than those computed using conventional building code techniques. For more information, see *Res. Spectrum Seismic Data* (on page 459).
SANS 10160-4:2010	South African National Standard 10160, Section 2, 2010 edition. For more information, see *SANS 10160-4:2010 Seismic Data* (on page 463).
UBC 1994 **UBC 1997**	Uniform Building Code, 1994 and 1997 editions For more information, see *UBC 1994 Seismic Data* (on page 464). For more information, see *UBC 1997 Seismic Data* (on page 465).
No Seismic Loads	

Percent Seismic for Hydrotest - Enter the percentage of the total seismic horizontal force that is applied during hydrotest. Although you cannot predict an earthquake (as you can high winds) some designers use a reduced seismic load for hydrotest on the theory that the odds of an earthquake during the test are very low, and the hazards of a water release small.

In This Section

AS/NZ 1170.4 Seismic Data

Enter data for the Australian/New Zealand AS/NZ-1170.4 seismic code, 1993 or 2007.

Code Year - Select the revision year of the code: **1993** or **2007**.

Importance Factor (I) - Enter the value of the importance factor *I*. The software uses this value directly without modification. Values are taken from Table 16-K of the UBC 1997 standard:

Structure Type	Description	Importance Factor
I	Structures include buildings not of type II or type III.	**1.0**
II	Structures include buildings that are designed to contain a large number of people, or people of restricted or impaired mobility.	**1.0**
III	Structures include buildings that are essential to post-earthquake recovery or associated with hazardous facilities.	**1.25**

Structural Response Factor (Rf) - Enter the he structural response factor $R_{f,}$ taken from table 6.2.6(b) of the code. For vessels on legs, use **2.1**. For towers, stacks and chimney type structures, use **2.8**.

Site Factor (S) - Enter the site factor *S*. taken from table 2.4(a) or 2.4(b). The factor is a function of the type of soil on which the vessel sits. This value can range between 0.67 and 2.0. A value of 2 is the most conservative and represents a vessel sitting on a foundation of loose sand or clay, while 0.67 represents a vessel sitting on a rock bed.

Soil Profile Table 2.4(a) **General Structures**	**Site factor (S)**
A profile of rock materials with rock strength Class L (low) or better.	**0.67**
A soil profile with either: ▪ Rock materials Class EL (extreme low) or VL (very low). ▪ Not more than 30m of medium dense to very dense coarse sands and gravels; firm , stiff or hard clays; or controlled fill.	**1.0**
A soil profile with more than 30m of: medium dense to very dense coarse sands and gravels; firm , stiff or hard clays; or controlled fill.	**1.25**
A soil profile with a total depth of 20m or more and containing 6 to 12m of: very soft to soft clays; very loose or loose sands; silts; or uncontrolled fill.	**1.5**
A soil profile with more than 12m of: very soft to soft clays; very loose or loose sands; silts; or uncontrolled fill characterized by shear wave velocities less than 150m/s.	**2.0**

Acceleration Coefficient (ax) - Enter the acceleration coefficient a_x, taken from Table 2.3 or Figures 2.3(b) to 2.3(g). This value ranges from 0.04 to 0.22. The higher the acceleration coefficient, the higher the load on the vessel.

Design Category - Select the design category **A**, **B**, **C**, **D**, or **E**. The software uses this value to determine if it is necessary to apply vertical accelerations. If the selected category is **D** or **E**, vertical accelerations are applied. The vertical acceleration is taken to be 0.5 times **Acceleration Coefficient (ax)** in the horizontal direction, according to paragraph 6.8.

Probability Factor (kp) - Enter a value for the probability factor k_p.

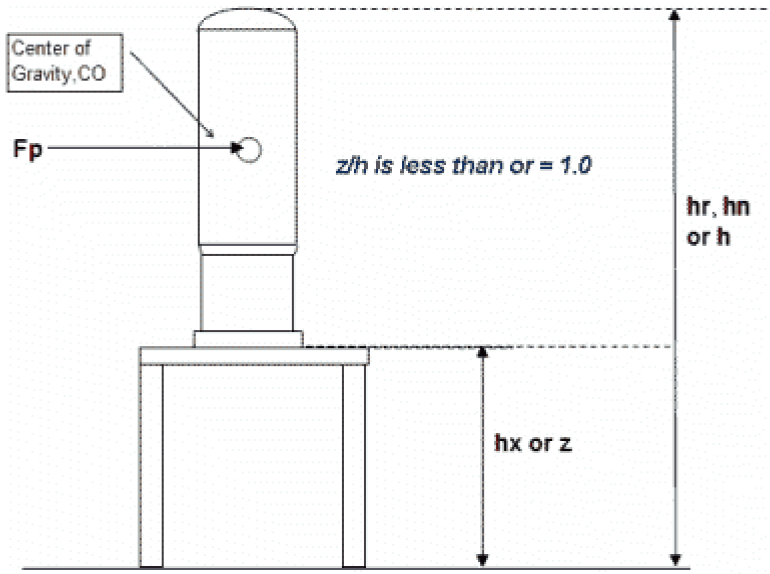
Hazard Factor (Z) - Enter the hazard factor.

Structural Performance Factor (Sp) - Enter the structural performance factor.

Structural Ductility Factor (mu) - Enter the structural ductility factor.

Soil Type - Enter the soil type.

Component Elevation Ratio (hx/hn) - Enter a value for the elevation ratio if the vessel is supported by a building. The ratio, needed for proper analysis, is the height from the bottom of the vessel *hx* to the building height *hn*. A value for **Attachment Factor (ac)** must also be entered. The base shear is calculated according to equation 5.2.1. After the base shear F_p (V) is calculated, it is applied according to the equations in section 6 of the code.



Attachment Amplification Factor (ac) - Enter the attachment amplification factor a_c. This value is typically **1.0** unless unusually flexible connections are provided (which is not for vessels). Because the load is in linear proportion with a_c, the higher the value is, the higher the load becomes. You must also enter a value for **Component Elevation Ratio (hx/hn)**.

ASCE-88 Seismic Data

Enter data for the ASCE 7-88 seismic code.

Importance Factor - Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from Table 22 of the ASCE 7-88 standard:

Category	**Classification**	**Importance Factor**
I	Buildings not listed below. NOTE Most petrochemical structures use this category.	**1.0**
II	High occupancy buildings.	**1.25**
III	Essential facilities.	**1.5**
IV	Low hazard buildings.	not applicable

Soil Type - Select a soil type:

- **Soil 1** - Soil Profile S1. Rock or stiff soil conditions (S Factor = 1.0).
- **Soil 2** - Soil Profile S2. Deep cohesionless deposits or stiff clay conditions (S Factor = 1.2).
- **Soil 3** - Soil Profile S3. Soft to medium-stiff clays and sands (S Factor = 1.5).

The soil type indicates a soil profile coefficient *S*, found Table 24 of the code. Soil profiles are defined in Section 9.4.2 of the code. When soil properties are not known, select **Soil 1** or **Soil 2**, based on the value producing the larger value of CS (C is defined in Eq. 8 of the code).

Horizontal Force Factor - Enter the seismic force factor according to ANSI A58.1, Table 24:

Typical Value	Description
1.33	Buildings with bearing walls
1.00	Buildings with frame systems
2.50	Elevated tanks
2.00	Other structures

NOTE The value most often used is 2.0, though 2.5 is sometimes chosen for tanks supported by structural steel or legs.

Seismic Zone - Select the seismic zone, according to ASCE 7-93, Figure 14:

Zone	Description
0	Gulf and other coastal areas
1	Inland areas
2	Rockies, other mountain areas
3	Central Rockies, other mountains
4	California fault areas

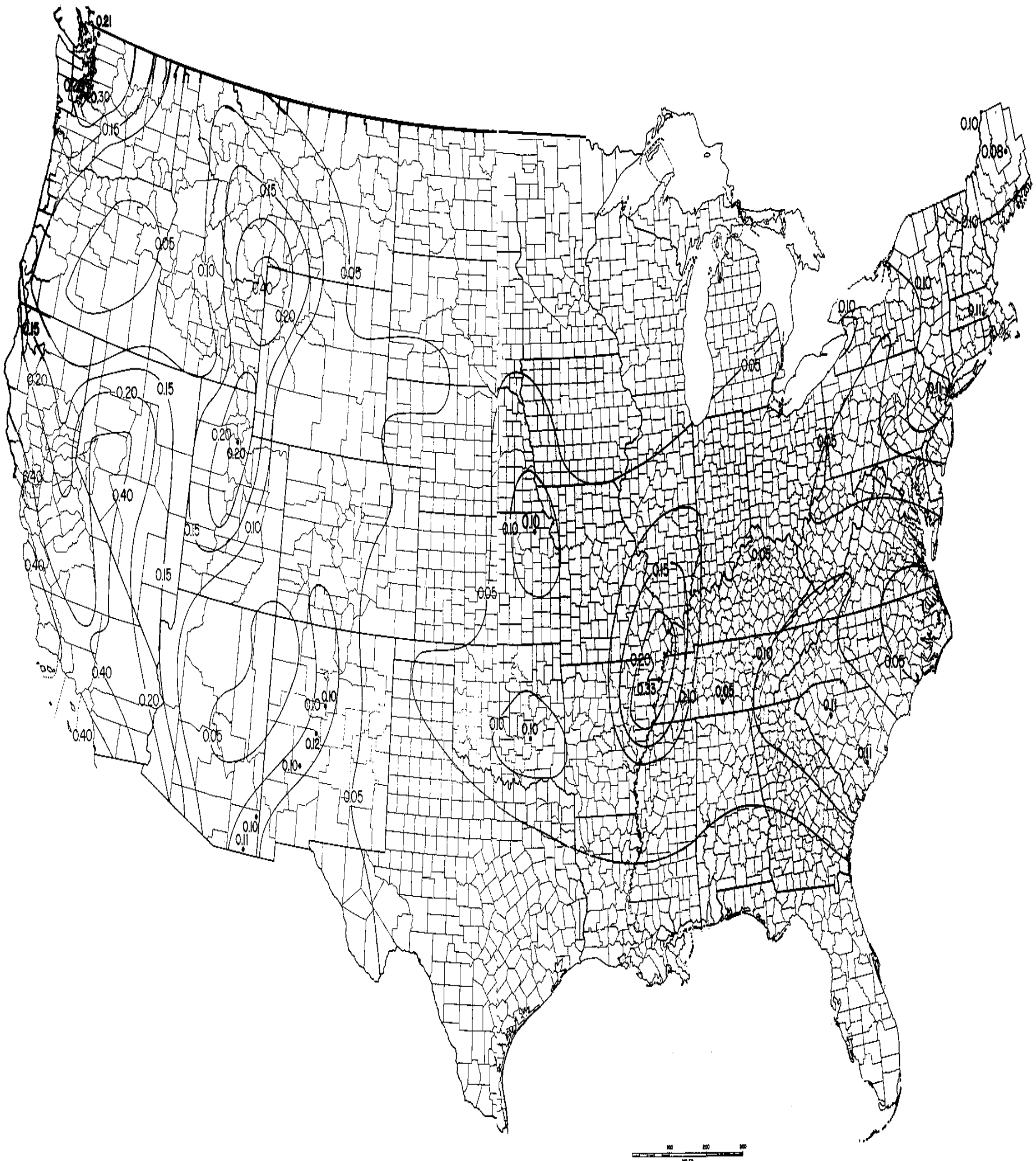
NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 4 indicates the greatest chance of an earthquake.

ASCE-93 Seismic Data

Enter data for the ASCE 7-93 seismic code.

NOTE The ASCE 7 earthquake standards released in 1993 are significantly more involved than the previous standards, more strictly limited to buildings, and thus not as easily applied to vessels. The software does not implement the dynamic analysis according to this standard. However, it does address the computation of the element mass multiplier as outlined on page 62 of the standard. In effect, the factors Av, Cc, P, and ac are first multiplied together and then by the weight of the element to obtain the lateral force on the element. The software then calculates the moments on the tower based on these results. You should have a good understanding of this code before using it.

Seismic Coefficient Av - Enter the seismic coefficient *Av*, representing the effective peak velocity-related acceleration according to the ASCE 7-93 standard, *Minimum Design Loads for Buildings and Other Structures*. This value is obtained from the map on pages 36 and 37 of this standard. According to section 9.1.4.1 of the ASCE Code, this value generally ranges from .05 (low incidence of earthquake) to .4 (high incidence of earthquake).



Seismic Coefficient Cc - Enter the seismic coefficient *Cc* for mechanical and electrical components from table 9.8-2, page 63 of the ASCE 7-93 Code. For tanks, vessels and heat exchangers this value is normally taken as **2.**

Performance Factor - Enter the performance criteria factor *P,* from table 9.8-2, page 63 of the ASCE 7-93 code. This factor depends on the Seismic Hazard Exposure Group., defined in Section 9.1.4.2:

P	Definition
1.5	Seismic Hazard Exposure Group III: Essential facilities required for post-earthquake recovery
1.0	Seismic Hazard Exposure Group II: Buildings that have a substantial public hazard due to occupancy or use
0.5	Seismic Hazard Exposure Group I: All other buildings

Amplification Factor - Enter the amplification factor ac, according to ASCE 7-93, Table 9.8-3:

Component Supporting Mechanism	Attachment Amplification Factor (a_c)
Fixed or direct connection seismic-activated restraining device	1.0
Resilient support system where: Tc/T < 0.6 or Tc/T > 1.4 Tc/T D 0.6 or Tc/T W 1.4	 1.0 2.0

NOTE **T** is the fundamental period of the building in seconds. **Tc** is the fundamental period of the component in seconds.

ASCE-95 Seismic Data

Enter data for the ASCE 7-95 seismic code.

Importance Factor - Enter the importance factor, as given in paragraph 9.3.1.5 of ASCE 95. The value is **1.5** for the following situations:

- Life-safety component required to function after an earthquake (such as, fire protection sprinkler system).
- Component contains material that would be significantly hazardous if released.
- Component poses a significant life safety hazard if separated from primary structure.
- Component can block a means of egress or exit way if damaged (such as, exit stairs).

Otherwise the value is **1.0**. The value may also be specified by the customer, but should always be greater than or equal to 1.0.

Force Factor R - Enter the seismic force factor according to ANSI A58.1, Table 24:

Typical Value	Description
1.33	Buildings with bearing walls
1.00	Buildings with frame systems
2.50	Elevated tanks
2.00	Other structures

NOTE The value most often used is 2.0, though 2.5 is sometimes chosen for tanks supported by structural steel or legs.

Seismic Coefficient Ca - Enter the value of seismic coefficient *Ca* (for shaking intensity) according to table 9.1.4.2.4A on page 55 of ASCE7-95. This factor is a function of the soil profile type and the value of Aa.

Soil Profile Type	**Seismic Coefficient *Ca* for:**						
	Aa<0.05g	**Aa=0.05g**	**Aa=0.10g**	**Aa=0.20g**	**Aa=0.30g**	**Aa=0.40g**	**Aa>=0.5g** [b]
A	**Aa**	**0.04**	**0.08**	**0.16**	**0.24**	**0.32**	**0.40**
B	**Aa**	**0.05**	**0.10**	**0.20**	**0.30**	**0.40**	**0.50**
C	**Aa**	**0.06**	**0.12**	**0.24**	**0.33**	**0.40**	**0.50**
D	**Aa**	**0.08**	**0.16**	**0.28**	**0.36**	**0.44**	**0.50**
E	**Aa**	**0.13**	**0.25**	**0.34**	**0.36**	**0.36**	**a**

a - Site specific geotechnical information and dynamic site response analyses shall be performed.

b - Site specific studies required per Section 9.2.2.4.3 may result in higher values of *Av* than included on hazard maps, as may the provisions of Section 9.2.6.

NOTE For intermediate values, the higher value of straight-line interpolation shall be used to determine the value of *Ca*.

Seismic Coefficient Cv - Enter the value of seismic coefficient *Cv* (for shaking intensity) according to table 9.1.4.2.4B on page 55 of ASCE7-95. This factor is a function of the soil profile type and the value of Aa.

Soil Profile Type	Seismic Coefficient *Cv* for:						
	Aa<0.05g	**Aa=0.05g**	**Aa=0.10g**	**Aa=0.20g**	**Aa=0.30g**	**Aa=0.40g**	**Aa>=0.5g** [b]
A	**Av**	**0.04**	**0.08**	**0.16**	**0.24**	**0.32**	**0.40**
B	**Av**	**0.05**	**0.10**	**0.20**	**0.30**	**0.40**	**0.50**
C	**Av**	**0.09**	**0.17**	**0.32**	**0.45**	**0.56**	**0.65**
D	**Av**	**0.12**	**0.24**	**0.40**	**0.54**	**0.64**	**0.75**
E	**Av**	**0.18**	**0.35**	**0.64**	**0.84**	**0.96**	**a**

a - Site specific geotechnical information and dynamic site response analyses shall be performed.

b - Site specific studies required per Section 9.2.2.4.3 may result in higher values of *Av* than included on hazard maps, as may the provisions of Section 9.2.6.

NOTE For intermediate values, the higher value of straight-line interpolation shall be used to determine the value of *Cv*.

ASCE 7-98 Seismic Data

Importance Factor I - Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R - For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa - Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
Site Class	**S_s <= 0.25**	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analysis shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv - Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
Site Class	**S_l <= 0.1**	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analyses shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss - Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl - Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

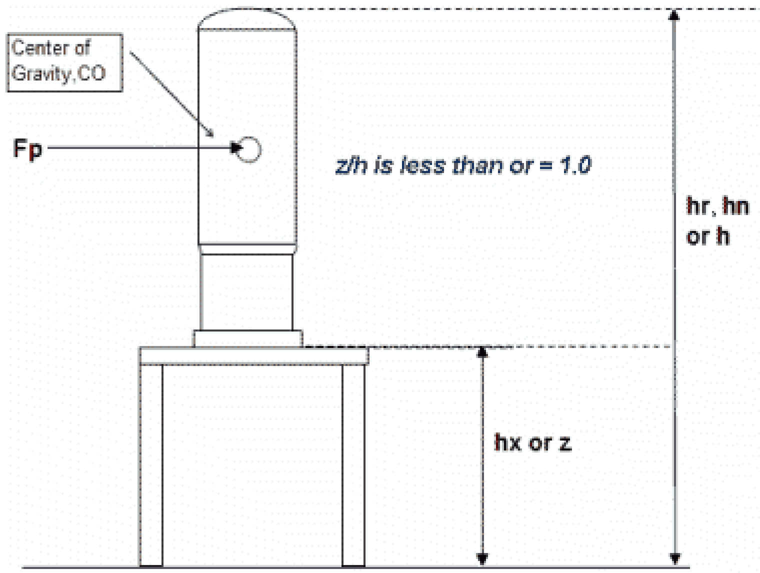
Moment Reduction Factor tau - Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE this value is no longer used and is therefore not applicable.

Site Class - Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code? - Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h - Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap - Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

ASCE 7-02/05 Seismic Data

Importance Factor I - Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R - For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa - Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
	S_s <= 0.25	S_s = 0.5	S_s = 0.75	S_s = 1.0	S_s >= 1.25[2]
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analysis shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv - Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
	S_l <= 0.1	S_l = 0.2	S_l = 0.3	S_l = 0.4	S_l >= 0.5[2]
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analyses shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss - Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl - Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

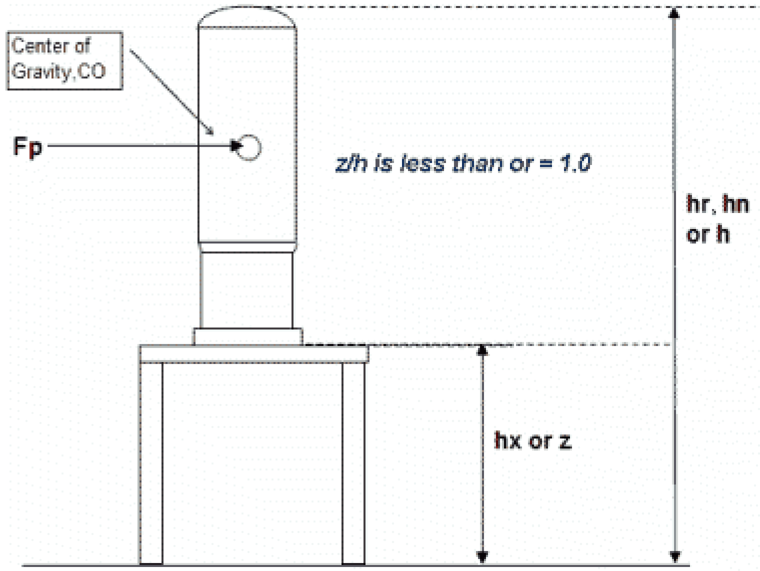
Moment Reduction Factor tau - Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE this value is no longer used and is therefore not applicable.

Site Class - Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code? - Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h - Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap - Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

ASCE-2010 Seismic Data

Enter data for the ASCE-2010 seismic code.

Importance Factor I - Enter the occupancy importance factor, I, according to Sections 11.5.1 and 15.4.1.1, based on Table 1.5-2. The importance factor accounts for loss of life and property. This value typically ranges between **1.0** and **1.5**.

Response Modification Factor R - Enter the seismic force factor, R, from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2, as required. R is usually equal to **2.5** for inverted pendulum systems and cantilevered column systems. For elevated tanks use a value of **4**. For horizontal vessels, leg supported vessels and others use a value of **3.0**.

Acc. based Factor Fa - Enter the short-period site coefficient (at 0.2 s-period), F_a, from Table 11.4-1 as required. For more information, see Section 11.4.3.

Site Class	**S_S<=0.25**	**S_S=0.5**	**S_S=0.75**	**S_S=1.0**	**S_S>=1.25[b]**
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.2	1.2	1.1	1.0	1.0
D	1.6	1.4	1.2	1.1	1.0
E	2.5	1.7	1.2	0.9	[0.9][a]
F	See Section 11.4.7	a	a	a	a

Acc based Factor Fv - Enter the long-period site coefficient (at 1.0 s-period), F_v, from Table 11.4-2 as required. For more information, see Section 11.4.3.

Site Class	**S_1<=0.1**	**S_1=0.2**	**S_1=0.3**	**S_1=0.4**	**S_1>=0.5[b]**
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.7	1.6	1.5	1.4	1.3
D	2.4	2.0	1.8	1.6	1.5
E	3.5	3.2	2.8	2.4	[2.4][a]
F	See Section 11.4.7				

[a] Site specific geotechnical information and dynamic site response analyzes shall be performed.

[b] Site specific studies required per Section 9.4.1.2.4 might result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.

NOTE For intermediate values, the higher value of straight-line interpolation shall be used to determine the value of S_s or S_l.

Max. Mapped Res. Acc. Ss - Enter the value for S_s, the risk-adjusted maximum considered earthquake ground motion parameter for short periods (0.2 seconds), shown in Figures 22-1, 22-3, 22-5, and 22-6. When S_1 is less than or equal to 0.04, and S_s is less than or equal to 0.15, the structure is permitted to be assigned to Seismic Design Category A and is only required to comply with Section 11.7. Do not use percentage values in these fields; use a multiplier of g. For example, if S_s is 25%, use **0.25**.

Max Mapped Res. Acc. S1 - Enter the value for S_1, the risk-adjusted maximum considered earthquake ground motion parameter for long periods (1.0 seconds), shown in Figures 22-2, 22-4, 22-5, and 22-6. When S_1 is less than or equal to 0.04, and S_s is less than or equal to 0.15, the structure is permitted to be assigned to Seismic Design Category A and is only required to comply with Section 11.7. Do not use percentage values in these fields; use a multiplier of g. For example, if S_s is 25%, type **0.25**.

Moment Reduction Value tau - This value is no longer used.

Site Class - Select the site class, as defined by Table 20.3-1.

Class	**Description**
A	Hard Rock
B	Rock
C	Very dense soil and soft rock
D	Stiff soil
E	Soft clay soil
F	Soils requiring site response analysis in accordance with Section 21.1

Component Elevation ratio z/h - Enter the elevation ratio, z/h. If the vessel is attached to another structure, such as a building, the value of the elevation ratio needs to be used for proper analysis. This ratio is the height in the structure where the vessel is attached to the average height of the roof with respect to the base. This value is generally less than or equal to **1**. For more information, see Section 13.3.1.

Component Amplification Factor ap - Enter a value between **1.0** to **2.5**. For vessels, a value of **2.5** is typical. For more information, see Section 13.3.1.

Consider Vertical Accelerations - Select to enable vertical load calculations (YEq).

Force Factor - Enter the force factor applied to the shear vertical load.

Minimum Acceleration Multiplier - Enter the force factor, G_y, applied to the total weight of the vessel.

Sds - Enter the value for S_{DS}, the 5 percent damped, spectral response acceleration parameter at short periods, as defined in Section 11.4.4.

NOTE The software checks Tables 11.6-1 and 11.6-2 to determine the seismic use group. If S_{D1} is greater than or equal to **0.2** and S_1 is greater than or equal to **0.75** then the additional check of equation 16-38 (IBC) is made. The same logic is used if S_{DS} is greater than or equal to **0.50**.

Sd1 - Enter the value for S_{DS}, the 5 percent damped, spectral response acceleration parameter at a period (1 second), defined in Section 11.4.4.

NOTE The software checks Tables 11.6-1 and 11.6-2 to determine the seismic use group. If S_{D1} is greater than or equal to **0.2** and S_1 is greater than or equal to **0.75** then the additional check of equation 16-38 (IBC) is made. The same logic is used if S_{DS} is greater than or equal to **0.50**.

Chile NCh2369 Seismic Data

Chile NCh2369 En Español (on page 415)

Enter data for the Chilean Code for Seismic Analysis of Vessels.

Eff. Max. Ground Acc. (Ao) - Enter the effective maximum ground acceleration A_o, as defined in Table 5.2 Maximum Effective Acceleration, NCh2369 page 39:

Seismic Zone	A_o
1	0.20 g
2	0.30 g
3	0.40 g

Soil Parameter (T') - Enter the soil parameter *T'*, as defined in Table 5.4 Value of type of soil dependent parameters, NCh2369 page 40:

Type of Soil	T' (s)	n
I	0.20	1.00
II	0.35	1.33
III	0.62	1.80
IV	1.35	1.80

T' : Soil type dependent parameter
n : number of levels parameter dependent by the type of soil

Soil Parameter (n) - Enter the soil parameter *n*, as defined in Table 5.4 Value of type of soil dependent parameters, NCh2369 page 40:

Type of Soil	T' (s)	n
I	0.20	1.00
II	0.35	1.33
III	0.62	1.80
IV	1.35	1.80

T' : Soil type dependent parameter
n : number of levels parameter dependent by the type of soil

Damping Ratio (Xi) - Enter the damping ratio ξ, as defined in Table 5.5 Damping Ratios, NCh2369 page 40:

Resistant System	**ξ**
Welded steel shell; silos; hoppers; pressure vessels; process towers; piping, etc	**0.02**
Bolted or riveted steel shell;	**0.03**
Welded steel frames with or without bracings	**0.02**
Steel frames with field bolted connections, with or without bracings	**0.03**
Reinforced concrete and masonry structures	**0.05**
Precast reinforced concrete, purely gravitational structures	**0.05**
Precast reinforced concrete structures with wet connections, connected to the non-structural elements and incorporated into the structural model	**0.05**
Precast reinforced concrete structures with wet connections, non-connected to the non-structural elements	**0.03**
Precast reinforced concrete structures with dry connections, non-connected and connected: With bolted connections and connections by means of bars embedded in filling mortar With welded connections	**0.03** **0.02**
Other structures not included in above list nor similar to the foregoing ones	**0.02**

NOTE See notes at the bottom of Table 5.5 in NCh2369, page 40.

Response Mod. Factor (R) - Enter the response modification factor R, as defined in Table 5.6, NCh2369 page 41:

Resistant System		R
1.	Structures designed for remain elastic	**1**
2.	Other Structures not included nor similar to those in this list[1].	**2**
7.	Tanks, vessels, stacks, silos and hoppers	
7.1	Stacks, silos and hoppers with continuous down-to-floor shells	**3**
7.2	Silos, hoppers and tanks supported on columns, with or without bracing between columns.	**4**
7.3	Vertical axis steel tanks with continuous down-to-floor shell	**4**
7.4	Vertical axis reinforced concrete tanks with continuous down-to-floor shell	**3**
7.5	Tanks and conduits of composite synthetic material (FRP, GFRP, HDPE and similar materials)	**3**
7.6	Horizontal vessels supported on cradles with ductile anchorages	**4**
8.	Towers, piping and equipment	
8.1	Process towers	**3**
8.2	Cooling towers made of wood or plastic	**4**
8.3	Electric control cabinets resting on floor	**3**
8.4	Steel piping except their connections	**5**
9.	Storage racks	**4**
1. Except that a study proves that an R value other than 2 can be used. Structures whose resistant system is explicitly included in this table are not assimilable to this classification.		

NOTE For more information, see Table 5.5 in NCh2369, page 41.

Coefficient of Importance (I) - Enter the coefficient of importance I, as defined in Section 4.3.1 Classification, NCh2369 page 20:

Category	Description	Coefficient of Importance
C1	Critical structures and equipment based on if its vital, dangerous and essential (such as pressure vessels, tanks, silos).	**1.20**
C2	Normal structures and equipment, which may be affected by normal easily repairable failures and do not hazard other category C1 structures.	**1.00**
C3	Structures include buildings that are essential to post-earthquake recovery or associated with hazardous facilities.	**0.80**

Chile NCh2369 En Español

In English (see "**Chile NCh2369 Seismic Data**" on page 412)

Seccion 4.3.1 Clasificación (NCh2369 pagina 20)

La clasificación apropiada según su importancia es así:

Categoria C1: Estructuras críticas y los equipos utilizados se basaron si son vitales, peligrosos y esenciales. (Recipientes de presión, tanques, silos, etc)

Categoria C2: Las estructuras normales y el equipo utilizado, que pueden ser afectados por los fracasos normalmente y fácilmente reparables y que no arriesguen otras estructuras de categoría C1.

Categoria C3: Estructuras menores o provisionales y el equipo utilizado, cuyo fracaso sísmico no crea peligro a otra estructura de categoría C1 y C2.

Nota: Lea NCh2369 4.3.1 Categorias de Clasificacion para mas informacion

Seccion 4.3.2 Coeficiente de Importancia (NCh2369 pagina 20)

El coeficiente de importancia I para cada categoría tiene los valores siguientes:

C1I : 1.20
C2I : 1.00
C3I : 0.80

Seccion 5. Tablas (NCh2369 pagina 35)

Tabla 5.2 Máxima Aceleración Effectiva (NCh2369 pagina 39)

A_o : aceleración máxima efectiva de suelo

Zona Seismica	A_o
1	0.20 g
2	0.30 g

Tabla 5.4 Parámetros Dependientes del valor de tipo de suelo (NCh2369 pagina 40)

Escoja parámetros dependientes del tipo de suelo en la Tabla 5.3 (NCh2369 pagina 39)

T' : parametro dependiente por el tipo de suelo
n : numero de niveles de parametro dependiente por el tipo de suelo

Tipo de Suelo	T' (s)	n
I	0.20	1.00
II	0.35	1.33
III	0.62	1.80
IV	1.35	1.80

Tabla 5.5 Proporcion de Amortiguamiento (NCh2369 pagina 40)

Sistema Resistente	ξ
Recipiente soldado de acero; silos; depósitos de alimentación; recipientes de presión; torres de proceso; tubería, etc	0.02
Recipiente empernado o remachado de acero;	0.03
Recipientes soldados de acero con o sin refuerzos	0.02
Recipientes de acero con conexiones empernadas, con o sin refuerzos	0.03
Cemento reforzado y estructuras de albañilería	0.05
Cemento reforzado prefabricado, estructuras de gravitación puras.	0.05
Estructuras reforzadas de cemento con conexiones prefabricadas, conectadas a elementos no estructurales e integrados en el modelo estructural	0.05
Estructuras reforzadas de cemento con conexiones prefabricadas, no conectadas a estructuras no elementales	0.03

Estructuras reforzadas de cemento con conexiones secas prefabricadas, no conectadas y conectadas: con conexiones empedernadas y conexiones por medio de barras empotradas en mortero con conexiones soldadas	0.03 0.02
Otras estructuras no incluidas en la lista previa o que asimile a las mencionadas	0.02

Nota: Lea Tabla 5.5 notas finales en NCh2369 pagina 40

Tabla 5.6 Valores máximos del factor de modificación de respuesta (NCh2369 pagina 41)

Sistema Resistente		**R**
1.	Estructuras diseñadas que se quedan elásticas	1
2.	Otras estructuras no incluidas ni semejantes a ésas en esta lista[1].	2
7.	Tanques, recipientes, pilas, silos y depositos de alimentacion	
7.1	Pilas, silos y depositos de alimentacion con continuos recipientes de bajo del piso	3
7.2	Silos, depositos de alimentacion y tankques soportados en columnas, con o sin refuerzo entre columnas.	4
7.3	Tanques de acero con eje vertical con continuo recipiente bajo del piso	4
7.4	Tanques de concreto con eje vertical con continuo recipiente bajo del piso	3
7.5	Tanques y conductos de materia sintética compuesta (FRP, GFRP, HDPE y materiales similares)	3
7.6	Recipientes horizontales apoyados en cunas con anclajes dúctiles	4
8.	Torres, tuberias y equipo	
8.1	Torres de Proceso	3
8.2	Torres de refrigeración de madera o plástico	4
8.3	Gabinetes eléctricos de control que descansan en el piso	3
8.4	Tubería de acero menos sus conexiones	5
9.	Sporte de almacenamiento	4
1. Amenos que un estudio demuestre que el valor de R valga otro que 2 pueden ser utilizados. Estructuras cuyos sistemas resistentes sean incluidos explicitamente en esta tabla no son asimilables a esta clasificacion.		

Nota: Para mas informacion lea Tabla 5.5 en NCh2369 pagina 41

Tabla 5.7 Valores máximos del coeficiente sísmico (NCh2369 pagina 43)

R	Cmax		
	ξ = 0.02	ξ = 0.03	ξ = 0.05
1	0.79	0.68	0.55
2	0.60	0.49	0.42
3	0.40	0.34	0.28
4	0.32	0.27	0.22
5	0.26	0.23	0.18

Nota: Estos valores son validos para la zona sismica 3. Para aplicacion de zona 2 y 1, estos valores seran multiplicados por 0.75 y 0.50, respectivamente.

Seccion 5. Figuras 5.1 (NCh2369 pagina 35)

Figura 5.1 a) Zonificacion Sismica de las Regiones I, II, y III (NCh2369 pagina 44)

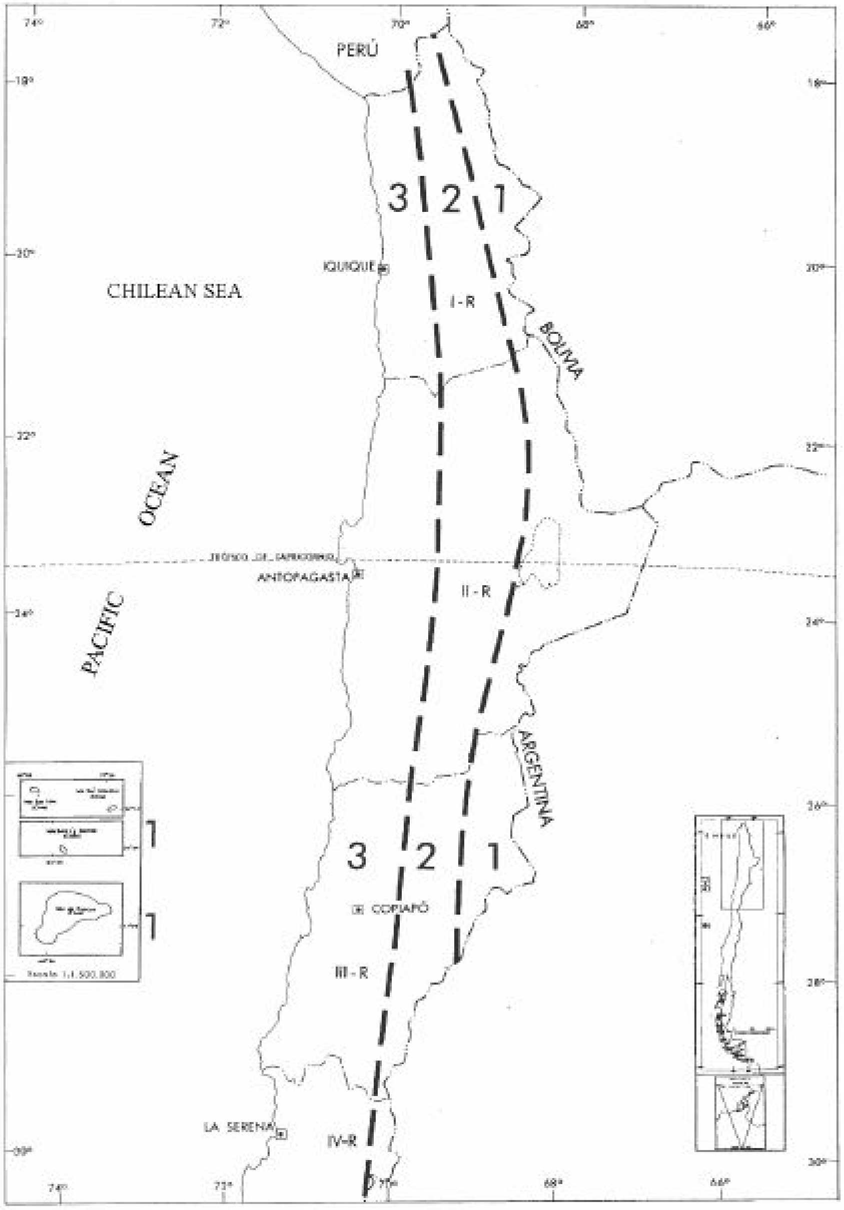


Figura 5.1 b) Zonificacion Sismica de las Regiones IV, V, VI, VII, VIII, IX, X y Region Metropolitana (NCh2369 pagina 45)

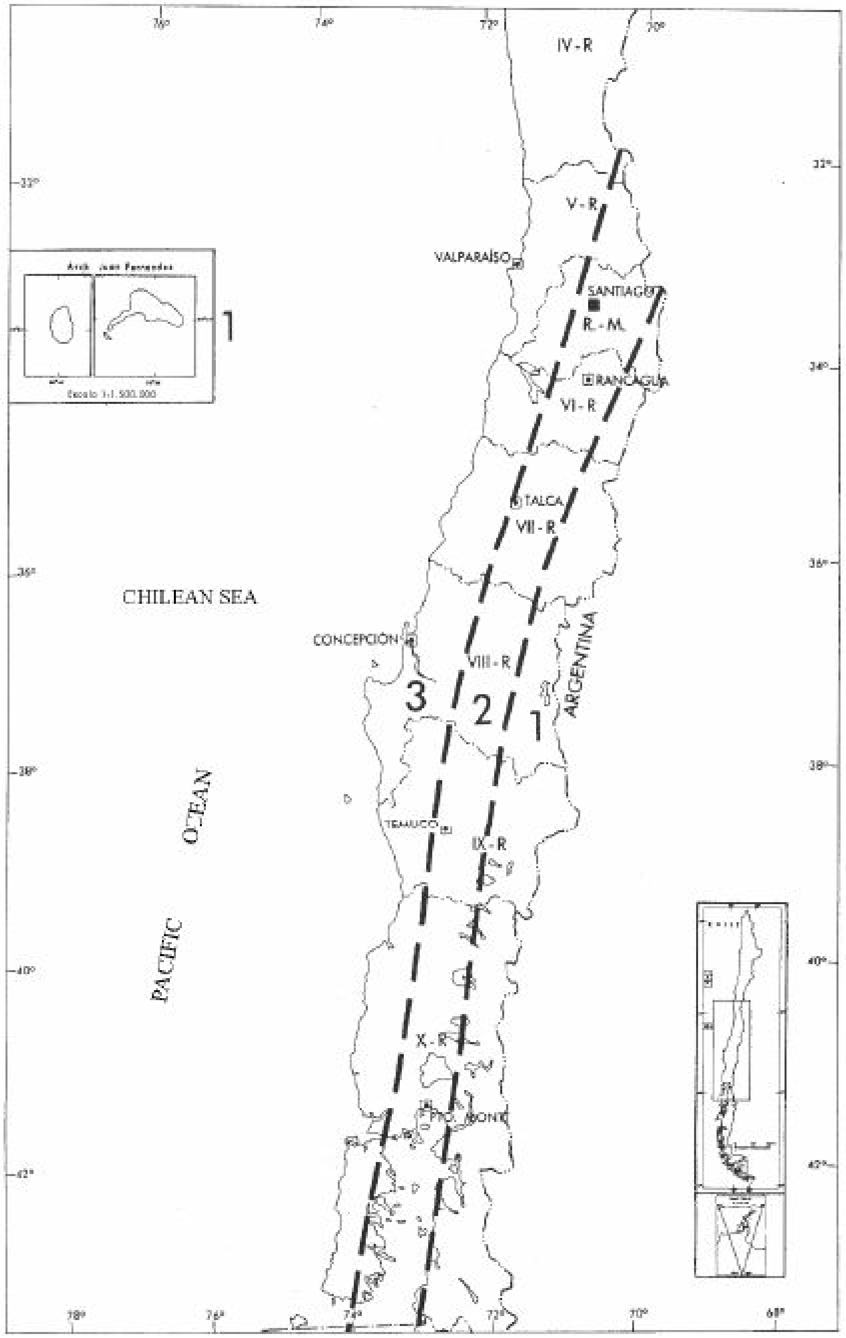
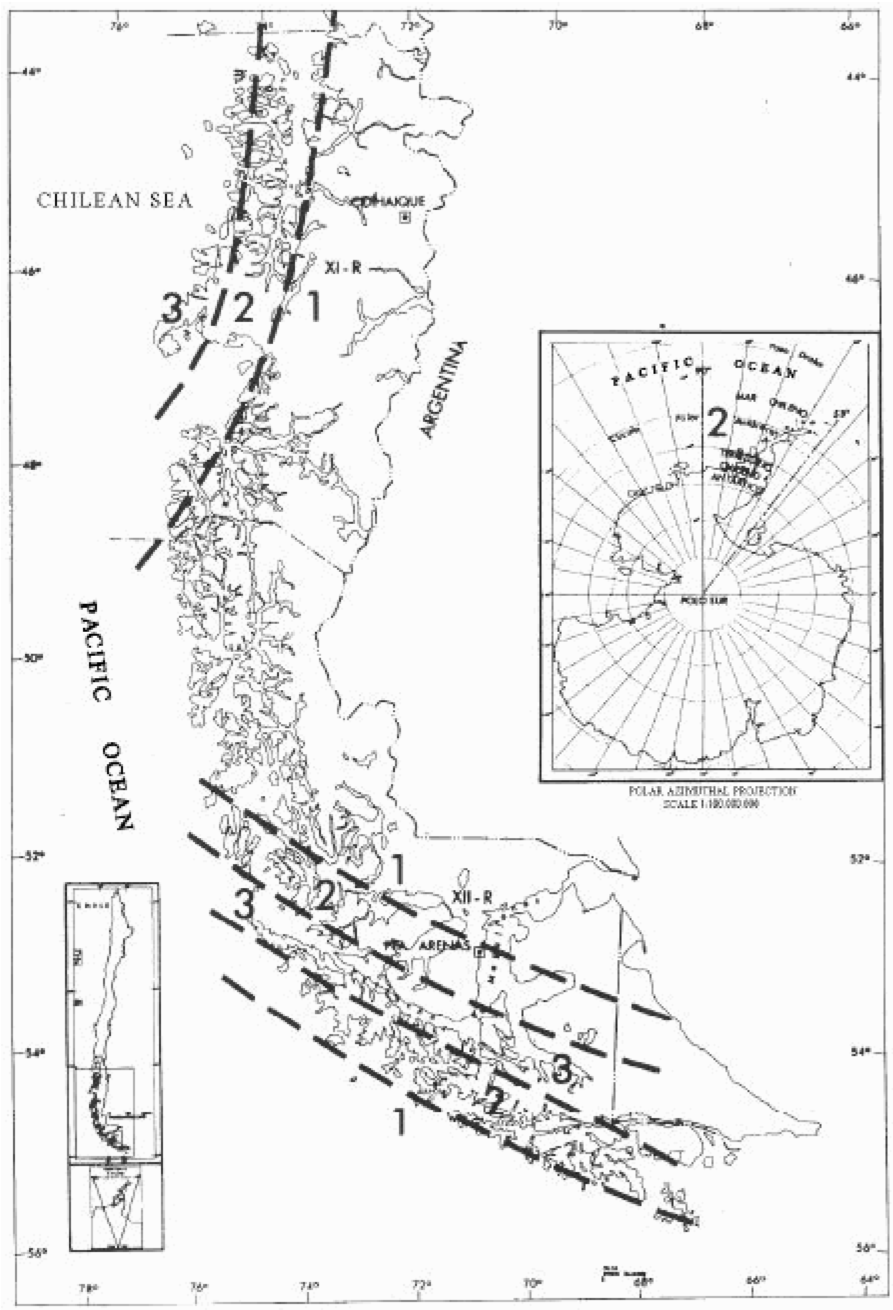


Figura 5.1 c) Zonificacion Sismica de las Regiones XI y XII (NCh2369 pagina 46)



China GB 50011 Seismic Data

Enter data for China's GB 50011-2001 seismic code.

Alpha1 from 5.1.4 - Enter a value for the seismic coefficient, *α.* This value is used to determine F_{Ek} from equation 5.2.1-1. For multistory brick buildings, multistory structures with interior frames, or framed single-story buildings, $\alpha = \alpha_{max}$, the maximum seismic coefficient, as given in Table 5.1.4-1:

Intensity	**VI**	**VII**	**VIII**	**IX**
α_{max}	0.04	0.08 (0.12)	0.16 (0.24)	0.32

DeltaN from 5.2.1 - Enter a value for the additional seismic coefficient, δ_n. This value is used in conjunction F_{Ek} to compute ΔF_n.This is an optional entry for additional seismic action at the top level of the building, as given in Table 5.2.1:

T_g (s)	**$T_1 > 1.4*T_g$**	**$T_1 <= 1.4*T_g$**
<= 0.35	$0.08*T_1 + 0.07$	
> 0.35~0.55	$0.08*T_1 + 0.01$	0.0
> 0.55	$0.08*T_1 - 0.02$	

In addition to calculating lateral loads on elements, the software calcuates vertical loads according to section 5.3.1.

Costa Rica 2002 Seismic Data

The Costa Rica 2002 seismic design code has these options.

Importance Factor [I] - Enter the importance factor in accordance with Table 4.1.

Group	**Description**	**Factor**
A	Buildings and essential facilities such as hospitals, police and fire stations, airport terminals, and emergency generator facilities.	1.5
B	Buildings and hazardous facilities such as buildings containing toxic chemicals or explosives. Buildings whose failure could put other A or B group buildings in danger.	1.5

C	Buildings for educational activities with more than 300 students. Buildings for adult education with more than 500 students. Health centers with 50 or more disabled residents, but not included in Group A. All constructions with occupancy greater than 5000 people not included in Groups A or B. Buildings and equipment in power generation stations, and other public facilities not included in Group A and required to maintain continuous operation.	1.0
D	All work room, office, retail or industry and other non-residential building not specified in Groups A, B, C and E.	1.0
E	Agricultural construction and low occupancy buildings. Sheds and storage buildings of that do not contain toxic material or have low occupancy. Boundary walls and retaining walls that do not represent high risk to bystanders. Temporary facilities for construction.	0.75

Over resistance factor [SR] - Enter the over resistance factor as defined in Chapter 3, paragraph 3 (d). When using static or dynamic analysis methods of Articles 7.4 and 7.5, the over resistance is equal to 2.0 for frame-like structures and dual wall, and equal to 1.2 for cantilever and other structure types. When using Article 7.7 alternative analysis methods, the over resistance is 1.2 on the ultimate capacity calculated in the analysis for all structural systems.

Soil Type - Select the soil type that matches the local soil conditions:

- S1 - A rock-like material, characterized by a velocity shear wave exceeding 760 m/s or other means proper classification. Hard soil conditions or dense, where the depth of soil is less than 50 m.
- S2 - Medium-dense to dense or medium-stiff to stiff whose depth exceeds 50 m.
- S3 - More than 6 m consistency of soft clay to medium stiff cohesive soils or low or medium density. It includes profiles of more than 12 m of soft clay.
- S4 - Soil characterized by a shear wave velocity less than 150 m/s over 12 m of soft clay.

When the properties of site are not known, use S3. Use S4 if the engineer responsible believes it is needed.

Seismic Zone - Select the seismic zone you need, which are shown in Figure 2.1 in the code.

FIGURA 2.1. Zonificación sísmica

Global Ductility [mu] - The overall ductility μ assigned to each of the structural types in Article 4.2, is defined in Table 4.3 according to the structural system classifications section 4.3, and the local ductility of components and joints according to paragraph 4.4.1.

NOTES

- When the system contains structural elements and precast concrete components that are part of seismic resistant systems, overall ductility is 1.5 unless the engineer responsible for the design justifies a higher value by supporting experimental tests and analytical calculations.
- Buildings with severe irregularities, according to paragraph 4.3.4, have overall ductility of 1.0, unless the engineer responsible for the design justifies a higher value. In any case the overall ductility can be assigned greater than the corresponding structures moderate irregularity.
- Structural systems consisting of walls, frames, and frames braced by mezzanines that behave as rigid diaphragms, as subsection (e) of Chapter 3, cannot have a global ductility greater than 1.5.
- For cantilever structures, use a global ductility of 1.0 on the foundation design.
- For steel structures and OCBF and OMF types as defined in the Chapter 10, you must use a global ductility of 1.5.

Peak acceleration [aef] - Enter the design effective peak acceleration expressed as a fraction of gravity. The value is based on the seismic zoning and soil type from Table 2.2.

Soil Type	Zone II	Zone III	Zone IV
S_1	0.20	0.30	0.40
S_2	0.24	0.33	0.40
S_3	0.28	0.36	0.44
S_4	0.34	0.36	0.36

Spectral Dynamic Factor [FED] - Enter the degree-of-freedom change in acceleration a system suffers with respect to the ground acceleration. The area is a function of seismic soil type of the global ductility μ and the period assigned. This value is shown in Figure 5.1 through Figure 5.12 in the code for each seismic zone and soil type.

G Loading Seismic Data

Enter data for seismic loads in G's

Importance Factor - Enter the importance factor. The value is usually between **1.0** and **1.5**. Loads generated form G loading are multiplied directly by this value.

Long. Acceleration (Gx), **Lateral Acceleration (Gz)**, and **Vertical Acceleration (Gy)** - Enter in the value of *G* in each direction for the vessel. Typical values of lateral G loads are from **0** to **0.4**. Vertical G loads can be positive or negative. Negative G loads increase the saddle loads and vertical G loads decrease the saddle loads.

For vertical vessels, the horizontal component used is the maximum of the *Gx* and *Gz* values. The calculated horizontal force is equal to the weight of the element times this maximum G factor. This force times its distance to the support is calculated and summed with all other forces. The *Gy* value is also considered. This value is usually 2/3 of the *Gx* or *Gz* value. Any of these values can be zero.

For horizontal vessels, the lateral (*Gz*) and longitudinal (Gx) directions are considered independently. The vertical load component (*Gy*) acting on the saddle supports is also calculated.

IBC 2000 Seismic Data

Importance Factor I - Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R - For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa - Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	**F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:**				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9][1]**
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analysis shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv - Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	**F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:**				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**

C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analyses shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss - Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl - Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

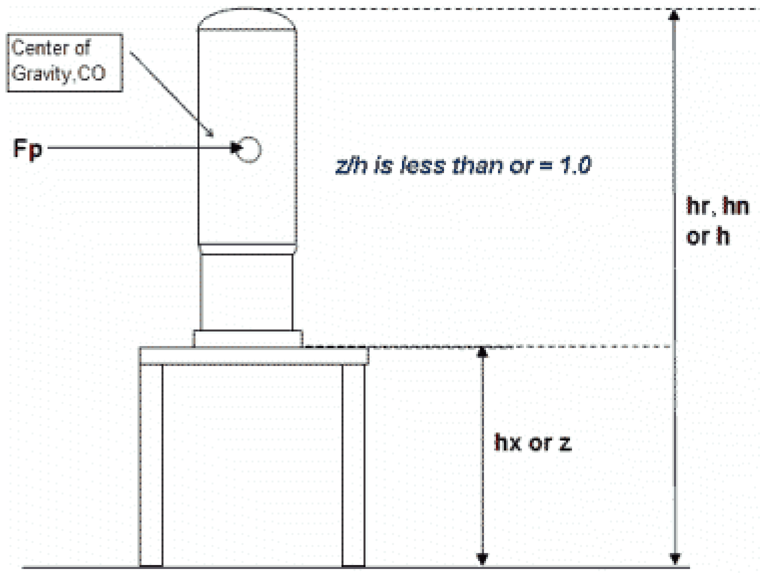
Moment Reduction Factor tau - Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE this value is no longer used and is therefore not applicable.

Site Class - Select the site class: **A**, **B**, **C**, **D**, **E**, or **F.** The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code? - Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h - Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap - Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

IBC 2003 Seismic Data

Importance Factor I - Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R - For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa - Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	**F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:**				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analysis shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv - Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	**F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:**				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]

F	**See note 1**

NOTES

1. Site-specific geotechnical information and dynamic site response analyses shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss - Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl - Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

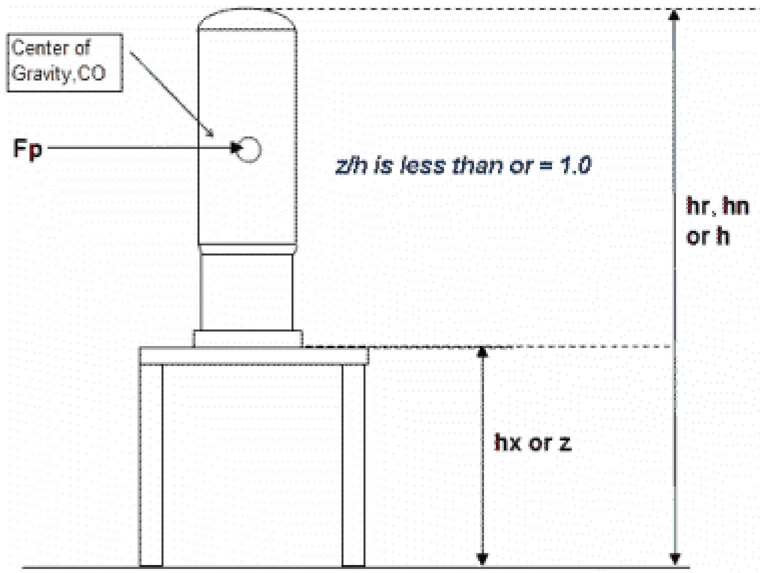
Moment Reduction Factor tau - Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE this value is no longer used and is therefore not applicable.

Site Class - Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code? - Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h - Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap - Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

IBC 2006 Seismic Data

Importance Factor I - Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R - For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa - Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
Site Class	**S_s <= 0.25**	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analysis shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv - Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
Site Class	**S_l <= 0.1**	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analyses shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss - Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl - Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

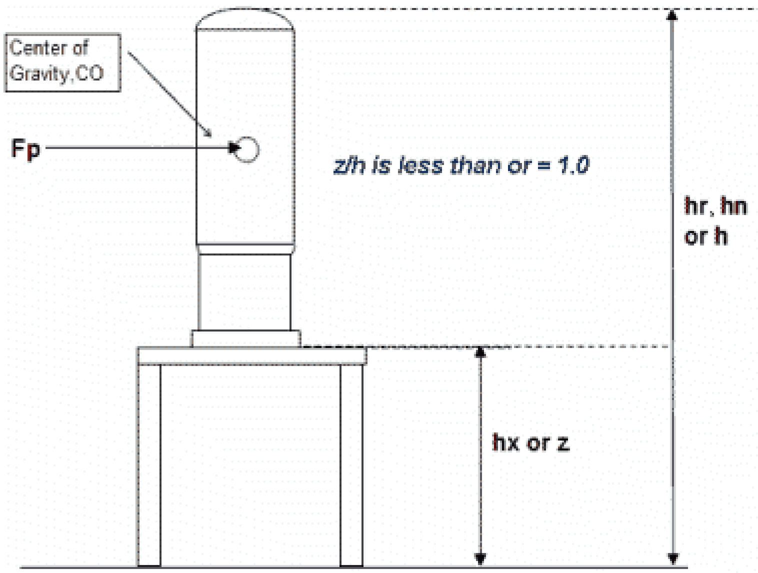
Moment Reduction Factor tau - Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE this value is no longer used and is therefore not applicable.

Site Class - Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code? - Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h - Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap - Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

IBC 2009 Seismic Data

Importance Factor I - Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R - For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa - Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analysis shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv - Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analyses shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss - Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl - Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

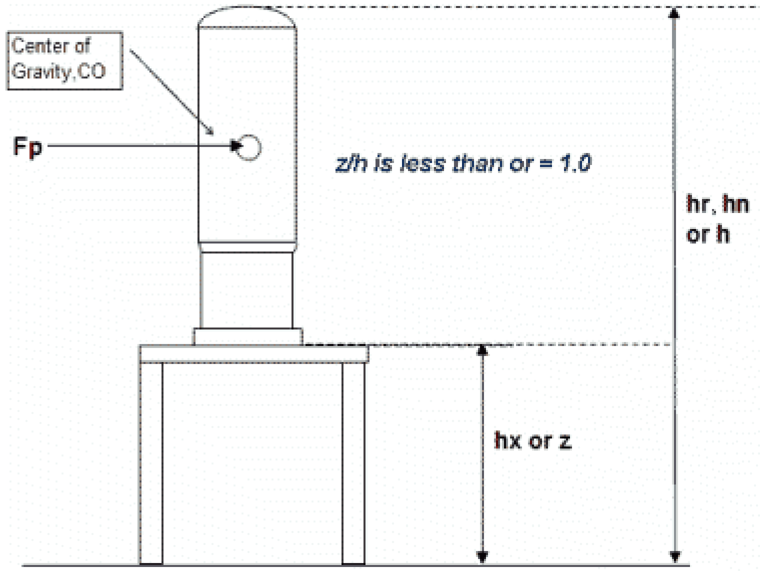
Moment Reduction Factor tau - Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE this value is no longer used and is therefore not applicable.

Site Class - Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code? - Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h - Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap - Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

IS-1893 RSM Seismic Data

Enter data for India's Earthquake Standard IS-1893 RSM.

IS1893 Code Edition - Select the edition of the code to use: **1984**, **2002**, or **2005 Simplified**. The Seismic Coefficient Method has been abandoned in the 2002 edition.

Importance Factor - Enter the importance factor *I,* according to Table 6, IS: 1893, 2002 Seismic Design Code. The following values are for guidance. You may chose an appropriate factor based on economy, strategy and other considerations. You may choose values greater than those listed here.

Structure	**Importance Factor**
Important service and community buildings such as hospitals, schools, monumental structures, emergency buildings like telephone exchange, television stations, radio stations, railway stations, fire station buildings; large assembly halls like cinemas, assembly halls, subway stations, power stations.	**1.5**
All other buildings	**1**

The older version of the IS 1893 also listed the following additional categories:

Structure	**Importance Factor**
Containment structures for atomic power reactors.	**6**
Dams of all types	**3**
Containers of inflammable or poisonous gases/liquids	**2**

Soil Factor - Enter the soil factor (Beta), according to Table 3, IS:1893 seismic design code. Values of the soil factor generally range between 1.0 and 1.5.

Soil Type	**Soil Factor**
Type I rock or hard soils	**1**
Type II medium soils	**1**
Type II medium soils - well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.2**
Type III soft soils	**Between 1.0 and 1.5**
Well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.5**
Combined or isolated RCC footings with tiebeams	**1.2**
Piles not on raft foundations	**1.2**
Other type III soft soils	**1.0**

Zone Number - Enter the zone number. In the 1984 edition of the standard, India is divided into five zones, from **1** to **5**. In the 2002 edition, the number of zones is reduced to four. See the Figure 1 in IS:1893 to obtain the zone number from the map.

Period of Vibration (Optional) - Enter the period of vibration. This is the first period of vibration of the vessel in the normal operating case. This value is used in conjunction with damping to determine *Sa/g*. The software calculates this value, but entering a value here overrides the calculated value. This entry is optional.

Percent Damping - Enter the damping factor of the vessel as a percentage of critical damping. The effect of internal friction, imperfect elasticity of material, slipping, and sliding, results in reducing the amplitude of vibration. This value is used to determine the average acceleration coefficient *Sa/g*.

In the 1984 edition of the IS:1893, Figure 2 shows various curves for damping and natural period of vibration versus Sa/g. Values for damping on this graph range between 2 and 20 percent.

In the 2002 edition of the IS:1893, Figure 2 shows the Sa/g values for 5% damping. If your damping percentage is different from 5% then Sa/g value from Figure 2 has to be multiplied by damping factor. Table 3 provides the damping factor for damping percentage between 0 to 30 percent.

A value outside of the ranges indicated above should not be entered. If a value outside the range is entered, the software simply uses that extreme value of damping. If the value does not have a direct factor provided (such as 2, 5, 10, 20 percentage, etc.) then linear extrapolation will be used to obtain the intermediate values.

Overriding "Fo" value if > 0 - Overrides the **Fo** value calculated by the software. This option defaults to a value of **0**.

Soil Type - Select a soil type: **Rocky or Hard Soils**, **Medium Soils**, or **Soft Soils**.

Force Factor R - Enter the seismic force factor.

Vessel is attached to a building and projects over the roof? - Select if this condition is true.

Optional Sa/g vs. Time Period Input - Click **Review and Edit Spectrum Points** to open the **Spectrum Data Points** dialog box. Enter your values for **Sa/g** at each needed **Time Period (T)**. This table must be completely filled in.

After entering values into the dialog box, select **Do not apply Damping Correction Factor**. The table values are then used in lieu of the internally stored data values of *Sa/g* for various amounts of critical damping. The software does not use any other tables to determine *Sa/g*.

IS-1893 SCM Seismic Data

Enter data for India's Earthquake Standard IS-1893 SCM.

Importance Factor - Enter the importance factor *I,* according to Table 6, IS: 1893, 2002 Seismic Design Code. The following values are for guidance. You may chose an appropriate factor based on economy, strategy and other considerations. You may choose values greater than those listed here.

Structure	**Importance Factor**
Important service and community buildings such as hospitals, schools, monumental structures, emergency buildings like telephone exchange, television stations, radio stations, railway stations, fire station buildings; large assembly halls like cinemas, assembly halls, subway stations, power stations.	**1.5**
All other buildings	**1**

The older version of the IS 1893 also listed the following additional categories:

Structure	**Importance Factor**
Containment structures for atomic power reactors.	**6**

Dams of all types	**3**
Containers of inflammable or poisonous gases/liquids	**2**

Soil Factor - Enter the soil factor (Beta), according to Table 3, IS:1893 seismic design code. Values of the soil factor generally range between 1.0 and 1.5.

Soil Type	**Soil Factor**
Type I rock or hard soils	**1**
Type II medium soils	**1**
Type II medium soils - well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.2**
Type III soft soils	**Between 1.0 and 1.5**
Well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.5**
Combined or isolated RCC footings with tiebeams	**1.2**
Piles not on raft foundations	**1.2**
Other type III soft soils	**1.0**

Zone Number - Enter the zone number. In the 1984 edition of the standard, India is divided into five zones, from **1** to **5**. In the 2002 edition, the number of zones is reduced to four. See the Figure 1 in IS:1893 to obtain the zone number from the map.

User Alpha H - Enter a value for Alpha H.

Mexico Sismo Seismic Data

Mexico Sismo En Español (on page 441)

Enter data for the Mexican seismic code.

Seismic Zone - Select the seismic zone: **A**, **B**, **C,** or **D**. See the region map on page 1.3.29 of the Manual DE Diseno por Sismo (Seismic Design Manual) for Mexico. Zone D is the zone of highest seismic activity while zone A is the least active.

Structure Group - Select the structure group:

- **A - High Safety** - Structure requiring a high degree of safety during their design, such as towers and tanks.
- **B - Med. Safety** - Structure requiring an intermediate degree of safety during their design.
- **C - Low Safety** - Structure requiring a low degree of safety during their design.

Soil Type - Select the type of soil:

- **I** - Hard soil. Ground deposits formed exclusively by layers with propagation velocity b_0 = 700 m/s or modulus of rigidity >= 85000 t/m^2.
- **II** - Medium soil. Ground deposits with fundamental period of vibration and effective velocity of propagation which meets the condition $\beta_c T_s + \beta_s T_c > \beta_c T_c$.
- **III** - Soft soil. Ground deposits with fundamental period of effective vibration of propagation which meet the condition $\beta_c T_s + \beta_s T_c < \beta_c T_c$.

Behavior Factor Q - Enter the behavior factor *Q*. For chimneys and towers, use **3.0**. For smaller structures like tanks, use **2.0**. For more information, see page 1.3.23 in the Manual Sismo.

Effective Absorption Factor - Enter the absorption factor *zeta e hat,* used to compute the increment factor Xi. This factor should always be less than or equal to 0.02.

Orthogonal Increase Factor - Enter the factor to scale up the earthquake loads in a linear (scalar) fashion. This value is traditionally **1.118** and should always be greater than or equal to 1.0. The Mexican Earthquake Code considers an SRSS-type effect on the structure.

Mexico Sismo En Español

In English (see "**Mexico Sismo Seismic Data**" on page 440)

Zona Sismica

Se encuentran 4 opciones aqui. A, B, C y D. Estudie el Manual de Diseño por Sismo para Mexico. En la pagina 1.3.29 del manual, se encuentra un mapa con las diferentes regiones como se demuestra aqui. Se puede observar que la zona D es la zona que tiene la más alta actividad sismica, mientras que la zona A es la menos activa. Seleccione la zona correcta de el menu de opciones.

Grupo Estructural

Grupo A	Grado Alto de Seguridad
Grupo B	Grado Intermedio de Seguridad
Grupo C	Grado Bajo de Seguridad

Torres y tanques son considerados Estructuras del Grupo A porque se requiere un grado alto de seguridad durante su diseño. Estructuras Grupo B requieren un grado intermedio de seguridad y aquellos que son parte del grupo C requieren un grado bajo de seguridad.

Tipo de Terreno

I Terreno Duro:	Depositos del suelo formado exclusivamente por capas con velocidad de propagación b_0 = 700 m/s o el módulo de la rigidez >= 85000 t/m^2.
II Terreno Medio:	Depositos del suelo con el período fundamental de vibración y la velocidad efectiva de propagación que demuestra la condición: $\beta_c T_s + \beta_s T_c > \beta_c T_c$.
III Terreno Suave:	Depositos del suelo con el periodo fundamental de vibración efectiva de propagación que demuestra la condición: $\beta_c T_s + \beta_s T_c < \beta_c T_c$.

Factor de Conducta Q

El factor de conducta Q se encuentra en el Manual pagina 1.3.20. Para chimeneas y torres, este valor debe de ser 3.0. Para estructuras menores como tanques este valor puede ser 2.0. Vea la pagina 1.3.23 en el Manual de Sismo para informacion adicional.

Factor efectivo de Absorción

Este valor es zeta e hat y es utilizado para calcular el factor de incremento Xi. El EAF siempre debe de ser menor que o igual a 0.02.

Factor ortogonal de Aumento

El Codigo Sismico Mexicano considera un efecto SRSS sobre la estructura. Fundamentalmente, este valor aumenta las cargas sismicas en una forma lineal (Escalar). Este valor es traditionalmente 1.118 y siempre debe de ser > o igual a 1.0.

Notas de Analisis:

Al igual que con cada otro análisis de carga de terremoto, el objetivo es de calcular la fuerza cortante en el centro de masa de cada elemento del recipiente. Una vez que las fuerzas cortantes en cada elevación son encontradas, los momentos pueden ser acumulados a la base, apoyo de pierna u oreja.

El análisis empieza calculando las distancias de los pesos y centroides de todos los elementos del recipiente. Es muy importante modelar la estructura en secciones que son apropiadas de tamaño. Para cilindros, este valor está acerca de 10 o 12 pies (3m). Esto asegura que el programa tiene suficiente información para calcular el período natural de vibración con suficiente certeza.

Con los datos de entrada dados y pesos calculados de terremoto y frecuencia natural, PV Elite determina los valores de la tabla 3.1 del Código Sísmico Mexicano. Los valores son:

a_o	Coordenada Espectral para calcular a
c	Coordenada Espectral para calcular a
T_a(s)	Valor de Periodo para calcular a
T_b(s)	Valor de Periodo para calcular a
r	Exponente usado para calcular a

NOTE Para las estructuras del grupo A, los valores de las coordenadas espectrales (a_o, c) obtenidas de la tabla 3.1 son multiplicadas por 1.5.

Para este tipo de recipientes PV Elite utilizara el metodo de analisis statico como esta resumido en el codigo de sismo Mexicano. Si su pila o columna mide más de 60 metros o 192 pies, usted debe utilizar el método dinámico de análisis.

Después de que todos los varios factores son calculados, las fuerzas cortantes en cada nivel son calculadas en cada nivel según la fórmula en la página 1.3.88 del Código de Sismo. Este es el tipo tradicional de ecuaciones masivas de distribución de carga de suma de altura. Además de las fuerzas cortantes en cada nivel, la fuerza primera también es calculada por la ecuación

1.3.89. Después de que las cargas son calculadas, son multiplicadas por el factor ortogonal de aumento.

Con las fuerzas cortantes conocidas, los momentos son calculados de la cima a la base, utilizando tipicas ecuaciones de tipo estática.

NBC 1995 Seismic Data

Enter data for the Canadian NBC seismic code.

Importance Factor - Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from NBC 4.1.9.1 (10):

Classification	**Importance Factor**
Post-disaster buildings.	**1.5**
Schools.	**1.3**
All other buildings. NOTE Most petrochemical structures use this category.	**1.0**

Soil Type - Select a soil type:

- **Soil 1** - Category 1 From rock to stiff fine-grained soils up to 15 m deep.
- **Soil 2** - Category 2. From compact coarse-grained soils to soft fine-grained soils (stiff cohesionless clay) up to 15 m deep.
- **Soil 3** - Category 3. Very loose and loose coarse-grained soils with depth up to 15 m.
- **Soil 4** - Category 4: Very soft and soft fine-grained soils with depth greater than 15 m.

Force Modification Factor - Enter the force modification factor *R*, according to table 4.1.9.B and paragraphs 4.1.9.1 (8) and 4.1.9.3 (3).

R	**Definition**
1.0	Case 18 **-** Elevated tanks (such as equipment on legs), including the special provisions of paragraph 4.1.9.3 (3)
1.5	Case 6 - Ductile structures (such as towers on skirts).

Acceleration Seismic Zone - Select the acceleration seismic zone, according to the city list, Chapter 1 of the supplement to NBC:

Zone	**Example City**
0	Calgary, Alberta

1	Toronto, Ontario
2	Saint John, New Brunswick
3	Varennes, Quebec
4	Vancouver, British Columbia
5	Duncan, British Columbia
6	Port Hardy, British Columbia

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 6 indicates the greatest chance of an earthquake.

Velocity Seismic Zone - Select the velocity seismic zone, according to the city list, Chapter 1 of the supplement to NBC:

Zone	Example City
0	Steinbach, Manitoba
1	Calgary, Alberta
2	Montreal, Quebec
3	Quebec City, Quebec
4	Dawson, Yukon
5	Victoria, British Columbia
6	Destruction Bay, Yukon

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 6 indicates the greatest chance of an earthquake.IS-1893 SCM

NBC 2005 Seismic Data

Enter data for the NBC seismic code.

Importance Factor (IE) - Enter the earthquake importance factor *IE*, a value that accounts for loss of life and property. This value is referred to in Table 4.1.8.5 but is only shown for Ultimate Load State design. The software uses allowable stress design. This value should normally be greater than or equal to **1.0**.

Site Class - Enter the site class, designating the type of underlying strata the vessel is resting on. Site class A generates the least conservative result and site class E generates the most conservative result for the design base shear. The following table is adapted from Table 4.1.8.4.A, NBC 2005.

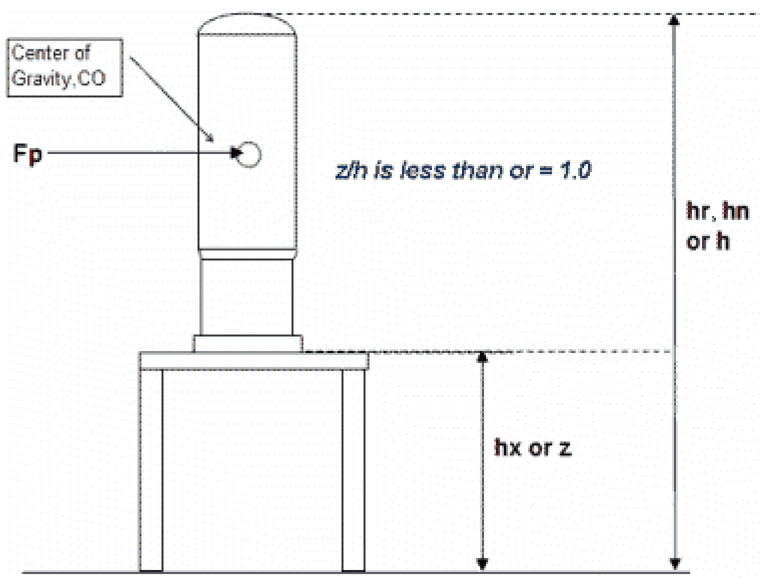
Site Class	Ground Profile Name
A	Hard rock
B	Rock
C	Very dense soil and soft rock
D	Stiff Soil
E	Soft Soil

Overstrength Factor (Ro) - Enter a value for the overstrength factor R_o. A value of **1** is conservative. A value greater than 1 leads to a less conservative, lower design base shear. R_o is taken from NBC table 4.1.8.9. The NBC Code is for buildings and support for vessels is lacking. You will not find a good value for R_o in the tables.

Ductility Factor (Rd) - Enter the ductility factor, R_d, which is the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior. A value of **1** is conservative. A value greater than 1 leads to a less conservative, lower design base shear. R_d is taken from NBC table 4.1.8.9.

Accelerations from Appendix C Division B - Enter acceleration factors S_a for different acceleration periods. The acceleration factors for various cities and regions in Canada are found in Appendix C, Division B, Volume 2 of the NBC Code. These accelerations are used to determine *S(T)* according to paragraph 6, section 4.1.8.4. The software calculates each candidate and performs a final interpolation of the result based on the fundamental period of vibration of the vessel.

Component Elevation Ratio (hx/hn) - Enter a value for the elevation ratio if the vessel is supported by structure, such as a building. The ratio is the height in the structure where the vessel is attached *hx* to height to the top of the building from grade *hn*. Generally, this value is less than or equal to **1**. If the value is zero, the vessel is not building-supported.



Component Amplification Factor (Rp) - Enter the element or component response modification factor R_p, from table 4.1.8.17. For vessels, a value of **2.5** is typical. The larger the value of R_p is, the more conservative the result becomes.

Element or Component Factor (Cp) - Enter the element or component factor C_p, from table 4.1.8.17. A value of **1.0** is typical and is the default value. The larger the value of C_p, the more conservative the result.

Component Force Amp. Factor (Ar) - Enter the element or component force amplification factor *Ar*, from table 4.1.8.17. A value of **2.5** is typical and is the default value. The larger the value of *Ar*, the more conservative the result.

NBC 2010 Seismic Data

Enter data for the NBC seismic code.

Importance Factor (IE) - Enter the earthquake importance factor of the structure, I_E, as described in Table 4.1.8.5.

Importance Category	**Importance Factor, I_E**
Low	0.8
Normal	1.0
High	1.3
Post-disaster	1.5

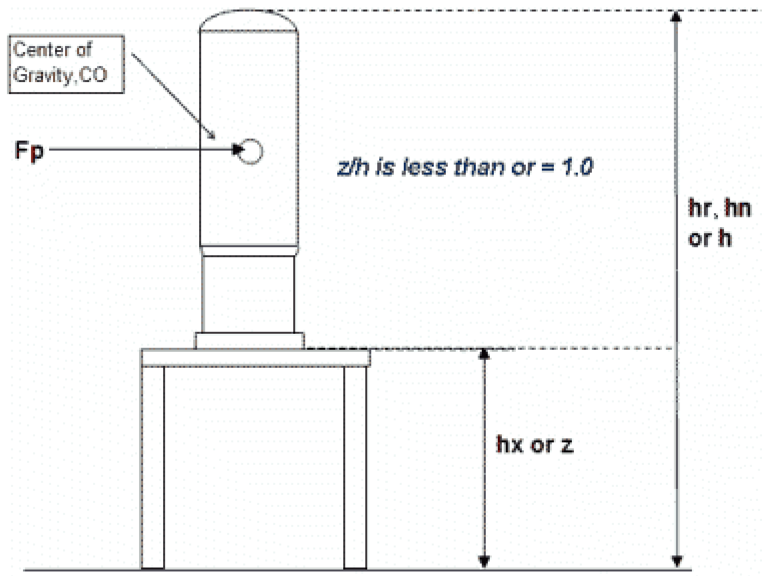
Site Class - Select the site class as defined in Tables 4.1.8.4.A, 4.1.8.4.B, or 4.1.8.4.C.

Overstrength Factor (Ro) - Enter the overstrength factor, R_o, as defined in Table 4.1.8.9.

Ductility Factor (Rd) - Enter the ductility factor, R_d, as defined in Table 4.1.8.9. When an R_d value is required, you must use the corresponding R_o value from the table.

Accelerations from Appendix C Division B - Enter the accelerations that you want to use. The acceleration factors for various cities and regions in Canada are found in Volume 2, Division B, Appendix C.

Component Elevation Ratio (hx/hn) - Enter a value for the elevation ratio, h_x/h_n , when the vessel is supported by structure, such as a building. The ratio is the height in the structure where the vessel is attached, h_x, to the height to the top of the building from grade, h_n. Generally, this value is less than or equal to **1**. If the value is **0**, the vessel is not building-supported.



Component Amplification Factor (Rp) - Enter the component amplification factor, R_p, as defined in Table 4.1.8.18.

Element or Component Factor (Cp) - Enter the seismic coefficient factor, C_p, as defined in Table 4.1.8.18.

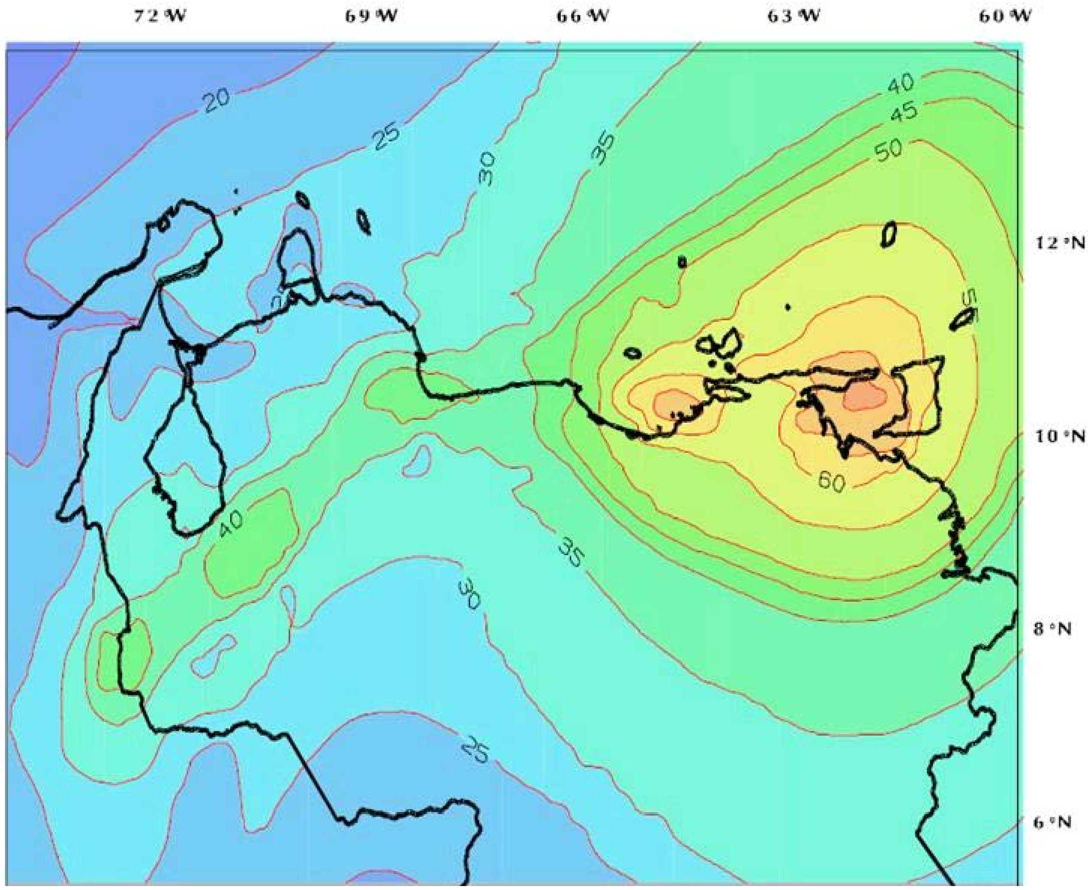
Component Force Amp. Factor (Ar) - Enter the response amplification factor, A_r, as defined in Table 4.1.8.18.

PDVSA Seismic Data

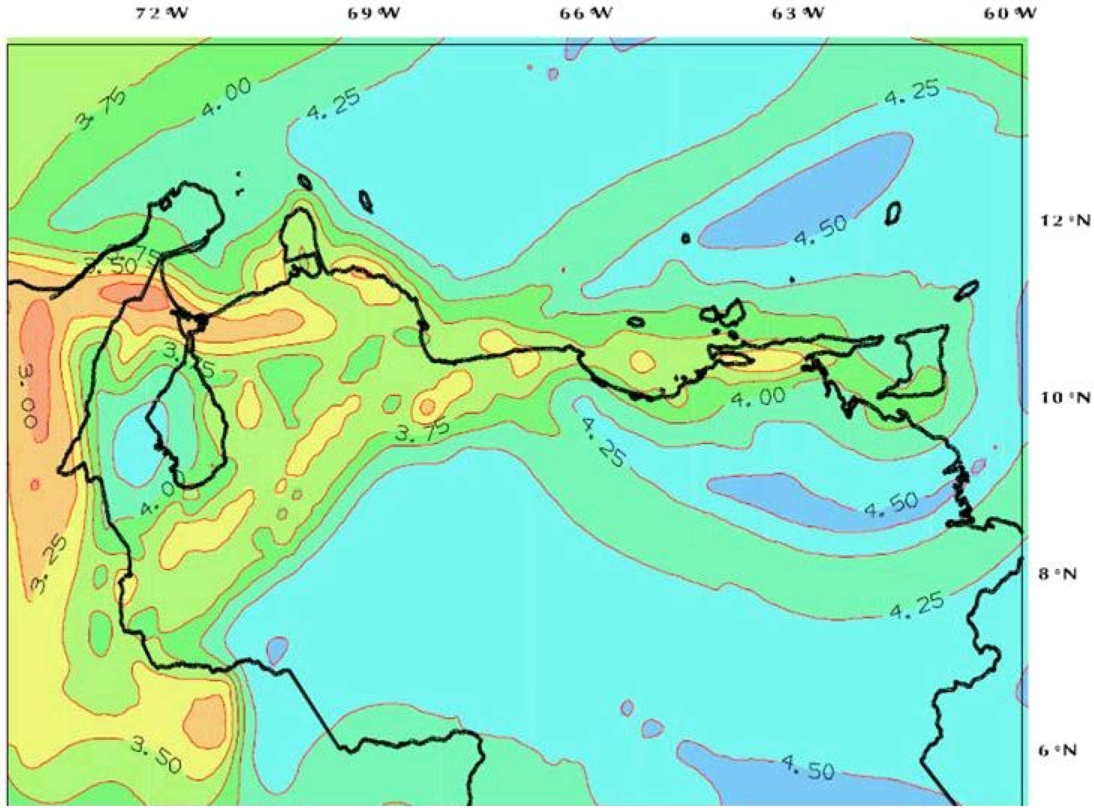
PDVSA En Español (on page 453)

Enter data for the Venezuelan Code for Seismic Analysis of Vessels.

Characteristic Accel. (a*) - Enter the characteristic acceleration *a**. The value is obtained from Figure 6.1 seismic threat map:



Gamma - Enter the gamma factor γ. The value is obtained from Figure 6.2 seismic threat map:



Annual Probability - Enter the risk classification annual probability P_1. Select the risk classification based on the most critical consequences defined in Section 4.1 Reference Scale, JA-221 page 9, Table 4.1 Risk Classification Scale:

Degree of Risk	**Conditions**				**P1 (10^{-3})**
	Number of people exposed	**Economic Loss**		**Environmental impact**	
		Materials	**Lost profits**		
A	Few (< 10)	Limited to installation	Worthless	Little or None	**≤ 2**
B	Important (11 to 100)	The installation and any neighbor	Significant. Between 1 and 50 US$	Recovery ≤ 3 years	**≤ 1**
C	Great number of people (100 to 500)	The installation and many neighbors	Between 50 and 250 US$	Recovery 3 to 10 years	**≤ 0.5**
D	> 500 people	Natural catastrophe	> 250 US$	Irreversible	**≤ 0.1**

NOTE For unknown or doubtful cases of risk, select the highest risk classification.

Damping Factor - Enter the damping factor *ζ*, as defined in Section 6, The Seismic Design Movements, JA-222 page 4, Table 6.1:

Type of Structure	Damping (%)
Welded steel	**3**
Bolted steel	**5**
Reinforced concrete	**5**
Metallic tanks	**See FJ-251**

Ductility Factor - Enter the damping factor *D*, Select *D* for the type of structure that dissipates energy in a way that does not compromise the stability of the vessel system, as defined in Section 8 Ductility Factor, JA-222 page 4, Table 8.1:

Type of Structure	D	Comments
Vertical Vessel (metallic skirt and connection bolts in the foundation)	**2**	The eventual failure of the skirt and connection bolts is ductile. It supplies sufficient deformation to the bolts by means of exposed backups.
Vertical Vessel (metallic skirt and connection bolts in the foundation)	**2**	A ductile failure in the connection bolts on the foundation. Must avoid premature fragile failure in the skirt. It supplies sufficient deformation to the bolts by means of exposed backups.
Vertical Vessel (metallic skirt and connection bolts in the foundation)	**1**	None-ductile failure in the system in the skirt or in the bolts.
Horizontal Vessel (on top of arcs of high hyperstatic moments)	**6**	The mechanical failure in the saddle must be ductile, under the concept of strong column weak beam. No fragile failure between the vessel and the saddle.
Horizontal Vessel (on top of arcs of high hyperstatic moments)	**4**	The mechanical failure in the saddle must be ductile, under the concept of strong column weak beam. No fragile failure between the vessel and the saddle.
Horizontal Vessel (on top of single columns)	**1.5**	Ductile failure on the column.
Horizontal Vessel (on top of a braced arc)	**4**	Ductile failure from the system.

Horizontal Vessel (on top of walls)	**1.5**	In the walls plane.
	2	In the walls vertical plane.
Horizontal Vessel (single diagonal braced columns)	**1.5**	In the non-bracing plane. Ductile failure on the column.
	3	In the bracing plane. It first yields the system of bracing in a ductile form.

Beta Factor - Enter the beta factor *β*, as defined in Section 6.3 Elastic Response Spectrum, Table 6.1 Spectrum Form Values, JA-221 page 15, Table 6.1 Spectrum Form Values:

Spectral Form	**β**	**T^o (s)**	**T* (s)**
S1	2.4	0.10	0.4
S2	2.6	0.20	0.8
S3	2.8	0.30	1.2
S4	3.0	0.40	1.6

Ad - Spectrum acceleration divided by the gravity's acceleration
g - Gravity acceleration
Ao - Maximum acceleration coefficient of the terrain, Ao = a/g
T^o - Value of the period that defines part of the normalized elastic spectral, in seconds
T* - Maximum value of the period in the interval where the normalized elastic spectra have a constant value, in seconds
β - One of the parameters that define the spectral form

T* Factor - Enter the T* factor, as defined in Section 6.3 Elastic Response Spectrum, Table 6.1 Spectrum Form Values, JA-221 page 15, Table 6.1 Spectrum Form Values:

Spectral Form	**β**	**T^o (s)**	**T* (s)**
S1	2.4	0.10	0.4
S2	2.6	0.20	0.8
S3	2.8	0.30	1.2
S4	3.0	0.40	1.6

Ad - Spectrum acceleration divided by the gravity's acceleration
g - Gravity acceleration
Ao - Maximum acceleration coefficient of the terrain, Ao = a/g
T^o - Value of the period that defines part of the normalized elastic spectral, in seconds
T* - Maximum value of the period in the interval where the normalized elastic spectra have a

constant value, in seconds
β - One of the parameters that define the spectral form

T+ Factor - Enter the T^+ factor, the minimum value of the period in the interval where the design spectral has a constant value, as defined in Section 7 Spectrum Design, JA-221 page 18, Table 7.1:

	T^+ (s)
D < 5	0.1 (D-1)
D ≥ 5	0.4
Should comply with $T^o \leq T^+ \leq T^*$	

Phi Factor - Enter the phi factor φ, as defined in Section 5.1 Factor φ and Spectral Form Selection, JA-221 page 11, Table 5.1 Factor φ and Spectral Form Selection:

Material	**V_{sp} (m/s)**	**H (m)**	**Spectral form**	**φ**
Healthy rock / fractured	> 700	Any	S1	**0.85**
Soft rock or moderate	> 400	≤ 50	S1	**0.90**
		> 50	S2	**0.95**
Very hard or very dense floors	> 400	< 30	S1	**0.90**
		30 - 50	S2	**0.95**
		> 50	S3	**1.00**
Hard or dense floors	250 - 400	< 15	S1	**0.90**
		15 - 50	S2	**0.95**
		50 - 70	S3 [2]	**1.00**
		> 70	S4	**1.00**
Firm floors / medium dense	170 - 250	≤ 50	S2 [3]	**1.00**
		> 50	S3 [2]	**1.00**
Soft floors / loose	< 170	≤ 15	S2 [3]	**1.00**
		> 15	S3 [2]	**1.00**
Soft strata interspersed	< 170	< H1	S2	**1.00**

with other more rigid floors [1]		> H1	S3	**0.90**

H - Depth to which the material has a velocity, Vs, greater than 500 m/s.
H1 - Depth from the surface up to the top of the soft strata (m): ≥ 0.25 H.
V_{sp} - Average velocity on the geotechnic profile (m/s).
φ - Correction factor for the horizontal acceleration coefficient.

NOTES

1. The thickness of the strata should be greater than 0.1 H.
2. If $A_o \leq 0.15$, use S4.

If $A_o \leq 0.15$, use S3.

PDVSA En Español

In English (see "**PDVSA Seismic Data**" on page 447)

Seccion 4.1 Escala de Referencia (JA-221 pagina 9)

Seleccionar el Grado de Riesgo asociado con el renglon de consecuencias mas desfavorables descritas en la Tabla 4.1

p1 : Probabilidad anual de excedencia

Tabla 4.1 Escala de Clasificacion de Riesgos

Grado de Riesgo	**CONDICIONES**				**P1 (10-3)**
	Numero de personas expuestas	**Perdidas economicas**		**Impacto Ambiental**	
		Materiales	**Lucro Cesante**		
A	Pocas (< 10)	Limitado a la instalacion	Despreciable	Poco o Nulo	**≤ 2**
B	Importante (11 to 100)	La instalacion y alguna vecina	Significativo Entre 1 y 50 MMUS$	Recuperacion ≤ 3 años	**≤ 1**
C	Elevado numero de personas (100 to 500)	La instalacion y numerosas vecinas	Entre 50 y 250 MMUS$	Recuperacion 3 a 10 años	**≤ 0.5**
D	> 500 personas	De naturaleza catastrofica	> 250 MMUS$	Irreversible	≤ 0.1

Nota: Cuando se presenten dudas en la seleccion del Grado de Riesgo, se adoptara el grado de mayor riesgo

Seccion 6 Movimientos Sismicos de Diseño (JA-222 pagina 4)

ζ : factor de amortiguamiento

Tabla 6.1 Factor de Amortiguamiento

Tipo de Estructura	**Amortiguamiento (%)**
Acero soldado	**3**
Acero empernado	**5**
Concreto reforzado	**5**
Tanques metalicos	**Segun Especificacion FJ-251**

Seccion 8 Factor de Ductilidad (JA-222 pagina 4)

El diseño debe garantizar que el mecanismo de disipacion de energia en el cual se fundamenta D no comprometa la estabilidad del sistema

D : factor de ductilidad

Tabla 8.1 Factores de Ductilidad

Tipo de Esctructura	**D**	**Comentarios**
Recipiente Vertical (falda metalica y pernos de conexion con la fundacion)	**2**	La eventual falla de la falda, asi como de los pernos de conexion, es ductil. Se suministra suficiente longitud de deformacion a los pernos mediante soportes expuestos ("silla para pernos").
Recipiente Vertical (falda metalica y pernos de conexion con la fundacion)	**2**	Falla ductil en los pernos de conexion con la fundacion. Se debe evitar la falla fragil de la falda. Se suministra suficiente longitud de deformacion a los pernos mediante soportes expuestos
Recipiente Vertical (falda metalica y pernos de conexion con la fundacion)	**1**	Falla no ductil del sistema, en la falda y/o pernos. No se recomienda esta situacion
Recipiente Horizontal (sobre porticos de momento de alta hiperestacidad)	**6**	El mecanismo de falla del portico debe de ser ductil, bajo el concepto de columna fuerte viga debil. No se produce falla fragil y/o prematura en la conexion entre el recipiente y el portico
Recipiente Horizontal (sobre porticos de momento de baja hiperestacidad)	**4**	El mecanismo de falla del portico debe de ser ductil, bajo el concepto de columna fuerte viga debil. No se produce falla fragil y/o prematura en la conexion entre el recipiente y el portico
Recipiente Horizontal (sobre monocolumnas)	**1.5**	Falla ductil de la columna

Recipiente Horizontal (sobre portico arriostrado)	**4**	Falla ductil del sistema
Recipiente Horizontal (sobre muros)	**1.5**	En el plano de los muros
	2	En el plano perpendicular a los muros
Recipiente Horizontal (monocolumnas arriostradas con diagonales)	**1.5**	En el plano no arriostrado. Falla ductil de la columna.
	3	En el plano arriostrado. Cede primero el sistema de arriostramiento en forma ductil

Seccion 10 Seleccion del Metodo de Analisis (JA-222 pagina 8)

Tabla 10.1 Seleccion del Metodo de Analisis

Altura de la Estructura (m)	**Ao**	**Grado de Riesgo**			
		A	**B**	**C**	**D**
≤ 10	≤ 0.15	Se	Se	Dy	Dy
	> 0.15	Se	Dy	Dy	Dy
> 10	Any	Dy	Dy	Dy	Dy

Es : Estatico Equivalente
Di: Analisis Dinamico

Nota : El Metodo Estatico Equivalente es el requerimiento minimo para los casos descritos; por tanto, puede ser reemplazado por el Metodo de Analysis Dinamico

Seccion 5.1 Seleccion de la Forma Espectral y del Factor φ (JA-221 pagina 11)

H : Profundidad a la cual se consigue material con velocidad, Vs, mayor que 500 m/s
H1 : Profundidad desde la superficie hasta el tope del estrato blando (m): ≥ 0.25 H
V_{sp} :Velocidad promedio de las ondas de corte en el perfil geotecnico (m/s)
φ :Factor de correccion del coeficiente de aceleracion horizontal

Table 5.1 Forma Espectral Tipificada y Factor φ

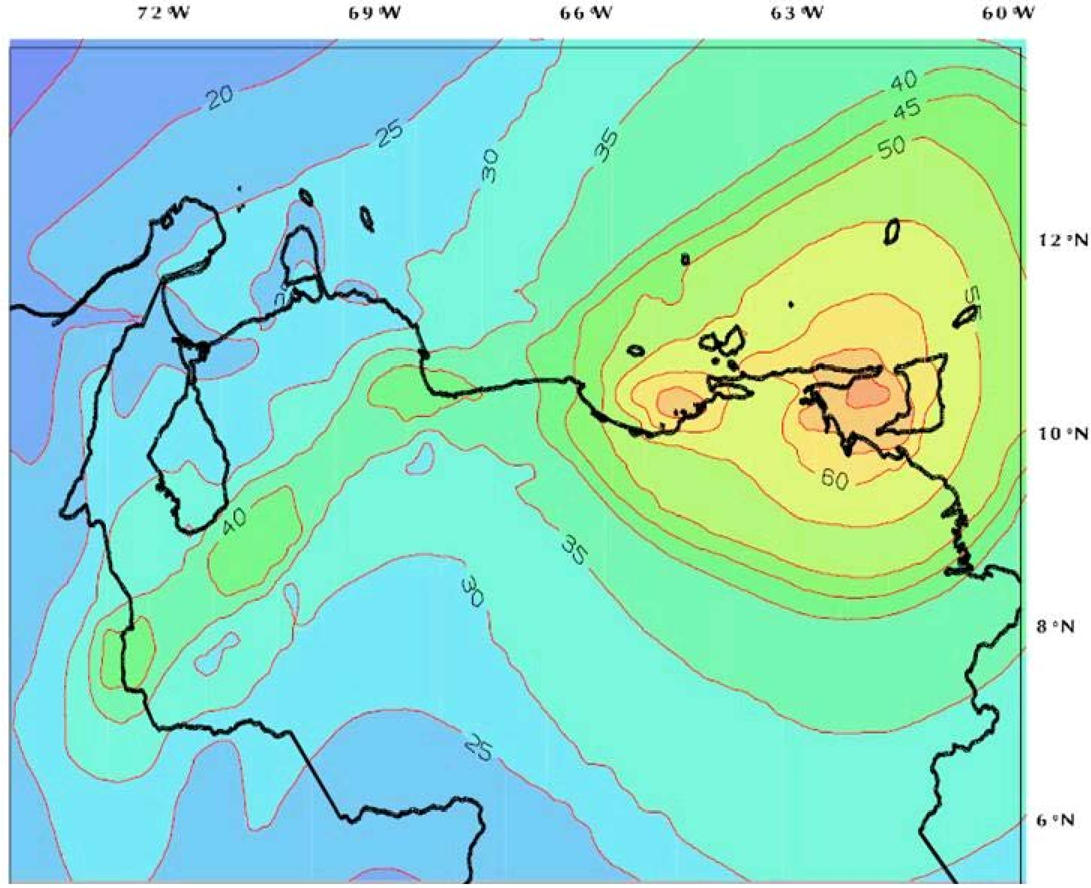
Material	**V_{sp} (m/s)**	**H (m)**	**Forma espectral**	**φ**
Roca sana / fracturada	> 700	Cualquier a	S1	**0.85**
Roca blanda o moderadamente meteorizada	> 400	≤ 50	S1	**0.90**
		> 50	S2	**0.95**
Suelos muy duros o muy densos	> 400	< 30	S1	**0.90**

		30 - 50	S2	**0.95**
		> 50	S3	**1.00**
Suelos duros o densos	250 - 400	< 15	S1	**0.90**
		15 - 50	S2	**0.95**
		50 - 70	S3 (b)	**1.00**
		> 70	S4	**1.00**
Suelos firmes / medio densos	170 - 250	≤ 50	S2 (c)	**1.00**
		> 50	S3 (b)	**1.00**
Suelos blandos / sueltos	< 170	≤ 15	S2 (c)	**1.00**
		> 15	S3 (b)	**1.00**
Estratos blandos intercalados con otros suelos mas rigidos(a)	< 170	< H1	S2	**1.00**
		> H1	S3	**0.90**

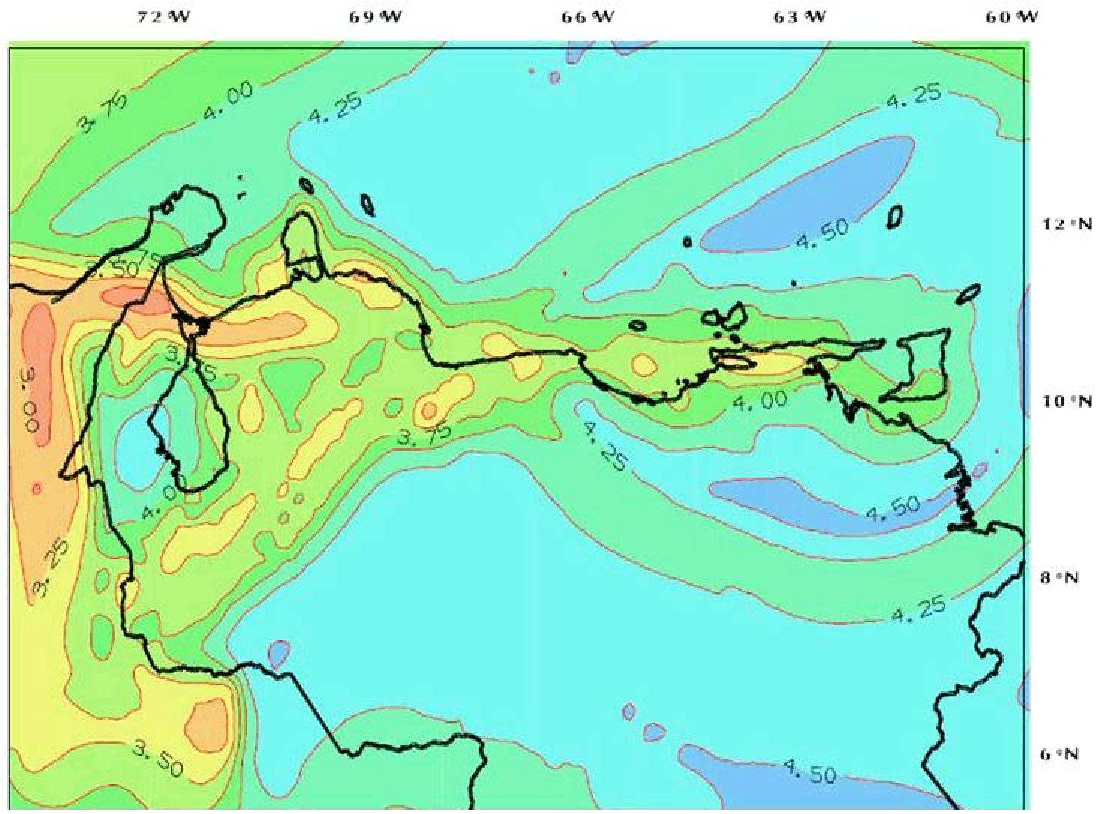
(a) El espesor de los estratos debe ser mayor que 0.1 H
(b) Si $A_o \leq 0.15$, usese S4
(c) Si $A_o \leq 0.15$, usese S3

Seccion 6.1 Mapas de Amenaza Sismica (JA-221 pagina 12)

a*, γ : Valores obtenidos de los Mapas de Amenaza Sismica dados en las Figuras 6.1 y 6.2



Nota: Fig. 6.1 Mapa Amenaza Sismica, valores de a*



Nota: Fig. 6.2 Mapa Amenaza Sismica, valores de γ

Seccion 6.3 Espectro de Respuesta Elastica (JA-221 pagina 15)

Ad : Aceleracion espectral dividida por la aceleracion de gravedad (g)
g : Aceleracion de gravedad
A_o : Coeficiente de aceleracion maxima del terreno, Ao = a/g
T^o : Valor maximo del periodo que define parte del espectro elastico normalizado, en segundos.
T* : Valor maximo del periodo en el intervalo donde los espectros elasticos normalizados tienen un valor constante, en segundos
β : Uno de los parametros que definen la forma de los espectros.

Tabla 6.1 Valores que definen la Forma del Espectro

Forma espectral	β	T^o (s)	T* (s)
S1	2.4	0.10	0.4
S2	2.6	0.20	0.8
S3	2.8	0.30	1.2
S4	3.0	0.40	1.6

Seccion 7 Espectros de Diseño (JA-221 pagina 18)

T^+ : Menor valor del periodo en el intervalo donde los espectros de diseño tienen un valor constante, en segundos.

Tabla 7.1 Valores de T^+ (segundos)

	T^+ (s)
D < 5	0.1 (D-1)
D ≥ 5	0.4
Se debe cumplir $T^o \leq T^+ \leq T^*$	

Res. Spectrum Seismic Data

Select this method to perform a dynamic analysis of the vessel, applying loads based upon the selected seismic response spectrum.

Initially, the vessel is modeled as a two-dimensional structure. An Eigen solution is then performed, which determines system mode shapes and modal natural frequencies. All modes with natural frequencies up through 100 Hz are calculated. The seismic response of each mode is extracted from the response spectrum according to the natural frequency of each mode, and then adjusted according to the mode participation factor. The system response is then determined by combining all of the modal responses.

For tall structures, this analysis gives a much more accurate calculation than the typical static equivalent method. The calculated loads are usually lower in magnitude than those calculated using conventional building code techniques.

Response Spectrum Name - Select the name of the spectra or code to use for the dynamic analysis:

- **User Defined** - When this option is selected, you must select values for **Range Type, Ordinate Type, and Include Missing Mass Components**. You must also click **Edit / Review Spectrum Points** and enter values in the table. The same spectrum is applied in the horizontal and vertical directions.
- **ELCENTRO** - This response spectrum is based on the May 18, 1940 El Centro, California earthquake, North-South component, 5-10% damping, as described in *Introduction to Structural Dynamics* by John Biggs. The same spectrum is applied in the horizontal and vertical directions.
- **ASCE** - Seismic analysis is performed according to the modal analysis procedure of ASCE Standard 7-98. The horizontal spectrum is built according to the ASCE-7 Section 9.4.1.2.6, while the vertical spectrum provides a flat acceleration of 0.2S.
- **IBC** - Seismic analysis is performed according to the modal analysis procedure of the International Building Code 2000, which mirrors ASCE-7. The horizontal spectrum is built according to IBC-2000 Section 1615.1, while the vertical spectrum provides a flat acceleration of 0.2 (according to IBC-2000 Section 1617. 1).

- **1.60D.5** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 0.5% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D2** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 2% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D5** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 5% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D7** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 7% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D10** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 10% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.

Importance Factor - Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property. For ASCE, this is ***I***, the occupancy importance factor according to ASCE-7 Section 9.14. For IBC, this is *Ie*, the occupancy importance factor according to IBC 1616.2.

Shock Scale Factor X | Y dir - Enter values for the shock scale factor in the X and Y directions. The factors are used as multipliers on the horizontal and vertical spectrum data points. The value is usually **1**, but can be higher or lower to scale the spectra up or down. For example, many seismic specifications require that the vertical spectrum be 2/3 the magnitude of the horizontal spectrum. This corresponds to an X value of **1.0** and a Y value of **0.6667**.

In the analysis of vertical vessels, the component in the vertical direction is typically ignored. You can enter a value of **0** in the Y direction field.

Zero Period Acceleration - Enter a value for the ground acceleration *ZPA*. This parameter does double duty, depending upon the analysis type. When used with certain predefined normalized response spectra, it is used as the acceleration factor (in g's) by which the spectrum is scaled. For example, when a spectrum analysis uses a **Response Spectrum Name** beginning with 1.60, such as, **1.60D.5** or **1.60D7**, the software constructs an earthquake spectrum according to the instructions given USNRC Regulatory Guide 1.60. That guide requires that the shape of the response spectrum be chosen from the curves shown in Figures 6-5 and 6-6, based upon the system damping value (such as the ".5" in the spectrum names 1.60H.5). The Reg Guide 1.60 curves are normalized to represent ZPA of one g; the true value is actually site dependent. Therefore, entering a ZPA value here appropriately scales any Regulatory Guide 1.60 curve to meet site requirements.

Combination Method - Enter the spectral summation method mode that most accurately captures the statistical correlation of the responses:

- **SRSS** - Square Root of the Sum of the Squares. This method states that the total system response is equal to the square root of the sum of the squares of the individual modal responses. (This is effectively the same as using the DSRSS method with all correlation coefficients equal to 0.0, or the **Group** method, with none of the modes being closely spaced.) This method is based upon the statistical assumption that all modal responses are completely independent, with the maxima following a relatively uniform distribution throughout the duration of the applied load. This is usually non-conservative, especially if there are any modes with very close frequencies, because those modes will probably experience their maximum DLF at approximately the same time during the load profile.

- **Group** - This method is defined in USNRC Regulatory Guide 1.92. The grouping method attempts to eliminate the drawbacks of the **Absolute** and **SRSS** methods by assuming that modes are completely correlated with any modes with similar (closely spaced) frequencies, and are completely uncorrelated with those modes with widely different frequencies. Effectively, this method dictates that the responses of any modes which have frequencies within 10% of each other first be added together absolutely, with the results of each of these groups then combined with the remaining individual modal results using the **SRSS** method.
- **Absolute** - This method states that the total system response is equal to the sum of the absolute values of the individual modal responses. (This is effectively the same as using the DSRSS method with all correlation coefficients equal to 1.0, or the **Group** method, with all modes being closely spaced.) This method gives the most conservative result, since it assumes that the all maximum modal responses occur at exactly the same time during the course of the applied load. This is usually overly-conservative, because modes with different natural frequencies will probably experience their maximum DLF at different times during the load profile.

Acc. Based Factor Fa - Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
Site Class	**S_s <= 0.25**	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analysis shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the

additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based Factor Fv - Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
Site Class	**S_l <= 0.1**	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4][1]**
F	**See note 1**				

NOTES

1. Site-specific geotechnical information and dynamic site response analyses shall be performed.
2. Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
3. The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss - Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl - Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Response Modification R - For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Coefficient Cd - Enter a value for C_d, the deflection amplitude factor given. For ASCE-7 it is determined from Table 9.5.2.2, while for IBC-2000 it is determined from Table 1617.6. This parameter is used in conjunction with the importance factor and the deflection at level x to calculate the modal deflection at each level. The values in the table range from **2** to **6.5**. The larger value leads to a more conservative result.

Range Type - Select the range type of the abscissa/ordinate axis of the spectrum: **Frequency** or **Period**. Click **Edit / Review Spectrum Points** to enter data points in ascending order.

Ordinate Type - Select the Y-axis ordinate type of the spectrum curve:

- **Displacement** - Diameter
- **Velocity** - Diameter/second
- **Acceleration** - Diameter/second^2 of G's

Include Missing Mass Components - Select to include a correction to the spectrum analysis which represents the contribution of higher order modes not explicitly extracted for the modal/dynamic response, thus providing greater accuracy without additional calculation time. When this option is selected, the software automatically calculates the net (in-phase) contribution of all non-extracted modes and combines it with the modal contributions, avoiding the long calculation time associated with the extraction of the high order modes and the possible excessive conservatism of the summation methods.

During spectrum (either seismic or force spectrum) or time history analyses, the response of a system under a dynamic load is determined by superposition of modal results. One of the advantages of this type of modal analysis is that usually only a limited number of modes are excited and need to be included in the analysis. Only modes with natural frequencies up to about 100 Hz are used. The drawback to this method is that although displacements may be obtained with good accuracy using only a few of the lowest frequency modes, the force, reaction, and stress results may require extraction of far more modes (possibly far into the rigid range) before acceptable accuracy is attained. Select **Include Missing Mass Components** to avoid this drawback.

Edit / Review Spectrum Points - Click to open the **Spectrum Data Points** dialog box. Enter your values for **Acc. G's** at each needed **Period Secs**. Enter data points in ascending order. Interpolation is made linearly for intermediate range values. A zero value is invalid.

SANS 10160-4:2010 Seismic Data

South African National Standard (SANS) seismic data code guidelines detail standards for structural design and seismic recommendations for buildings and industrial structures. More specifically, SANS 10160-4 specifies the seismic zones in which to design buildings that may be subject to earthquake. The standard focuses on preventing major catastrophic structural failures and loss of life, rather than just general building damage.

Reference Horizontal Peak Ground Acceleration - Specifies the horizontal peak ground acceleration .South African seismic zones are determined from a calculation that figures a peak acceleration with a 10 percent probability of occurring within 50 years. For a map of the seismic hazard zones of South Africa, see *SANS 10160-4:2010 Section 5.2, Figure 1*. The SANS 10160-4 equations define the reference horizontal peak ground acceleration factor as a_g for type 1 ground, where *g* refers to earth gravity acceleration.

Ground Type - Specify the ground type. SANS 10160-4 guidelines list the following ground types:

Ground Type	Description
1	Rock or other rock-like geological formation (with at most five meters of weaker surface material).
2	Dense sand, gravel, or very stiff clay deposits (at least several tens of meters in thickness with a gradual increase of mechanical properties with depth).
3	Dense or medium-dense sand, gravel, or stiff clay with deep deposits (with a thickness from several tens to many hundreds of meters).
4	Loose-to-medium cohesion-less soil (with or without soft cohesive layers) or predominately soft-to-firm cohesive soil.

See SANS 10160-4:2010 Section 5.2, Table 1 for more details on ground types and their parameters. Also see Section 5.3, Table 2 for ground type parameter values.

Behavior Factor - Enter the behavior factor (q). The SANS 10160-4:2010 guidelines advise to use a behavior factor in seismic calculations to reduce the elastic response spectrum. This spectrum helps when analyzing the elasticity of a structure. See SANS 10160-4:2010 Section 8.2 and Table 4 for more information.

UBC 1994 Seismic Data

Enter data for the UBC 1994 seismic code.

Importance Factor - Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from Table 23-L of the UBC standard:

Category	Classification	Importance Factor
I	Essential facilities	**1.25**
II	Hazardous facilities	**1.25**
III	Special occupancy structures	**1.0**
IV	Standard occupancy structures	**1.0**

Soil Type - Select a soil type, defined in Table 23-J of the code:

- **Soil 1** - Soil Profile S1. Rock or stiff soil conditions (S Factor = 1.0).
- **Soil 2** - Soil Profile S2. Deep cohesionless deposits or stiff clay conditions (S Factor = 1.2).
- **Soil 3** - Soil Profile S3. Soft to medium-stiff clays and sands (S Factor = 1.5).
- **Soil 4** - Soil Profile S4. More than 40 ft. of soft clay (S Factor = 12.0).

When soil properties are not known, select **Soil 3**.

Horizontal Force Factor - Enter the seismic force factor for non-building structure *Rw,* according to UBC Table 23-Q:

- **3** - Tanks, vessels, or pressurized spheres on braced or unbraced legs.
- **4** - Distributed mass cantilever structures such as stacks, chimneys, silos, and skirt supported vertical vessels.

Seismic Zone - Select the seismic zone, according to UBC-97, Figure 16-2:

Seismic Zone	**Description**	**Seismic Zone Factor Z (Table 16-I)**
0	Gulf and prairies	0.00
1	Rockies and Appalachian areas	0.075
2a	New England, Carolinas, and Ozarks	0.15
2b	Valley area west of the Rockies and the Pacific Northwest	0.20
3	Sierras	0.30
4	California fault areas	0.40

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 5 indicates the greatest chance of an earthquake.

UBC 1997 Seismic Data

Enter data for the UBC 1997 seismic code.

Importance Factor - Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from Table 16-K of the UBC 1997 standard:

Category	Classification	Importance Factor
1	Essential facilities	**1.25**
2	Hazardous facilities	**1.25**
3	Special occupancy structures	**1.0**
4	Standard occupancy structures	**1.0**

Seismic Coefficient Ca - Enter the value of seismic coefficient *CA* according to the project specifications and seismic table 16-Q of UBC 1997. This value is a function of the seismic zone *Z* and the soil profile type.

NOTE In zone 4, *CA* is also a function of near-source factor *Na,* table 16-S of UBC 1997. *Na* is a function of the Seismic Source Type, table 16-U pf UBC 1997.

Table 16-Q

Soil Profile Type	Seismic Coefficient *Ca* for Seismic Zone Factor Z of:				
	Z=0.075	Z=0.15	Z=0.2	Z=0.3	Z=0.4
Sa	**0.06**	**0.12**	**0.16**	**0.24**	**0.32Na**
Sb	**0.08**	**0.15**	**0.20**	**0.30**	**0.40Na**
Sc	**0.09**	**0.18**	**0.24**	**0.33**	**0.40Na**
Sd	**0.12**	**0.22**	**0.28**	**0.36**	**0.44Na**
Se	**0.19**	**0.30**	**0.34**	**0.36**	**0.36Na**
Sf	Site-specific geotechnical investigation and dynamic site response shall be performed to determine seismic coefficients for soil profile Type *Sf*.				

Table 16-S

Seismic Source Type	**Near-Source Factor *Na*[1] for Closest Distance to Known Seismic Source of: [2,3]**		
	<= 2 km	**5 km**	**>=10 km**
A	**1.5**	**1.2**	**1.0**
B	**1.3**	**1.0**	**1.0**
C	**1.0**	**1.0**	**1.0**

Table 16-U

Seismic Source Type[4]	**Seismic Source Description**	**Maximum Moment Magnitude, M**	**Slip Rate, SR (mm/year)**
A	Faults that are capable of producing large Magnitude events and that have a high rate of seismic activity	M>= 7	SR>=5
B	All Faults other than types A and C	M>=7	SR < 5
		M<7	SR>2
		M>=6.5	SR<2
C	Faults that are not capable of producing large magnitude earthquakes and that have a relatively low rate of seismic activity.	M<6.5	SR<=2

NOTES

1. The near-source factor may be based on the linear interpolation of values for distances other than those shown in the table.
2. The location and type of seismic sources to be used for design shall be established on approved geotechnical data (such as the most recent mapping of active faults by the United States Geological Survey or the California Division of Mines and Geology).
3. The closest distance to seismic source shall be taken as the minimum distance between the site and the area described by the vertical projection of the source on the surface (that is, surface projection from fault plane). The surface projection need not include portions of the source at depths of 10 km or greater. The largest value of the near-source factor considering all sources shall be used for design.
4. Subduction sources shall be evaluated on a site specific basis.

Seismic Coefficient Cv - Enter the value of seismic coefficient *CV* according to the project specifications and seismic table 16-R of UBC 1997. This value is a function of the seismic zone *Z* and the soil profile type.

NOTE In zone 4 *CV* is also a function of **Near Source Factor Nv**.

Table 16-R

Soil Profile Type	**Seismic Coefficient *Cv* for Seismic Zone Factor Z of:**				
	Z=0.075	**Z=0.15**	**Z=0.2**	**Z=0.3**	**Z=0.4**
Sa	**0.06**	**0.12**	**0.16**	**0.24**	**0.32Nv**
Sb	**0.08**	**0.15**	**0.20**	**0.30**	**0.40Nv**
Sc	**0.13**	**0.25**	**0.32**	**0.45**	**0.56Nv**
Sd	**0.18**	**0.32**	**0.40**	**0.54**	**0.64Nv**
Se	**0.26**	**0.50**	**0.64**	**0.84**	**0.96Nv**
Sf	Site-specific geotechnical investigation and dynamic site response shall be performed to determine seismic coefficients for soil profile Type *Sf*.				

Near Source Factor Nv - Enter the value of the near source factor *Nv*, table 16-Q of UBC 1997. This factor is only used in UBC Seismic Zone 4. This value ranges from 1 to 2 and is a function of the distance relative to the seismic source.

NOTE *Nv* is a function of the Seismic Source Type, table 16-U pf UBC 1997.

Table 16-T

Seismic Source Type	**Near-Source Factor *Nv*[1] for Closest Distance to Known Seismic Source of:** [2,3]			
	<= 2 km	**5 km**	**10 km**	**>=15 km**
A	**2.0**	**1.6**	**1.2**	**1.0**
B	**1.6**	**1.2**	**1.0**	**1.0**
C	**1.0**	**1.0**	**1.0**	**1.0**

Table 16-U

Seismic Source Type[4]	**Seismic Source Description**	**Maximum Moment Magnitude, M**	**Slip Rate, SR (mm/year)**
A	Faults that are capable of producing large Magnitude events and that have a high rate of seismic activity	M>= 7	SR>=5
B	All Faults other than types A and C	M>=7	SR < 5
		M<7	SR>2
		M>=6.5	SR<2
C	Faults that are not capable of producing large magnitude earthquakes and that have a relatively low rate of seismic activity.	M<6.5	SR<=2

NOTES

1. The near-source factor may be based on the linear interpolation of values for distances other than those shown in the table.
2. The location and type of seismic sources to be used for design shall be established on approved geotechnical data (such as the most recent mapping of active faults by the United States Geological Survey or the California Division of Mines and Geology).
3. The closest distance to seismic source shall be taken as the minimum distance between the site and the area described by the vertical projection of the source on the surface (that is, surface projection from fault plane). The surface projection need not include portions of the source at depths of 10 km or greater. The largest value of the near-source factor considering all sources shall be used for design.
4. Subduction sources shall be evaluated on a site specific basis.

Seismic Zone - Select the seismic zone, according to UBC-91 Figure No. 23-2:

Seismic Zone	**Description**	**Seismic Zone Factor Z (Table 23-I)**
0	Gulf and prairies	0.00
1	Rockies and Appalachian areas	0.075
2a	New England, Carolinas, and Ozarks	0.15
2b	Valley area west of the Rockies and the Pacific Northwest	0.20
3	Sierras	0.30
4	California fault areas	0.40

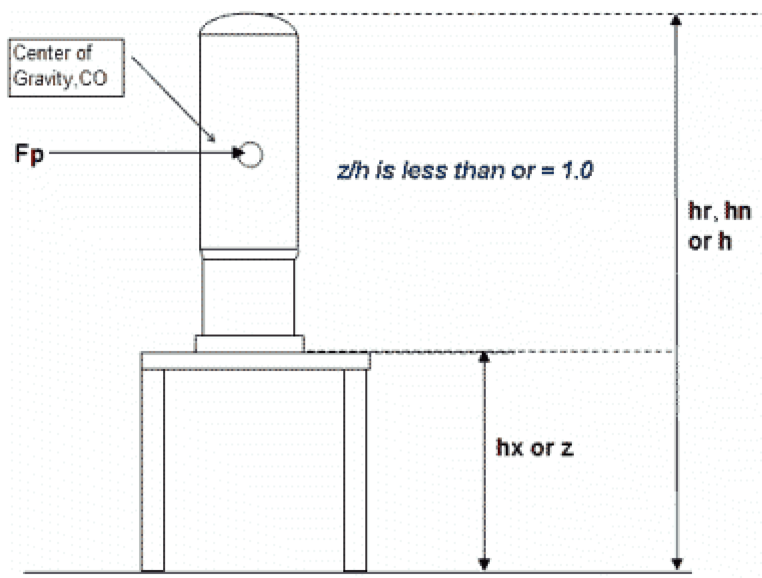
NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 5 indicates the greatest chance of an earthquake.

Force Factor R or Rp - Enter the seismic force factor R according to Table 16-P, UBC 1997. R is defined as the numerical coefficient representative of the inherent overstrength and global ductility of lateral force resisting systems.

- **2.2** - Tanks on braced or unbraced legs.
- **2.9** - Distributed mass cantilever structures such as stacks, chimneys, silos, and skirt supported vertical vessels.

Apply Allowables per 1612.3.2 - Select to use UBC 1997 provisions for earthquake loadings in paragraph 1612.3.2. Paragraph 1612.3.1 does not allow any increase in allowables, but 1612.3.2, the second (alternate) paragraph does. When this option is selected, the software applies an increase in allowable stresses of 1.33 to the skirt and a value of 1.2 to all other elements. **Apply Allowables per 1612.3.2** also overrides **Use Higher Long. Stress** on the **Design Constraints** tab.

Component Elevation Ratio hx/hr - Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap - Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

SECTION 17

PV Elite Analysis

PV Elite performs the analysis of heat exchangers according to the following codes:

- ASME Section VIII Division 1
- TEMA 1998
- PD 5500: 2006
- EN-13445 (U-tubes only at the time of this writing)

ASME Tubesheet (heat exchanger) analysis rules were formerly found in Appendix AA, but in 2003, were re-written and moved to the main body of the code, Part UHX. TEMA and PD 5500 methods of analysis have undergone little changes in recent years.

Formerly, tubesheets could only be analyzed using CodeCalc via **Component Analysis Data** (see "**Component Analysis**" on page 190). While this command is still available, analysis directly in PV Elite has a number of advantages:

- Tubesheets are integrated into a model with cylindrical shells (main shell and channels), heads, and nozzles.
- The total weight of the heat exchanger is calculated, including all of its component parts.
- Supports, such as saddles, are analyzed directly from the integrated model, ensuring that all weights and applied loads are addressed.
- Tubesheet reports are part of the overall analysis.
- For ASME, the MAWP/MAPnc of the entire exchanger is calculated, including tubesheet, tubes, expansion joint, and floating head. These allowable pressures are calculated for each side shell and channel. The hydrotest for each side can also be calculated.
- The tubesheet design code can be changed between TEMA and ASME with little modification to the input data.

Calculating and Displaying Vessel Analysis Results

Each of these steps calculates and displays specific results of a vessel analysis. A brief description of the key analysis steps is defined below:

Step 0: Error Checking

The input program will have already caught most of the errors that are easily made. However, there are some errors that can only be discovered after the analysis begins. There are also some warnings that may be helpful. This first routine check creates a report in the output. If any of the input errors prevents the software from running, execution stops here. Check the output to determine the exact error discovered by the program.

Step 1: Input Echo

PV Elite provides a complete listing of your input. This includes the geometry and materials for each element (head, shell, cone, flange, skirt, etc.) and the information for any details attached to that element.

Step 2: XY Coordinate Calculations

The program calculates the X and Y locations of the first end of every element.

Step 3: Internal Pressure Calculations

The geometry, material, and loading data from your model are used to calculate the required thickness and maximum allowable working pressure for each element (except skirts and flanges). The calculations are done using the ASME Code, Section VIII, Division 1 rules, or the British Standard PD:5500 rules. The internal design pressure at any point is taken to be the given design pressures for that element, plus the pressure due to liquid head, if any.

If you checked the **Increase Thickness For Internal Pressure** design flag and any element is too thin for the given pressure, the program will automatically (or under interactive control) increase the thickness of the element. There is a computation control (under Utilities on the Main Menu) that allows you to increase the element thickness to exactly that required, or to round the thickness up to the next nominal size.

If the program has increased the thickness, it will recalculate all the required thicknesses and maximum allowable working pressures for the vessel, and create a new table showing these results.

After the internal pressure calculation is complete, PV Elite prints the formulas and substitutions, as well the minimum design metal temperatures for the elements.

Step 4: Hydrotest Calculations

The user specifies what kind of hydrotest (and/or the hydrotest pressure) on the global input screens. The program uses this information to calculate the maximum allowed hydrotest pressure and required thickness at the given pressure for each element.

Step 5: External Pressure Calculations

The user explicitly defines two of the three key variables for external pressure calculations: diameter and thickness. The program calculates the third variable, length of section, for the given geometry. Thus if the vessel has two heads and some number of cylindrical elements with no stiffening rings, the program will calculate the design length for each cylinder using the full length of the vessel plus 1/3 the depth of the heads. If there are stiffening rings, the program will calculate an appropriately shorter value.

The program displays the formulas and substitutions for the external pressure calculations on each element. Then the same results are displayed in tabular form.

If the element is not thick enough for the external pressure (and you checked the design boxes in the input) the program will allow you to increase the thickness and/or add stiffening rings (which are created automatically and added to your model). If the thickness is increased the program has to go back to step 3. For rings it repeats this step with the new lengths.

British Standard PD:5500

When performing the PD:5500 external pressure calculations, the program first computes the length of section for the given geometry. The length of section is either the distance between stiffeners, or, if there are no stiffeners, it is the full length of the vessel plus 0.4 times the depth of the heads.

Using the length of section computed, the program first tests to see whether the thickness of the unsupported cylinder (or distance between supports) is satisfactory for the given pressure. A value of Pmax is determined.

If there are stiffeners, then the program performs the calculations described in section 3.6.2.3. The program first performs the computations described in Method A, and then performs the more rigorous calculations described in Method B. For each of these methods (and each value of n), a value of Pn and Fn are obtained.

Pn is the elastic instability pressure of the stiffened cylinder or cone. The value of Pn must not be less than 1.8*Pext in the case of fabricated or hot formed stiffeners and 2.0*Pext in the case of cold formed stiffeners.

Fn is the maximum stress in the stiffener flange divided by the yield stress of the stiffener. A value for Fn is computed for both fabricated or hot formed stiffeners and cold formed stiffeners. These values must be between 0.0 and 1.0.

Step 6: Weight of Elements

Element weights are calculated in both the corroded and uncorroded conditions. Note that for heads the distance given in the input program is taken as the length of the straight flange on the head. This step also calculates the volume of the element.

Step 7: Weight of Details

Each detail has a separate weight calculation. Of note is the fact that partial volumes of liquid in both the heads and the cylinders and in both the horizontal and vertical directions are correctly calculated.

Step 8: ANSI Flange MAWP

If you entered nozzles, you specified the material and class of the attached flanges. PV Elite has the full ANSI flange tables built in, and tells you the rating of the flanges at the operating temperature.

Steps 9 and 10: Total Weight and Detail Moment

Several weight cases are calculated including: empty, operating, and hydrotest. The various detail weights/loads are included in the following cases:

Detail	**Empty**	**Operating**	**Hydrotest**
Saddle	#	#	#
Platform	#	#	#
Packing	#		

Detail	Empty	Operating	Hydrotest
Liquid	#		
Insulation	#	#	#
Lining	#	#	#
Rings	#	#	#
Nozzles	#	#	#
Saddles	#	#	#
Trays	#		
Legs	#	#	#
Lugs	#	#	#
Weights	#	#	#
Forces/Moments		#	

This step also calculates the moment due to individual details, which may not be on the centerline of the vessel. These are usually small. Finally, this step calculates the forces at the support. The vertical force and bending moment (due to detail weights only) are calculated for the 'one support' case (skirts, legs, lugs) and the vertical force at each support is calculated when there is two saddle supports.

NOTE In addition to computing the above weights PV Elite also computes the fabricated weight, shop test weight, shipping weight, erected weight, empty weight and field test weight. The computed weights may or may not include removable or field installed items such as packing and other details. You can specify where these details are to be installed (either shop or field) in the Global Input. Switch to the global input screen and click the **Installation Miscellaneous Options** button located at the top of the screen. By default the program assumes that all details will be installed in the shop and calculate these various weights based on that assumption.

The cumulative weight on the vessel will look drastically different for horizontal vessels on saddle supports than for vertical vessels on skirts, legs, and lugs:

Horizontal cases: Expect the highest weight forces near the saddles, with almost no weight force at the ends or in the middle.

Vertical cases: Expect the weight forces to increase from zero at the top to a maximum at the support. If there are elements below the support, expect the weight force to be negative.

The cumulative moment includes only the moment due to eccentric details, and is usually quite small (except in the case of a large applied moment).

Step 11: Natural Frequency Calculation

PV Elite uses two classical solution methods to determine the first order natural frequencies of vessels. For vertical vessels, the program uses the Freese method, which is commonly used in industry. For horizontal vessels a similar method attributed to Rayleigh and Ritz is used. Each method works by calculating the static deflection of the vessel (for vertical, the vessel as a horizontal cantilever beam). The natural frequency is proportional to the square root of the deflection. PV Elite uses the matrix solution methods (Eigen Solution) to determine the modes of vibration. Horizontal vessels are assumed to be rigid and as such are assigned a frequency of 33 hertz, which is coincident of a ZPA for a rigid structure.

Step 12: Wind Load Calculation

PV Elite uses the rules of ASCE-7, NBC, UBC, and IS-875 to calculate wind loads. Each of these codes uses a basic wind pressure, a function of the velocity squared, along with several surface and site factors to determine the final wind pressure.

Step 13: Earthquake Load Calculation

The five codes used by the software - ASCE-7, UBC, NBC, IS-1893 RSM and IS-1893 SCM each use a static equivalent load to model the earthquake load. Simple site data and loading data are used to determine an expected static equivalent horizontal load on the vessel.

Step 14: Shear and Bending Moments due to Wind and Earthquake

These loadings generate horizontal loads, which are usually fine on a horizontal vessel, but can cause high overturning moments on a vertical vessel. The program calculates the cumulative shear and bending moment on the vessel, for use in later stress calculations.

Step 15: Wind Deflection

PV Elite calculates the deflection at every point in either horizontal or vertical vessels.

Step 16: Longitudinal Stress Constants

As the program prepares to do structural calculations on the vessel, it first calculates the cross sectional area and section modulus of each element in both the corroded and uncorroded condition.

Step 17: Longitudinal Allowable Stresses

There are four allowable stresses in the longitudinal direction for each element: (1) Longitudinal tension based on the basic allowable stress, often multiplied times 1.2 (as specified on the global input), (2) Hydrotest longitudinal tension - 1.5 times the allowable stress new & cold. (3) Longitudinal compression - based on paragraph UG-23 of the Code, and the material's external pressure chart. (4) Hydrotest allowable compression - the basic allowable compression new & cold, multiplied by 1.5.

Step 18: Longitudinal Stresses Due to . . .

Each load (wind, earthquake, weight, pressure) generates a stress. These are calculated individually and displayed by this routine. Note that bending stresses, though only displayed once, are actually positive on one side of the vessel and negative on the other.

Step 19: Stress Due to Combined Loads

In this step the various load cases combinations defined by the user are evaluated. If there are applied forces and moments in the model, then other identifiers such as BS, BN and so forth may appear in the load case definition.

There can be as many as twenty cases, combining pressure loads, weight loads, and moments in various ways. A fairly complete set of load cases is included as a default:

Load Case	**Definition**
1 NP+EW+WI+FW	No pressure + empty weight + wind
2 NP+EW+EQ+FS	No pressure + empty weight + earthquake
3 NP+OW+WI+FW	No pressure + operating weight + wind
4 NP+OW+EQ+FS	No pressure + operating weight + earthquake
5 NP+HW+HI	No pressure + hydrotest weight + hydro wind
6 NP+HW+HE	No pressure + hydrotest weight + hydro earthquake
7 IP+OW+WI+FW	Internal pressure + operating weight + wind
8 IP+OW+EQ+FS	Internal pressure + operating weight + earthquake
9 EP+OW+WI+FW	External pressure + operating weight + wind
10 EP+OW+EQ+FS	External pressure + operating weight + earthquake
11 HP+HW+HI	Hydrotest pressure + hydrotest weight + hydro wind
12 HP+HW+HE	Hydrotest pressure + hydrotest wind + hydro earthquake
13 IP+WE+EW	Internal pressure + wind empty + empty weight
14 IP+WF+CW	Internal pressure + wind filled + empty weight no ca
15 IP+VO+OW	Internal pressure + vortex shedding (OPE) + operating weight
16 IP+VE+OW	Internal pressure + vortex shedding (EMP) + operating weight
17 IP+VF+CW	Internal pressure+ vortex shedding (Filled) + empty weight no ca

The difference between wind loads and hydrotest wind loads is simply a ratio (percentage) defined by the user. This percentage is specified in the Wind Data definition of Global Data - usually about 33% (thus setting the hydrotest wind load at 33% of the operating wind load). Likewise, the hydrotest earthquake load is a percentage of the earthquake load; this percentage is defined in the Seismic Data definition of Global Data.

Some steps that are not applicable for horizontal vessels, such as natural frequency, will not be printed. Also, if a vessel has no supports, then there will be no calculations that involve wind or seismic loads.

Optional Steps

PV Elite includes the following analyses that are performed under specific circumstances:

1. **Cone Evaluation** - Cones are evaluated for internal and external pressure at the large and small ends, and any stiffening rings near the cones are included and evaluated.
2. **Zick Stresses** - Stresses due to saddle supports are evaluated and compared to allowable stresses using the method of L.P. Zick. Note that the stresses are calculated for each saddle, since in PV Elite each saddle can have different loading. Note also that the stresses are not evaluated at the mid span, since the program automatically does that for all the various load case combinations.
3. **AISC Leg Check:** After the software has computed all of the weights, forces and moments, it can then determine the overall state of stress by using the AISC unity check method. The software typically looks at the worst loads on the legs due to wind or seismic in the operating condition and then applies the AISC method of checking the legs. The unity check must be less than or equal to 1.0. Most typical designs fall in the 0.7 - 0.8 range, which is a good check both in terms of economy and safety.
4. **Lug Support Check -** Similar in manner to the **Leg Check** feature, the software gathers the worst loads on the support lugs and then evaluates them according to a set of acceptable standards. In this case, gussets are checked by the **AISC** method and the lug plates are checked by common industry standard methods. These methods are outlined in common pressure design handbooks.
5. **Baserings -** With known forces and moments at the base and the geometry of the basering, PV Elite will analyze or design the basering and gusset geometry.
6. **Flanges -** For main body flanges, the software computes the required thickness of the flange, all relevant stresses, and MAWP for the given geometry. The results seen in the output are based on the input thickness. The software additionally computes the required thickness of the flange. Please note that the software does not include the forces and moments to determine an equivalent design pressure. There are separate fields in the input that can be entered if these effects are to be considered. In order to do this, two runs would have to made. After run 1 was made, the forces and moments on the flange could be entered as needed.
7. **Nozzle Analysis -** Complete nozzle evaluation is incorporated into the software based on the rules in the ASME code. Design cases are made for Internal Pressure, External Pressure and MAPnc. The internal pressure can be based on the MAWP of the entire vessel or the exact pressure at the nozzle location. These options are located in the Global Input section of the input. In addition to perpendicular nozzles, hillside geometries are also considered. Nozzles at any angle can be entered by using the ANG=xx.x command in the nozzle description field. The nozzle analysis also computes MDMT, weld size and strength calculations along with provisions for large nozzles as outlined in appendix 1-7 of the ASME Code. Another description option is for small nozzles. If there is a small nozzle that must have area calculations performed, enter the text "#SN" as part of the nozzle description. By default PV Elite will not calculate small openings for Division 1 vessels per UG-36. If local loads have been defined on the nozzle, the nozzle report will display the results from WRC 107 or PD 5500 Annex G, or WRC 297 whichever one was selected.

8. **Fatigue Analysis** - The fatigue analysis is activated when the number of pressure cycles is specified on the **Design/Analysis Constraints** screen. Click the **Perform Fatigue Analysis** button to display the dialog. Change the number of pressure cycles. This value must be between 1 and 20. This cumulative damage analysis is in accordance with PD:5500 2000 Annex C. In order for this analysis to activate, at least one nozzle must be specified. In the **Nozzle** dialog, there is a check box and a pull down selection menu describing the class of the weld attachment per Annex C. After all of the data is specified, PV Elite produces the **Fatigue Analysis** Report.
9. **Tubesheet Analysis -** When the vessel design Code is ASME VIII or PD 5500, tubesheets are allowed to be defined. They can be attached to flange or cylinder parent elements. PV Elite computes tubesheet required thickness, shell and tube stresses per the rules of TEMA, ASME Part UHX or PD 5500.
10. **Skirt Hole Opening Analysis -** For vertical skirt supported vessels, PV Elite can compute bending and axial stresses due to missing material in skirt openings typically for pipe openings, vents and access openings.
11. **ASME App. EE Analysis -** If you have specified a helical half pipe jacket, this analysis will be performed per ASME Appendix EE.
12. **ASME App. 14 large Central Opening Analysis -** For Welded Flat heads, the analysis of large central opening can be performed per Appendix 14.
13. **Clip Analysis** - If support clips have been entered in, these items are analyzed using industry standard methods.
14. **Lifting Lugs** - Like clips, the stresses in the shell and lugs are calculated at various angles depending on the final orientation of the vessel.
15. **Tubesheet Analysis -** If the model contains tubesheet data, tubesheet analysis per the chosen Code (TEMA, ASME etc) will be listed in the report output.

Nozzle Analysis

PV Elite calculates required wall thickness and area of reinforcement for a nozzle in a pressure vessel shell or head, and compares this area to the area available in the shell, nozzle and optional reinforcing pad. The software also calculates the strength of failure paths for the nozzles. This calculation is based on the ASME Code, Section VIII, Division 1, Paragraph UG-37 through UG-45. The calculation procedure is based on figure UG-37.1.

The software calculates the required thickness (for reinforcement conditions) based on inside diameter for the following vessel components:

Component	**Paragraph**	**Limitations**
Cylinder	UG-27 (c) (1)	None
2:1 Elliptical Head	UG-32 (d) (1)	None
Torispherical Head	UG-32 (e) (1)	None
Spherical Head or Shell	UG-27 (d) (3)	None

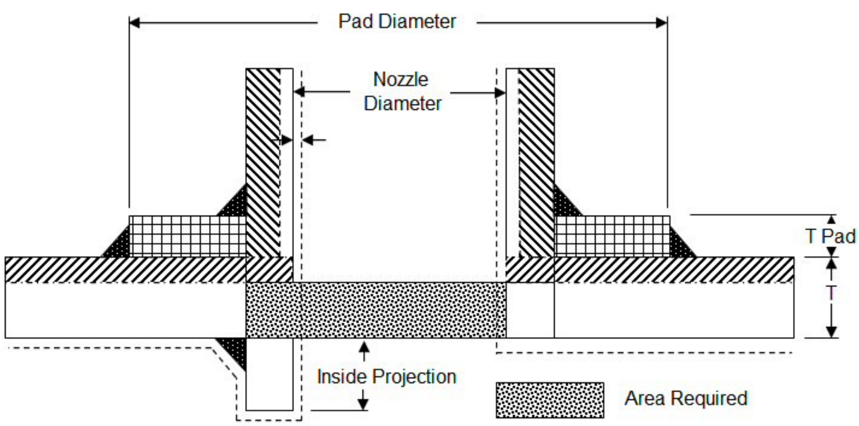
NOTE PV Elite also analyzes a large nozzle in a welded flat head, which is found in this user manual where the flat head is discussed.

The software evaluates nozzles at any angle (less than 90 degrees) away from the perpendicular, allowing evaluation of off angle or hillside nozzles.

Nozzle (on page 58) takes full account of corrosion allowance. You enter actual thickness and corrosion allowance, and the software adjusts thicknesses and diameters when making calculations for the corroded condition. **Nozzle** also performs UCS-66 Minimum Design Metal Temperature (MDMT) calculations for nozzles.

As the nozzle data is entered, PV Elite automatically performs the ASME area of replacement or PD:5500/EN-13445 nozzle compensation calculations. A calculation is performed every time the cursor is moved in between input cells. If there is any error in the input that will not allow the analysis to be performed, a status of failed appears at the bottom of the **Nozzle Input/Analysis** dialog box. The calculation is initiated once the pipe size is specified. If you are changing data, such as the pad thickness and are not moving between cells, press **F5** to force PV Elite to re-calculate and display the results. If the calculation has failed, the result will appear in red. A nozzle that has passed will have blue results. The result is typically the area and minimum nozzle overstress per 1-7. The program displays the text failed in brackets, even though the area of replacement may be sufficient. To effectively use this feature, we suggest that the entire vessel be modeled first, along with the liquid and nozzle pressure design options set. Also, for vessels that have ANSI or DIN flanges note that the flange pressure rating will be shown at the bottom of the nozzle dialog.

The figure below displays the nozzle geometry:



SECTION 18

Output Processor

Output from PV Elite analysis is stored in a binary data file that has the same name as the input file but with a ".T80" extension. Use *Review Reports* (on page 196) to review every report contained in the output from input echo through stress reports.

Each analysis module creates its own report in the output data file. Most of the reports take the form of tables with the rows related to the elements and the columns holding the values such as thickness, MAWP, and stress.

Output Processor Dialog Box

Create Word Document - Create Microsoft Word document of the selected report.

Create ASME Form - Creates an ASME U-1 or U-2 form for the vessel by creating a Microsoft Excel workbook and running a macro that populates worksheet fields with the calculated results from the .pvu analysis results file. This form is copied from the master template form in the PV Elite\System folder.

After the form is opened in Microsoft Excel, the results data can be transferred into the form by clicking **Import Latest Results** (an Excel macro) at the top of the worksheet.

IMPORTANT Because this is an Excel macro, ensure that macros are allowed to load.

If the analysis is run repeatedly, click **Import Latest Results** again. This overlays the latest results into the spreadsheet. If you have typed non-imported material (like the Inspector's name), this data is not overwritten.

IMPORTANT All information in the form must be correct and match the vessel. Please check the data carefully before submitting the form to the National Board or appropriate authorities. Some form information, such as the drawing number, position, and manufacturer is available by clicking **Tools** > **ASME Form Information** and opening the **Additional Vessel Information** dialog box. For more information, see *Enter U-1 Form Information* (on page 226).

Report Up - Select a report and click to move the report up the list.

Report Down - Select a report and click to move the report down the list.

What do you want to do?

- *Customize report header* (on page 482)
- *Customize company name* (on page 482)
- *Customize the title page* (on page 482)
- *Setting default fonts* (on page 483)
- *Save reports to Microsoft Word* (on page 483)

Customize report header

1. Select the **Home** tab.
2. Click **Input** **> Heading** in the **Input / Output** panel.
3. Enter your header information in Line 1, Line 2, and Line 3 boxes.
4. Click **Save**.

   The custom header is inserted the next time you review results.

Customize company name

1. Using Windows Explorer, navigate to the PV Elite System folder. By default, this location is C:\Users\Public\Documents\Intergraph CAS\PVElite\2013\system.
2. Using Notepad or another ASCII file editor, edit company.txt.
3. Put your company name in the first line of the file.
4. Save and exit the file.

   The next time you run an analysis, your company name will appear in the report results header.

NOTE This option is only available if you use SmartPlant License Manager.

Customize the title page

1. Using Windows Explorer, navigate to the PV Elite System folder. By default, this location is C:\Users\Public\Documents\Intergraph CAS\PVElite\2013\system.
2. Using Notepad or another ASCII file editor, edit TITLE.HED.
3. Edit the title page text as needed.
4. Save and exit the file.

   The next time you run an analysis, your custom title page text will appear in the Title Page report.

Setting default fonts

1. Click **Review** on the **Home** tab, **Analyze** panel.
2. At the bottom-left of **Output Processor**, select the **Options** tab.

Save reports to Microsoft Word

1. Click **Review** on the **Home** tab, **Analyze** panel.
2. Select the report or reports to output to Microsoft Word.
3. Click **Word Document** .

SECTION 19

Material Dialog Boxes

The **Material Database** and **Material Properties** dialog boxes are available in many commands throughout the software.

Material Database Dialog Box (on page 485)

Material Properties Dialog Box (on page 534)

Material Database Dialog Box

Displays materials and material properties. Select the needed material. To modify material properties, go to the **Tools** tab and select **Edit/Add Materials**.

Below are examples of standard ASME material names.

Plates and Bolting

- SA-516 55
- SA-516 60
- SA-516 65
- SA-516 70
- SA-193 B7
- SA-182-F1
- SA-182 F1
- SA-182 F11
- SA-182 F12
- SA-182 F22
- SA-105
- SA-36
- SA-106 B

Stainless Steels

- SA-240 304
- SA-240 304L
- SA-240 316
- SA-240 316L
- SA-193 B8

Aluminum

- SB-209
- SB-234

Titanium

- SB-265 1
- SB-265 26H

Nickel

- SB-409
- SB-424

NOTE If you used old CodeCalc material names in previous CodeCalc versions, see the CodeCalc appendix for comparisons with ASME code names.

Material Search String - Enter part of the material name to search against.

Find Next Match - Click to go to the next matching material name available.

UNS # Search String - Enter part of the UNS # to search against.

Select Material - Click to use the selected material.

Cancel - Exit the dialog box without selecting a material.

Material Database Notes

These notes are valid for the 2010 edition of *ASME Section II Part D*. If using an older database, these notes may not be correct or meaningful as they are periodically changed by ASME.

Division 1 Material Notes for Table 1A (Ferrous Materials) - Customary

(a)	The following abbreviations are used: Applic., Applicability; Cond., Condition; Desig., Designation; Smls., Seamless; and Wld., Welded.
(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures.
(c)	For Section VIII and XII applications, stress values in restricted shear such as dowel bolts or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area shall be 0.80 times the value in the above table.
(d)	For Section VIII and XII applications, stress values in bearing shall be 1.60 times the values in the above Table.
(e)	Stress values for –20 to 100F are applicable for colder temperatures when toughness requirements of Section III, Section VIII, or Section XII are met.
(f)	An alternative typeface is used for stress values obtained from time dependent properties (see notes T1 - T11)
(h)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(i)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings.
G1	To these stress values a casting quality factor as specified in PG-25 of Section I or UG-24 of Section VIII, Division 1, or TM-190 of Section XII shall be applied.
G2	These stress values include a joint efficiency factor of 0.60.
G3	These stress values include a joint efficiency factor of 0.85.

G4	For Section I applications, these stresses apply when used for boiler, water wall, superheater, and economizer tubes that are enclosed within a setting. A joint efficiency factor of 0.85 is included in values above 850F.
G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3 % but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors which, when applied to the yield strength values shown in table Y-1, will give allowable stress values that will result in lower values of permanent strain.
G6	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 1500F and shall be considered in the design.
G7	For Section VIII applications, these stress values are based on expected minimum values of 45,000 psi tensile strength and yield strength of 20,000 psi resulting from loss of strength due to thermal treatment required for the glass coating operation. UG-85 does not apply.
G8	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the flexibility of the flange and bolts and corresponding relaxation properties.
G9	For Section III applications, the stress-rupture test is not required for design temperatures 800F and below.
G10	Upon prolonged exposure to temperatures above 800F, the carbide phase of carbon steel may be converted to graphite.
G11	Upon prolonged exposure to temperatures above 875F, the carbide phase of carbon–molybdenum steel may be converted to graphite.
G12	At temperatures above 1000F, these stress values apply only when the carbon is 0.04% or higher on heat analysis.
G13	These stress values at 1050F and above shall be used only when the grain size is ASTM No. 6 or coarser.

G14	These stress values shall be used when the grain size is not determined or is determined to be finer than ASTM No. 6.
G15	For Section I applications, use is limited to stays as defined in PG-13 except as permitted by PG-11.
G16	For Section III Class 3 applications, these S values do not include a casting quality factor. Statically and centrifugally cast products meeting the requirements of NC-2570 shall receive a casting quality factor of 1.00.
G17	For Section III Class 3 applications, statically and centrifugally cast products meeting the requirements of NC-2571(a) and (b), and cast pipe fittings, pumps, and valves with inlet piping connections of 2 in. nominal pipe size and less, shall receive a casting quality factor of 1.00. Other casting quality factors shall be in accordance with the following: a. for visual examination, 0.80; b. for magnetic particle examination 0.85; c. for liquid penetrant examination, 0.85; d. for radiography, 1.00; e. for ultrasonic examination, 1.00; and f. for magnetic particle or liquid penetrant plus ultrasonic examination or radiography, 1.00.
G18	See Table Y-1 for yield strength values as a function of thickness over this range. Allowable stresses are independent of yield strength in this thickness range.
G19	This steel may be expected to develop embrittlement after service at moderately elevated temperature; see Nonmandatory Appendix A, A-207, and A-208.
G20	These stresses are based on weld metal properties.
G21	For Section I, use is limited to PEB-5.3. See PG-5.5 for cautionary note.
G22	For Section I applications, use of external pressure charts for material in the form of barstock is permitted for stiffening rings only.
G23	For temperatures above the maximum temperature shown on the external pressure chart for this material, Fig. CS-2 may be used for the design using this material.
G24	A factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.

G25	For Section III applications, for both Class 2 and Class 3, the completed vessel after final heat treatment shall be examined by the ultrasonic method in accordance with NB-2542 except that angle beam examination in both the circumferential and the axial directions.
G26	Material that conforms to Class 10, 11, or 12 is not permitted.
G27	Material that conforms to Class 11 or 12 is not permitted.
G28	Supplementary Requirement S15 of SA-781, Alternate Tension Test Coupons and Specimen Locations for Castings, is mandatory.
G29	For Section III applications, impact testing in accordance with the requirements of NC-2300 is required for Class 2 components and in accordance with ND-2300 for Class 3 components.
H1	For temperatures above 1000F, these stress values may be used only if the material is solution treated by heating to the minimum temperature specified in the material specification, but not lower than 1900F, and quenching in water or rapidly cooling by other means.
H2	For temperatures above 1000F, these stress values may be used only if the material is heat treated by heating to a minimum temperature of 2000F, and quenching in water or rapidly cooling by other means.
H3	Normalized and tempered.
H4	Solution treated and quenched.
H5	For Section III applications, if heat treatment is performed after forming or fabrication, it shall be performed at 1500–1850F for a period of time not to exceed 10 min at temperature, followed by rapid cooling. For Section VIII applications involving consideration of heat treatment after forming or welding, see table UHA-32 for P-No. 10K, group No.1 materials.
H6	Material shall be solution annealed at 2010F to 2140F, followed by a rapid cooling in water or air.
S1	For Section I applications, stress values at temperatures of 850F and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.

S2	For Section I applications, stress values at temperatures of 900F and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S3	For Section I applications, stress values at temperatures of 1000F and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S4	For Section I applications, stress values at temperatures of 1150F and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S5	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds 3/4 in.
S6	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds 1-1/4 in.
S7	The maximum thickness of unheat-treated forgings shall not exceed 3-3/4 in. The maximum thickness as-heat-treated may be 4 in.
S8	The maximum section thickness shall not exceed 3 in. for double-normalized-and-tempered forgings, or 5 in. for quenched-and-tempered forgings.
S9	Both NPS 8 and larger, and schedule 140 and heavier.
S10	The maximum pipe size shall be NPS 4 (DN 100) and the maximum thickness in any pipe size shall be Schedule 80.
S11	Either NPS 8 and larger and less than schedule 140 wall, or less than NPS 8 and all wall thicknesses.
T1	Allowable stresses for temperatures of 700ºF and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 750ºF and above are values obtained from time-dependent properties.
T3	Allowable stresses for temperatures of 850ºF and above are values obtained from time-dependent properties.
T4	Allowable stresses for temperatures of 900ºF and above are values obtained from time-dependent properties.
T5	Allowable stresses for temperatures of 950ºF and above are values obtained from time-dependent properties.

T6	Allowable stresses for temperatures of 1000ºF and above are values obtained from time-dependent properties.
T7	Allowable stresses for temperatures of 1050ºF and above are values obtained from time-dependent properties.
T8	Allowable stresses for temperatures of 1100ºF and above are values obtained from time-dependent properties.
T9	Allowable stresses for temperatures of 1150ºF and above are values obtained from time-dependent properties.
T10	Allowable stresses for temperatures of 800ºF and above are values obtained from time-dependent properties.
T11	Allowable stresses for temperatures of 650ºF and above are values obtained from time-dependent properties.
W1	Not for welded construction.
W2	Not for welded construction in Section III.
W3	Welded.
W4	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 100 ksi.
W5	Welded, with the tensile strength of the Section IX reduced tension test less than 100 ksi but not less than 95 ksi.
W6	This material may be welded by the resistance technique.
W7	In welded construction for temperatures above 850ºF, the weld metal has a carbon content of greater than 0.05%.
W8	Welding and oxygen or other thermal cutting processes are not permitted when carbon content exceeds 0.35% by heat analysis.
W9	For Section I applications, for pressure retaining welds in 2-1/4Cr–1Mo materials, other than circumferential butt welds less than or equal to 3-1/2 in. in outside diameter, when the design metal temperatures exceed 850ºF, the weld metal shall have a carbon content greater than 0.05%.
W10	For Section III applications, material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted for Class 2 and Class 3 construction when a weld efficiency factor of 1.00 is used in accordance with Note W12.

W11	For Section VIII applications, Section IX, QW-250 Variables QW-404.12, QW-406.3, QW-407.2, and QW-409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Part UF.
W12	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC-2550, shall provide a longitudinal weld efficiency factor of 1.0. Materials welded with filler metal meeting the requirements of NC-2560 shall receive a longitudinal weld efficiency factor of 1.00. Other long. weld efficiency factors shall be in accordance with the following: a. for single butt weld, with filler metal, 0.80; b. for single or double butt weld, without filler metal, 0.85; c. for double butt weld, with filler metal, 0.90; d. for single or double butt weld, with radiography, 1.00.
W13	For Section I applications, electric resistance and autogenous welded tubing may be used with these stresses, provided the following additional restrictions and requirements are met: a. The tubing shall be used for boiler, waterwall, superheater, and economizer tubes that are enclosed within the setting. b. The maximum outside diameter shell be 3.5 in. c. The weld seam of each tube shall be subjected to an angle beam ultrasonic inspection per SA-450. d. A complete volumetric inspection of the entire length of each tube shall be performed in accordance with SA-450. e. Material test reports shall be supplied.
W14	These S values do not include a weld factor. For Section VIII Division 1, and Section XII applications using welds made without filler metal, the tabulated tensile strength values should be multiplied by 0.85. For welds made with filler metal, check UW-12 of Section VIII Division 1 or TW-130.4 for Section XII, as applicable.
W15	The Nondestructive Electric Test requirements of SA-53 Type E pipe are required for all sizes. The pipe shall be additionally marked "NDE" and so noted on the material certification.

Division 1 Material Notes for Table 1A (Ferrous Materials) - Metric

(a)	The following abbreviations are used: Norm. rld., Normalized rolled; Smls., Seamless; Sol. ann., Solution annealed; and Wld., Welded.

(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures are rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained.
(c)	For Section VIII and XII applications, stress values in restricted shear, such as dowel bolts or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area, shall be 0.80 times the values in Division 1 Material Notes for Table 1A (Ferrous Materials) - Customary.
(d)	For Section VIII and XII applications, stress values in bearing shall be 1.60 times the values in Division 1 Material Notes for Table 1A (Ferrous Materials) - Customary.
(e)	Stress values for -30ºC to 40ºC are applicable for colder temperatures when the toughness requirements of Section III, VIII, or XII are met.
(f)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T10).
(g)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 are used when SA-516M Grade 485 is used in construction.
(h)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(i)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings.
G1	To these stress values a casting quality factor as specified in PG-25 of Section I; UG-24 of Section VIII, Division 1; or TM-190 of Section XII is applied.
G2	These stress values include a joint efficiency factor of 0.60.

G3	These stress values include a joint efficiency factor of 0.85.
G4	For Section I applications, these stresses apply when used for boiler, water wall, superheater, and economizer tubes that are enclosed within a setting. A joint efficiency factor of 0.85 is included in values above 450ºC.
G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3 % but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors which, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower values of permanent strain.
G6	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 825ºC and are considered in the design.
G7	For Section VIII applications, these stress values are based on expected minimum values of 310 MPa tensile strength and yield strength of 140 MPa resulting from loss of strength due to thermal treatment required for the glass coating operation. UG-85 does not apply.
G8	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the flexibility of the flange and bolts and corresponding relaxation properties.
G9	For Section III applications, the use of these materials shall be limited to materials for tanks covered in Subsections NC and ND, component supports, and for nonpressure-retaining attachments (NC/ND-2190).
G10	Upon prolonged exposure to temperatures above 425ºC, the carbide phase of carbon steel may be converted to graphite. See Appendix A, A-240.
G11	Upon prolonged exposure to temperatures above 475ºC, the carbide phase of carbon–molybdenum steel may be converted to graphite. See Appendix A, A-240.
G12	At temperatures above 550ºC, these stress values apply only when the carbon is 0.04% or higher on heat analysis.

G13	These stress values at 575ºC and above shall be used only when the grain size is ASTM No. 6 or coarser.
G14	These stress values shall be used when the grain size is not determined or is determined to be finer than ASTM No. 6.
G15	For Section I applications, use is limited to stays as defined in PG-13 except as permitted by PG-11.
G16	For Section III Class 3 applications, these S values do not include a casting quality factor. Statically and centrifugally cast products meeting the requirements of NC-2570 shall receive a casting quality factor of 1.00.
G17	For Section III Class 3 applications, statically and centrifugally cast products meeting the requirements of NC-2571(a) and (b), and cast pipe fittings, pumps, and valves with inlet piping connections of 2 in. nominal pipe size and less, shall receive a casting quality factor of 1.00. Other casting quality factors shall be in accordance with the following ▪ For visual examination, 0.80. ▪ For magnetic particle examination 0.85. ▪ For liquid penetrant examination, 0.85. ▪ For radiography, 1.00. ▪ For ultrasonic examination, 1.00. ▪ For magnetic particle or liquid penetrant plus ultrasonic examination or radiography, 1.00.
G18	See Table Y-1 for yield strength values as a function of thickness over this range. Allowable stresses are independent of yield strength in this thickness range.
G19	The steel may be expected to develop embrittlement after service at a moderately elevated temperature. See Nonmandatory Appendix A, A-207 and A-208.
G20	These stresses are based on weld metal properties.
G21	For Section 1, use is limited to PEB-5.3. See PG-5.5 for cautionary note.
G22	For Section I applications, use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G23	For temperatures above the maximum temperature shown on the external pressure chart for this material, Fig. CS-2 may be used for the design using this material.

G24	A factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.
G25	For Section III applications, for both Class 2 and Class 3, the completed vessel after final heat treatment is examined by the ultrasonic method in accordance with NB-2542 except that angle beam examination in both the circumferential and the axial directions may be performed in lieu of the straight beam examination in the axial direction. The tensile strength does not exceed 860 MPa.
G26	Material that conforms to Class 10, 11, or 12 is not permitted.
G27	Material that conforms to Class 11 or 12 is not permitted.
G28	Supplementary Requirement S15 of SA-781, Alternate Mechanical Test Coupons and Specimen Locations for Castings, is mandatory.
G29	For Section III applications, impact testing in accordance with the requirements of NC-2300 is required for Class 2 components and in accordance with ND-2300 for Class 3 components.
H1	For temperatures above 550ºC, these stress values may be used only if the material is solution treated by heating to the minimum temperature specified in the material specification, but not lower than 1040ºC, and quenching in water or rapidly cooling by other means.
H2	For temperatures above 550ºC, these stress values may be used only if the material is heat treated by heating to a minimum temperature of 1095ºC, and quenching in water or rapidly cooling by other means.
H3	Normalized and tempered.
H4	Solution treated and quenched.
H5	For Section III applications, if heat treatment is performed after forming or fabrication, it shall be performed at 825ºC–1000ºC for a period of time not to exceed 10 min at temperature, followed by rapid cooling.
S1	For Section I applications, stress values at temperatures of 450ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S2	For Section I applications, stress values at temperatures of 475ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.

S3	For Section I applications, stress values at temperatures of 550ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S4	For Section I applications, stress values at temperatures of 625ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S5	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds19 mm.
S6	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds 32 mm.
S7	The maximum thickness of unheat-treated forgings does exceed 95 mm. The maximum thickness as-heat-treated may be 100 mm.
S8	The maximum section thickness does exceed 75 mm for double-normalized-and-tempered forgings, or 125 mm for quenched-and-tempered forgings.
S9	Both DN 200 and larger, and schedule 140 and heavier.
S10	The maximum pipe size is NPS 4 (DN 100) and the maximum thickness in any pipe size is Schedule 80.
S11	Either DN 200 and larger and less than schedule 140 wall, or less than DN 200 and all wall thicknesses.
T1	Allowable stresses for temperatures of 370ºC and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 400ºC and above are values obtained from time-dependent properties.
T3	Allowable stresses for temperatures of 455ºC and above are values obtained from time-dependent properties.
T4	Allowable stresses for temperatures of 480ºC and above are values obtained from time-dependent properties.
T5	Allowable stresses for temperatures of 510ºC and above are values obtained from time-dependent properties.
T6	Allowable stresses for temperatures of 540ºC and above are values obtained from time-dependent properties.

T7	Allowable stresses for temperatures of 565ºC and above are values obtained from time-dependent properties.
T8	Allowable stresses for temperatures of 595ºC and above are values obtained from time-dependent properties.
T9	Allowable stresses for temperatures of 620ºC and above are values obtained from time-dependent properties.
T10	Allowable stresses for temperatures of 425ºC and above are values obtained from time-dependent properties.
T11	Allowable stresses for temperatures of 350ºC and above are values obtained from time-dependent properties.
W1	Not for welded construction.
W2	Not for welded construction in Section III.
W3	Welded.
W4	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 690 MPa.
W5	Welded, with the tensile strength of the Section IX reduced tension test less than 690 MPa but not less than 655 MPa.
W6	This material may be welded by the resistance technique.
W7	In welded construction for temperatures above 450ºC, the weld metal has a carbon content of greater than 0.05%.
W8	Welding and oxygen or other thermal cutting processes are not permitted when carbon content exceeds 0.35% by heat analysis.
W9	For Section I applications, for pressure retaining welds in 2¼Cr–1Mo materials, other than circumferential butt welds less than or equal to 89 mm in outside diameter, when the design metal temperatures exceed 450ºC, the weld metal has a carbon content greater than 0.05%.
W10	For Section III applications, material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted for Class 2 and Class 3 construction when a weld efficiency factor of 1.00 is used in accordance with Note W12.
W11	For Section VIII applications, Section IX, QW-250 Variables QW-404.12, QW-406.3, QW-407.2, and QW-409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Part UF.

W12	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC-2550, shall provide a longitudinal weld efficiency factor of 1.0. Other long. weld efficiency factors shall be in accordance with the following: ▪ For single butt weld, with filler metal, 0.80. ▪ For single or double butt weld, without filler metal, 0.85. ▪ For double butt weld, with filler metal, 0.90. ▪ For single or double butt weld, with radiography, 1.00.
W13	For Section I applications, electric resistance and autogenous welded tubing may be used with these stresses, provided the following additional restrictions and requirements are met: ▪ The tubing is used for boiler, waterwall, superheater, and economizer tubes that are enclosed within the setting. ▪ The maximum outside diameter is 89 mm. ▪ The weld seam of each tube is subjected to an angle beam ultrasonic inspection per SA-450. ▪ A complete volumetric inspection of the entire length of each tube is performed in accordance with SA-450. ▪ Material test reports are supplied.
W14	These S values do not include a weld factor. For Section VIII, Division 1 and Section XII applications using welds made without filler metal, the tabulated tensile strength values should be multiplied by 0.85. For welds made with filler metal, consult UW-12 of Section VIII, Division 1, or TW-130.4 for Section XII, as applicable.
W15	The Nondestructive Electric Test requirements of SA-53 Type E pipe are required for all sizes. The pipe shall be additionally marked "NDE" and so noted on the material certification.

Division 1 Material Notes for Table 1B (Non-Ferrous Materials) - Customary

(a)	The following abbreviations are used: ann., annealed; Applic., Applicability; Cond., Condition; cond., condenser; Desig., Designation; exch., exchanger; extr., extruded; fin., finished; fr., from; rel., relieved; rld., rolled; Smls., Seamless; Sol., Solution; treat., treated; and Wld., Welded.
(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures.

(c)	For Section VIII and XII applications, stress values in restricted shear, such as dowel bolts, rivets, or similar construction in which the shearing is so restricted that the section under consideration would fail without reduction of areas, shall be 0.80 times the values in this table.
(d)	For Section VIII and XII applications, stress values in bearing shall be 1.60 times the values in this Table.
(e)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1-T19).
(f)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 shall be used when SB-407M Grade N08800 is used in construction.
(g)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(h)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings.
G1	For steam at 250 psi (406F), the values given for 400F may be used.
G2	At temperatures over 1000F, these stress values apply only when the carbon is 0.04% or higher.
G3	In the absence of evidence that the casting is of high quality throughout, values not in excess of 80% of those given in the Table shall be used. This is not intended to apply to valves and fittings made to recognized standards.
G4	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 1500F and shall be considered in the design.

G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3 % but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors which, when applied to the yield strength values shown in table Y-1, will give allowable stress values that will result in lower values of permanent strain.
G6	Maximum temperature for external pressure not to exceed 350F.
G7	Use 350F curve for all temperature values below 350F.
G8	The stresses for this material are based on 120 ksi minimum tensile strength because of weld metal strength limitations.
G9	Use Fig. NFC-6 up to and including 300F. Use the 600F curve of Fig. NFC-3 above 300F up to and including 400F. Maximum temperature for external pressure not to exceed 400F.
G10	Maximum temperature for external pressure not to exceed 450F.
G11	Referenced external pressure chart is applicable up to 700F.
G12	Referenced external pressure chart is applicable up to 800F.
G13	For Section VIII and XII applications, use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G14	For Section VIII applications, factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.
G15	To these stress values a quality factor as specified in Section III, ND-3115 or UG-24 of Section VIII, Division 1; or Section XII TM-190 shall be applied for castings. This is not intended to apply to valves and fittings made to recognized standards.
G16	Allowable stress values shown are 90% of those for the corresponding core material.

G17	Copper-silicon alloys are not always suitable when exposed to certain media and high temperatures, particularly steam above 212F. The user should ensure that the alloy selected is satisfactory for the service for which it is to be used.
G18	Because of the occasionally contingent danger from the failure of pressure vessels by stress corrosion cracking, the following is pertinent. These materials are suitable for engineering use under a wide variety of ordinary corrosive conditions with no particular hazard in respect to stress corrosion.
G19	Few alloys are completely immune to stress corrosion cracking in all combinations of stress and corrosive environments and the supplier of the material should be consulted. Reference may also be made to the following sources: (1) Stress Corrosion Cracking Control Measures B.F. Brown, U.S. National Bureau of Standards (1977), available from NACE, Texas; (2) The Stress Corrosion of Metals, H.L. Logan, John Wiley and Sons, New York, 1966.
G20	For plate only.
G21	The maximum operating temperature is arbitrarily set at 500F because harder temper adversely affects design stress in the creep rupture temperature range.
G22	The minimum tensile strength of reduced tension specimens in accordance with QW-462.1 of Section IX shall not be less than 110,000 psi.
G23	This alloy is subject to severe loss of impact strength at room temperature after exposure in the range of 1000F to 1400F.
G24	For stress relieved tempers (T351, T3510, T3511, T451, T4510, T4511, T651, T6510, T6511), stress values for materials in the basic temper shall be used.
G25	The tension test specimen from plate 0.500 in. and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values for thickness less than 0.500 in. shall be used.
G26	The tension test specimen from plate 0.500 in. and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values shown are 90% of those for the core material of the same thickness.
G27	Alloy N06022 in the solution annealed condition is subject to severe loss of impact strength at room temperatures after exposure in the range of 1000F to 1250F.

G28	For external pressure design, the maximum design temperature is limited to 1000F.
G29	External pressure chart NFN-2 may be used for temperatures between 400F and 600F.
H1	For temperatures above 1000F, these stress values may be used only if the material is annealed at a minimum temperature of 1900F and has a carbon content of 0.04% or higher.
H2	For temperatures above 1000F, these stress values may be used only if the material is heat treated by heating it to a minimum temperature of 1900F and quenching in water or rapidly cooling by other means.
H3	For Section I applications, cold drawn pipe or tube shall be annealed at 1900F minimum.
H4	The material shall be given a 1725F to 1825F stabilizing heat treatment.
T1	Allowable stresses for temperatures of 250F and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 300F and above are values obtained from time dependent properties.
T3	Allowable stresses for temperatures of 350F and above are values obtained from time dependent properties.
T4	Allowable stresses for temperatures of 400F and above are values obtained from time dependent properties.
T5	Allowable stresses for temperatures of 500F and above are values obtained from time dependent properties.
T6	Allowable stresses for temperatures of 550F and above are values obtained from time dependent properties.
T7	Allowable stresses for temperatures of 600F and above are values obtained from time dependent properties.
T8	Allowable stresses for temperatures of 750F and above are values obtained from time dependent properties.
T9	Allowable stresses for temperatures of 800F and above are values obtained from time dependent properties.
T10	Allowable stresses for temperatures of 850F and above are values obtained from time dependent properties.

T11	Allowable stresses for temperatures of 900F and above are values obtained from time dependent properties.
T12	Allowable stresses for temperatures of 950F and above are values obtained from time dependent properties.
T13	Allowable stresses for temperatures of 1000F and above are values obtained from time dependent properties.
T14	Allowable stresses for temperatures of 1050F and above are values obtained from time dependent properties.
T15	Allowable stresses for temperatures of 1100F and above are values obtained from time dependent properties.
T16	Allowable stresses for temperatures of 1150F and above are values obtained from time dependent properties.
T17	Allowable stresses for temperatures of 1200F and above are values obtained from time dependent properties.
T18	Allowable stresses for temperatures of 1250F and above are values obtained from time dependent properties.
T19	Allowable stresses for temperatures of 450F and above are values obtained from time dependent properties.
W1	No welding or brazing permitted.
W2	For Section VIII applications, UNF-56(d) shall apply for welded constructions.
W3	For welded and brazed constructions, stress values for O (annealed) temper material shall be used.
W4	The stress values given for this material are not applicable when either welding or thermal cutting is employed.

W5	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC 2550, shall provide a longitudinal weld efficiency factor of 1.00. Materials welded with filler metal meeting the requirements of NC-2560 shall receive a longitudinal weld efficiency factor of 1.00. Other long. weld efficiency factors shall be in accordance with the following: a. for single butt weld, with filler metal, 0.80; b. for single or double butt weld, without filler metal, 0.85; c. for double butt weld, with filler metal, 0.90; d. for single or double butt weld, with radiography, 1.00.
W6	Filler metal shall not be used in the manufacture of welded pipe or tubing.
W7	Strength of reduced-section tensile specimen required to qualify welding procedures. See QW-150, Section IX.
W8	After welding, heat treat at 1150-1200F, hold 1-1/2 hr at temperature for the first inch of cross-section thickness and 1/2 hr for each additional inch, and air cool. For castings used in pumps, valves, and fittings 2 in. nominal pipe size and less, PWHT is not required for socket welds and attachment welds when the castings have been temper annealed at 1150 to 1200F prior to welding.
W9	If welded or brazed, the allowable stress values for the annealed condition shall be used and the minimum tensile strength of the reduced tension specimen in accordance with QW-462.1 of Section IX shall not be less than 30.0 ksi.
W10	When nonferrous materials conforming to specifications in Section II, Part B are used in welded or brazed construction, the maximum allowable working stresses shall not exceed the values given herein for annealed material at the metal temperature shown.
W11	These maximum allowable stress values are to be used in welded or brazed constructions.
W12	These S values do not include a weld factor. For Section VIII, Division 1 applications using welds made without filler metal, the tabulated tensile stress values shall be multiplied by 0.85. For welds made with filler metal, consult UW-12 of Section VIII, Division 1, or TW-130.4 of Section XII, as applicable.

W13	For service at 1200F or higher, the deposited weld metal shall be of the same nominal chemistry as the base metal.
W14	No welding permitted.
W15	For Section VIII and XII applications, no welding is permitted.
W16	Use NFA-12 when welded with 5356 or 5556 filler metal, all thickness, or 4043 or 5554 filler metal, thickness <= 3/8 in. Use NFA-13 when welded with 4043 or 5554 filler metal, thickness > 3/8 in.

Division 1 Material Notes for Table 1B (Non-Ferrous Materials) - Metric

(a)	The following abbreviations are used: ann., annealed; cond., condenser; exch., exchanger; extr., extruded; fin., finished; fr., from; rel., relieved; rld., rolled; Smls., Seamless; Sol., Solution; treat., treated; and Wld., Welded.
(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures are rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated.
(c)	For Section VIII and XII applications, stress values in restricted shear, such as dowel bolts, rivets, or similar construction in which the shearing is so restricted that the section under consideration would fail without reduction of areas, are 0.80 times the values in this Table.
(d)	For Section VIII and XII applications, stress values in bearing are 1.60 times the values in this Table.
(e)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1-T19).
(f)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 are used when SB-407M Grade N08800 is used in construction.
(g)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.

(h)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings.
G1	For steam at 1700 kPa (208ºC), the values given for 200ºC may be used.
G2	At temperatures over 550ºC, these stress values apply only when the carbon is 0.04% or higher.
G3	In the absence of evidence that the casting is of high quality throughout, values not in excess of 80% of those given in the Table are used. This is not intended to apply to valves and fittings made to recognized standards.
G4	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 825ºC and are considered in the design.
G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors which, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower values of permanent strain.
G6	Maximum temperature for external pressure not to exceed 175ºC.
G7	Use 350ºF curve for all temperature values below 175ºC.
G8	The stresses for this material are based on 828 MPa minimum tensile strength because of weld metal strength limitations.
G9	Use Fig. NFC-6 up to and including 150ºC. Use the 315ºC curve of Fig. NFC-3 above 150ºC up to and including 200ºC. Maximum temperature for external pressure not to exceed 200ºC.
G10	Maximum temperature for external pressure does not exceed 225ºC.
G11	Referenced external pressure chart is applicable up to 375ºC.

G12	Referenced external pressure chart is applicable up to 425ºC.
G13	For Section VIII and XII applications, use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G14	For Section VIII applications, a factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.
G15	To these stress values a quality factor as specified in ND-3115 of Section III; UG-24 of Section VIII, Division 1; or TM-190 of Section XII shall be applied for castings. This is not intended to apply to valves and fittings made to recognized standards.
G16	Allowable stress values shown are 90% of those for the corresponding core material.
G17	Copper-silicon alloys are not always suitable when exposed to certain media and high temperatures, particularly steam above 100ºC. The user should ensure that the alloy selected is satisfactory for the service for which it is to be used.
G18	Because of the occasionally contingent danger from the failure of pressure vessels by stress corrosion cracking, the following is pertinent. These materials are suitable for engineering use under a wide variety of ordinary corrosive conditions with no particular hazard in respect to stress corrosion.
G19	Few alloys are completely immune to stress corrosion cracking in all combinations of stress and corrosive environments and the supplier of the material should be consulted. Reference may also be made to the following sources: ▪ Stress Corrosion Cracking Control Measures B.F. Brown, U.S. National Bureau of Standards (1977), available from NACE, Texas ▪ The Stress Corrosion of Metals, H.L. Logan, John Wiley and Sons, New York, 1966.
G20	For plate only.
G21	The maximum operating temperature is arbitrarily set at 250ºC because harder temper adversely affects design stress in the creep rupture temperature range.
G22	The minimum tensile strength of reduced tension specimens in accordance with QW-462.1 of Section IX is not less than 760 MPa.

G23	This alloy is subject to severe loss of impact strength at room temperature after exposure in the range of 550ºC to 750ºC..
G24	For stress relieved tempers (T351, T3510, T3511, T451, T4510, T4511, T651, T6510, T6511), stress values for materials in the basic temper are used.
G25	The tension test specimen from plate 13 mm and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values for thickness less than 13 mm are used.
G26	The tension test specimen from plate 13 mm and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values shown are 90% of those for the core material of the same thickness.
G27	Alloy N06022 in the solution annealed condition is subject to severe loss of impact strength at room temperatures after exposure in the range of 550ºC to 675ºC.
G28	For external pressure design, the maximum design temperature is limited to 550ºC.
G29	The maximum allowable stress values for greater than 900ºC are 9.7 MPa (927ºC), 7.6 MPa (954ºC), and 5.0 MPa (982ºC).
G30	The maximum allowable stress values for greater than 900ºC are 5.0 MPa (925ºC), 4.0 MPa (950ºC), 3.2 MPa (975ºC), and 2.6 MPa (1000ºC). The maximum use temperature is 982ºC; the value listed at 1000ºC is provided for interpolation purposes only.
G31	The maximum allowable stress values for greater than 900°C are 7.8 MPa (925°C), 5.2 MPa (950°C), 3.5 MPa (975°C), and 2.4 MPa (1000°C). The maximum use temperature is 982°C; the value listed at 1000°C is provided for interpolation purposes only.
G32	The maximum allowable stress values for greater than 900°C are 6.6 MPa (925°C), 4.4 MPa (950°C), 2.9 MPa (975°C), and 2.0 MPa (1000°C). The maximum use temperature is 982°C; the value listed at 1000°C is provided for interpolation purposes only.
G33	External pressure chart NFN-2 may be used for temperatures between 205°C and 315°C.
H1	For temperatures above 550ºC, these stress values may be used only if the material is annealed at a minimum temperature of 1040ºC and has a carbon content of 0.04% or higher.

H2	For temperatures above 550ºC, these stress values may be used only if the material is heat treated by heating it to a minimum temperature of 1040ºC and quenching in water or rapidly cooling by other means.
H3	For Section I applications, cold drawn pipe or tube is annealed at 1038ºC minimum.
H4	The material is given a 940ºC to 995ºC stabilizing heat treatment.
T1	Allowable stresses for temperatures of 125ºC and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 150ºC and above are values obtained from time dependent properties.
T3	Allowable stresses for temperatures of 175ºC and above are values obtained from time dependent properties.
T4	Allowable stresses for temperatures of 205ºC and above are values obtained from time dependent properties.
T5	Allowable stresses for temperatures of 260ºC and above are values obtained from time dependent properties.
T6	Allowable stresses for temperatures of 290ºC and above are values obtained from time dependent properties.
T7	Allowable stresses for temperatures of 315ºC and above are values obtained from time dependent properties.
T8	Allowable stresses for temperatures of 400ºC and above are values obtained from time dependent properties.
T9	Allowable stresses for temperatures of 425ºC and above are values obtained from time dependent properties.
T10	Allowable stresses for temperatures of 455ºC and above are values obtained from time dependent properties.
T11	Allowable stresses for temperatures of 480ºC and above are values obtained from time dependent properties.
T12	Allowable stresses for temperatures of 510ºC and above are values obtained from time dependent properties.
T13	Allowable stresses for temperatures of 540ºC and above are values obtained from time dependent properties.

T14	Allowable stresses for temperatures of 565ºC and above are values obtained from time dependent properties.
T15	Allowable stresses for temperatures of 595ºC and above are values obtained from time dependent properties.
T16	Allowable stresses for temperatures of 620ºC and above are values obtained from time dependent properties.
T17	Allowable stresses for temperatures of 650ºC and above are values obtained from time dependent properties.
T18	Allowable stresses for temperatures of 675ºC and above are values obtained from time dependent properties.
T19	Allowable stresses for temperatures of 450°C and above are values obtained from time dependent properties.
W1	No welding or brazing permitted.
W2	For Section VIII applications, UNF-56(d) applies for welded constructions.
W3	For welded and brazed constructions, stress values for O (annealed) temper material are used.
W4	The stress values given for this material are not applicable when either welding or thermal cutting is employed.
W5	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC 2550, shall provide a longitudinal weld efficiency factor of 1.0. Other long. weld efficiency factors are in accordance with the following: ▪ For single butt weld, with filler metal, 0.80. ▪ For single or double butt weld, without filler metal, 0.85. ▪ For double butt weld, with filler metal, 0.90. ▪ For single or double butt weld, with radiography, 1.00.
W6	Filler metal is not used in the manufacture of welded pipe or tubing.
W7	Strength of reduced-section tensile specimen required to qualify welding procedures. See QW-150, Section IX.

W8	After welding, heat treat at 625ºC - 650ºC, hold 1½ hr at temperature for the first 25 mm of cross-section thickness and ½ hr for each additional 25 mm, and air cool. For castings used in pumps, valves, and fittings DN 50 and less, PWHT is not required for socket welds and attachment welds when the castings have been temper annealed at 625ºC - 650ºC prior to welding.
W9	If welded or brazed, the allowable stress values for the annealed condition are used and the minimum tensile strength of the reduced tension specimen in accordance with QW-462.1 of Section IX is not less than 205 MPa.
W10	When nonferrous materials conforming to specifications in Section II, Part B are used in welded or brazed construction, the maximum allowable working stresses do not exceed the values given herein for annealed material at the metal temperature shown.
W11	These maximum allowable stress values are to be used in welded or brazed constructions.
W12	These S values do not include a weld factor. For Section VIII, Division 1and and Section XII applications using welds made without filler metal, the tabulated tensile stress values are multiplied by 0.85. For welds made with filler metal, consult UW-12 of Section VIII, Division 1, or TW-130.4 of Section XII, as applicable.
W13	For service at 650ºC or higher, the deposited weld metal is of the same nominal chemistry as the base metal.
W14	No welding permitted.
W15	For Section VIII and XII applications, no welding is permitted.
W16	Use NFA-12 when welded with 5356 or 5556 filler metal, all thicknesses, or 4043 or 5554 filler material, thickness <= 10 mm. Use NFA-13 when welded with 4043 or 5554 filler material, thickness > 10 mm.

Division 1 Superseded Material Notes

Notes for the year 1943

(a)	Allowable working stresses in single shear = 0.8 times the given values.
(b)	Allowable working stresses in double shear = 1.6 times the given values.
(c)	Allowable working stresses in bearing = 1.8 times the given values.

(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	Values of stresses above 700 F are based upon steel in annealed condition.
1	Limited to plates not over 3/4 in. in thickness and to temperatures not above 750 F.
2	Maximum value for tensile strength permitted in design, 55,000 psi.
3	For present, limited to temperatures not above 750 F.
4	Only seamless steel pipe or tubing, or electric-fusion-welded pipe may be used for temperatures above 750 F.
5	Limited to temperatures not above 450 F.
6	Limited to temperatures not above 750 F.
7	Limited to temperatures not above 850 F.

Notes for the year 1952

(a)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
1	See Par. UG-6
2	Flange quality in this specification not permitted over 850 F.
3	These stress values are one-fourth the specified minimum tensile strength multiplied by a quality factor of 0.92, except for SA-283, Grade D. and SA-7.

4	For service temperatures above 850 F it is recommended that killed steels containing not less than 0.19% residual silicon be used. Killed steels which have been deoxidized with large amounts of aluminum and rimmed steels may have creep and stress-rupture properties in the temperature range above 850 F, which are somewhat less than those on which the values in the above table are based.
5	Between temperatures of 650 F and 1000 F, inclusive, the stress values for Specification SA-201, Grade B, may be used until high temperature test data become available.
6	Only (silicon) killed steel shall be used above 900 F.
7	To these stress values a quality factor as specified in Par. UG-24 shall be applied.
8	These stress values apply to normalized and drawn material only.
9	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints, where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the relative flexibility of the flange and bolts, and corresponding relaxation properties.
10	Between temperatures of —20 to 400 F, stress values equal to the lower of the following will be permitted: 20% of the specified tensile strength, or 25% of the specified yield strength.
11	Not permitted above 450 F; allowable stress value 7000 psi.
12	Between temperatures 0(750 F to 1000 F, inclusive, the stress values for Specification SA-212, Grade B, may be used until high temperature test data become available.
13	The stress values to be used for temperatures below —20 F when steels are made to conform with Specification SA-300 shall be those that are given in the column for —20 to 650 F.

Notes for the year 1965: (TABLE UCS-23)

(a)	Stress values in restricted shear such as dowel bolts, rivets, or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area shall be 0.80 times the given values.
(b)	Stress values in bearing shall be 1.60 times the given values.

(c)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
1	See Par. UCS-6(b).
2	Flange quality in this specification not permitted over 850 F.
3	These stress values are one-fourth the specified minimum tensile strength multiplied by a quality factor of 0.92, except for SA-283, Grade D. SA-7 and SA-36.
4	For service temperatures above 850 F it is recommended that killed steels containing not less than 0.10% residual silicon be used. Killed steels which have been deoxidized with large amounts of aluminum and rimmed steels may have creep and stress-rupture properties in the temperature range above 850 F:, which are somewhat less than those on which the values in the above table are based.
5	Between temperatures of 650 F and 1000 F, inclusive, the stress values for Specification SA-201, Grade B, may be used until high temperature test data become available.
6	Only (silicon) killed steel shall be used above 900 F.
7	To these stress values a quality factor as specified in Par. UG-24 shall be applied.
8	These stress values apply to normalized and drawn material only.
9	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints, where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary JS determined from the relative flexibility of the flange and bolts, and corresponding relaxation properties.
10	Between temperatures of —20 to 400 F, Stress values equal to the lower of the following will be permitted: 20% of the specified tensile strength, or 25% of the specified yield strength.
11	Not permitted above 450 F; allowable stress value 7000 psi.

<table>
<tr><td>12</td><td>Between temperatures of 750 F to 1000 F. inclusive, the stress values for Specification SA-212, Grade B, may be used until high temperature test data become available.</td></tr>
<tr><td>13</td><td>The stress values to be used for temperatures below —20 F when steels are made to conform with Specification SA-300 shall be those that are given in the column for —20 to 650 F.</td></tr>
<tr><td>15</td><td>For temperatures below 400 F, stress values equal to 20 per cent of the specified minimum tensile strength will be permitted.</td></tr>
<tr><td>19</td><td>These allowable stress values apply also to structural shapes and bars.</td></tr>
<tr><td>20</td><td>Stress values apply to normalized, or normalized and tempered or oil quenched and tempered material only, as per applicable specification.</td></tr>
<tr><td>21</td><td>Stress values apply to quenched and tempered material only, as per applicable specification.</td></tr>
<tr><td>22</td><td>Welding not permitted when carbon content exceeds 0.35 per cent by ladle analysis except for repairs or non-pressure attachments as outlined in Part UF.</td></tr>
<tr><td>23</td><td>Welding or brazing not permitted on liquid quenched and tempered material.</td></tr>
<tr><td>24</td><td>Maximum allowable stress values shall be as follows:
<table>
<tr><td>Grade</td><td>Liquid Quenched and Tempered (-20 to 200F)</td><td>Other Than Liquid Quenched and Tempered (-20 to 200F)</td></tr>
<tr><td>I</td><td>15,000</td><td>15,000</td></tr>
<tr><td>II</td><td>18,750</td><td>18,750</td></tr>
<tr><td>III</td><td>22,500</td><td>22,500</td></tr>
<tr><td>IV</td><td>26,250</td><td>26,250</td></tr>
<tr><td>V(A,B&E)</td><td>30,000</td><td></td></tr>
<tr><td>V(C&D)</td><td>30,000</td><td></td></tr>
</table>
</td></tr>
<tr><td>25</td><td>See Par. UCS-6 (c).</td></tr>
<tr><td>26</td><td>This material shall not be used in thicknesses above 0.58 in.</td></tr>
</table>

Notes for the year 1965:(TABLE UHA-23)

1	Due to the relatively low yield strength of this material, the higher stress values at temperatures from 200 through 1050F were established to permit the use of this material where slip-hay greater deformation is acceptable. The stress values within the above range exceed 621/2 per cent, but do not exceed 90 percent of the yield strength at temperature. These stress values are not recommended for the design of flanges or piping.
2	These stress values at temperatures of 1050F and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6.
3	These stress values shall be considered basic values to be used when no effort is made to control or check the grain size of the steel.
4	These stress values are the basic values multiplied by a joint efficiency factor of 0.85.
5	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints where freedom from leakage over a long period of time without retightening is required, lower values may be necessary as determined from the flexibility of the flange and bolts and corresponding relaxation.
6	These stress values a quality factor as specified in Par. UG-24 shall be applied.
7	These stress values permitted for material that has been carbide-solution treated.
8	For temperatures below 100F, stress values equal to 20 percent of the specified minimum tensile strength will be permitted.
9	This steel may be expected to develop embrittlement at room temperature after service at temperatures above 800F: consequently, its use at higher temperatures is not recommended unless due caution is observed.
10	At temperatures over 1000F, these stress values apply only when the carbon is 0.04 percent or higher.
11	For temperatures above 800F, the stress values apply only when the carbon content is 0.04 percent and above.
12	These stress values shall be applicable to forgings over 5 inches in thickness.

Notes for the year 1974

(a)	Stress values in restricted shear such as dowel bolts, rivets, or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area shall be 0.80 times the given values.
(b)	Stress values in bearing shall be 1.60 times the given values.
(c)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
1	See UCS-6(b).
3	These stress values are one fourth the specified minimum tensile strength multiplied by a qualify factor of 0.92, except for SA-283, Grade D, and SA-36.
4	For service temperatures above 850 F it is recommended that killed steels containing not less than 0.10 percent residual silicon be used. Killed steels which have been deoxidized with large amounts of aluminum and rimmed steels may have creep and stress rupture properties in the temperature range above 850 F. which are somewhat less than those on which the values in the above Table are based.
5	Between temperatures of 650 and 1000 F, inclusive, the stress values for Specification SA-201. Grade B. may be used until high temperature test data become available.
6	Only killed steel shall be used above 850 F.
7	To these stress values a quality factor as specified in UG-24 shall be applied for castings.
8	These stress values apply to normalized and drawn material only.
9	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints, where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the relative flexibility of the flange and bolts, and corresponding relaxation properties.
11	Not permitted above 450F; allowable stress value 7000 psi.
12	Between temperatures of 750 and 1000 F, inclusive, the stress values for Specification SA-515, Grade 70. May be used until high temperature test data become available.
13	The stress values to be used for temperatures below —20F when steels are made to conform with supplement (5)SA-20 shall be those that are given in the column for —20 to 650 F.

15	For temperatures below 400 F, stress values equal to 20 percent of the specified minimum tensile strength will be permitted.
19	These allowable stress values apply also to structural shapes and bars.
20	Stress values apply to normalized, or normalized and tempered or oil quenched and tempered material only, as per applicable specification.
21	Stress values apply to quenched and tempered material only, as per applicable specification.
22	Welding or brazing is not permitted when carbon content exceeds 0.35 percent by ladle analysis except for limited types of welding as allowed in Part UF.
23	Welding or brazing not permitted on liquid quenched and tempered material.
24	Maximum allowable stress values shall be as follows:

Grade	**Normalized or Normalized and Tempered**			**Liquid Quenched and Tempered**				
	-20 to 650	-20 to 100	200	300	400	500	600	650
I	15,000	15,000	15,000					
II	18,750	18,750	18,750					
III	22,500	22,500	22,500					
IV	26,250	26,250	25,050	24,600	24,600	24,600	24,600	24,600
VA	30,000	28,850	28,850	28,850	28,850	28,850	28,850	
VB	30,000	29,050	28,500	28,500	28,200	27,800	26,750	
VE	30,000	29,800	28,700	28,700	28,700	28,700	27,500	
VC&D		30,000	30,000					
VIII		33,700	32,300	32,100	31,900	31,600	31,400	30,000

26	This material shall not be used in thicknesses above 0.58 in.
27	Upon prolonged exposure to temperatures above 800 F. the carbide phase of carbon steel may be converted to graphite.
28	Upon prolonged exposure to temperatures above 875 F, the carbine phase of carbon-molybdenum steel may be converted to graphite.

29	The material shall not be used in thickness above 0.375 in.
30	For temperatures above which stresses are given, the allowable stresses for the annealed plate shall be used.
31	Where the fabricator performs the heat treatment the requirements of UHT-81 shall be met.
32	Section IX, QW-250 Variables QW404.12, QW406.3, QW407.2, and QW-409.1 of QW-422 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Part UF of Division I.

Division 2 Material Notes for Table 2A (Ferrous Materials) - Customary

(a)	The following abbreviations are used: Smls., Seamless; Temp., Temperature; and Wld., Welded.
(b)	An alternative typeface is used for stress values based on successful experience in service (see Notes E1 and E2).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 are used when SA-516M Grade 485 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures are rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See **Nonmandatory Appendix A** for more information.
E1	For values at 650ºF and above, the design stress intensity values are based on successful experience in service.
E2	For values at 700ºF and above, the design stress intensity values are based on successful experience in service.
G1	Material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted.
G2	Material that conforms to Class 11 or 12 is not permitted.

G3	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds ¾ in.
G4	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds 1¼ in.
G5	A product analysis is required on this material.
G6	SA-723 is not used for minimum permissible temperature below 40°F.
G7	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3 % but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. **Table Y-2** lists multiplying factors that, when applied to the yield strength values shown in **Table Y-1**, give allowable stress values that will result in lower values of permanent strain.
G8	This material has reduced toughness at room temperature after exposure at high temperature. The degree of embrittlement depends on composition, heat treatment, time and temperature. The lowest temperature of concern is about 500ºF. See **Appendix A, A-360**.
G9	At temperatures over 1000ºF, these stress intensity values apply only when the carbon is 0.04% or higher. This note is applicable only when stresses above 1000ºF are published.
G10	For temperatures above 1000°F, these stress intensity values may be used only if the material has been heat treated by heating to a minimum temperature of 1900°F and quenching in water or rapidly cooling by other means. This note is applicable only when stresses above 1000ºF are published.
G11	These stress intensity values at temperatures of 1050°F and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6. This note is applicable only when stresses above 1000°F are published.
G12	These stress intensity values are considered basic values to be used when no effort is made to control or check the grain size of the steel.
G13	This steel may be expected to develop embrittlement after service at moderately elevated temperature. See **Appendix A, A-340** and **A-360**.

G14	All forgings have a maximum tensile strength not in excess of 35 ksi above the specified minimum.
G15	Fabricated from SA-387 Grade 12 Class 1 plate.
G16	Fabricated from SA-387 Grade 12 Class 2 plate.
H1	Annealed.
H2	Normalized and tempered.
H3	Pieces that are formed (after quenching and tempering) at a temperature lower than 25ºF below the final tempering temperature are heat-treated after forming when the extreme fiber strain from forming exceeds 3%. Heat treatment shall be 1075ºF minimum, but not higher than 25ºF below the final tempering temperature for a minimum time of one hour per inch of thickness. Pieces formed at temperatures within 25ºF higher than the original tempering temperature are requenched and tempered, either before or after welding into the vessel.
S1	The maximum thickness of forgings does not exceed 3¾ in. (4 in. as heat treated).
S2	Both NPS 8 and larger, and schedule 140 and heavier.
S3	The minimum thickness of pressure-retaining parts is ¼ in..
S4	The minimum thickness of shells, heads, and other pressure-retaining parts is ¼ in.. The maximum thickness is limited only by the ability to develop the specified mechanical properties.
W1	Not for welded construction.
W2	In welded construction, for temperatures above 850°F, the weld metal has a carbon content of greater than 0.05%.

W3	The following, in addition to the variables in Section IX, QW-250, are considered as essential variables requiring requalification of the welding procedure. ▪ An increase in the maximum or a decrease in the minimum specified preheat or interpass temperatures. The specified range of preheat temperatures does not exceed 150ºF. ▪ A change in the thickness T of the welding procedure qualification test plate as follows: a. For welded joints that are quenched and tempered after welding, any increase in thickness (the minimum thickness qualified in all cases is ¼ in.). b. For welded joints that are not quenced and tempered after welding, any changes as follows: (a) For T less than 5/8 in., any decrease in thickness (the maximum thickness qualified is 2T); (b) for T equal to 5/8 in. and over, any departure from the range of 5/8 in. to 2T.

Division 2 Material Notes for Table 2A (Ferrous Materials) - Metric

(a)	The following abbreviations are used: Smls., Seamless; Temp., Temperature; and Wld., Welded.
(b)	An alternative typeface is used for stress values based on successful experience in service (see Notes E1 and E2).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 shall be used when SA-516M Grade 485 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures are rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See **Nonmandatory Appendix A** for more information.
E1	For values at 350°C and above, the design stress intensity values are based on successful experience in service.
E2	For values at 375°C and above, the design stress intensity values are based on successful experience in service.

G1	Material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted.
G2	Material that conforms to Class 11 or 12 is not permitted.
G3	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds 19 mm.
G4	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds 32 mm.
G5	A product analysis is required on this material.
G6	SA-723 is not used for minimum permissible temperature below +5°C.
G7	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. **Table Y-2** lists multiplying factors that, when applied to the yield strength values shown in **Table Y-1**, will give allowable stress values that will result in lower levels of permanent strain.
G8	This material has reduced toughness at room temperature after exposure at high temperature. The degree of embrittlement depends on composition, heat treatment, time, and temperature. The lowest temperature of concern is about 250°C. See Appendix A, A-360.
G9	At temperatures over 550°C, these stress intensity values apply only when the carbon is 0.04% or higher. This note is applicable only when stresses above 550°C are published.
G10	For temperatures above 550°C, these stress intensity values may be used only if the material has been heat treated by heating to a minimum temperature of 1040°C and quenching in water or rapidly cooling by other means. This note is applicable only when stresses above 550°C are published.
G11	These stress intensity values at temperatures of 575°C and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6. This note is applicable only when stresses above 550°C are published.
G12	These stress intensity values are considered basic values to be used when no effort is made to control or check the grain size of the steel.

G13	This steel may be expected to develop embrittlement after service at moderately elevated temperature; see Appendix A, A-340 and A-360.
G14	All forgings have a maximum tensile strength not in excess of 175 MPa above the specified minimum.
G15	Fabricated from SA-387 Grade 12 Class 1 plate.
G16	Fabricated from SA-387 Grade 12 Class 2 plate.
H1	Annealed.
H2	Normalized and tempered.
H3	Pieces that are formed (after quenching and tempering) at a temperature lower than 15°C below the final tempering temperature are heat treated after forming when the extreme fiber strain from forming exceeds 3%. Heat treatment shall be 580°C minimum, but not higher than 15°C below the final tempering temperature for a minimum time of 1 h per 25 mm of thickness. Pieces formed at temperatures within 15°C higher than the original tempering temperature are requenched and tempered, either before or after welding into the vessel.
S1	The maximum thickness of forgings does not exceed 95 mm (100 mm as heat treated).
S2	Both DN 200 and larger, and schedule 140 and heavier.
S3	The minimum thickness of pressure-retaining parts is 6 mm.
S4	The minimum thickness of shells, heads, and other pressure-retaining parts is 6 mm. The maximum thickness is limited only by the ability to develop the specified mechanical properties.
W1	Not for welded construction.
W2	In welded construction, for temperatures above 450°C, the weld metal has a carbon content of greater than 0.05%.
W3	The following, in addition to the variables in Section IX, QW-250, is considered as essential variables requiring requalification of the welding procedure: ▪ An increase in the maximum or a decrease in the minimum specified preheat or interpass temperatures. The specified range of preheat temperatures shall not exceed 85°C. ▪ A change in the thickness T of the welding procedure qualification test plate as follows: a. For welded joints that are quenched and tempered after welding, any increase in thickness (the minimum thickness

	qualified in all cases is 6 mm). b. For welded joints that are not quenched and tempered after welding, any change as follows: (a) for T less than 16 mm, any decrease in thickness (the maximum thickness qualified is 2T) (b) for T equal to 16 mm and over, any departure from the range of 16 mm to 2T.

Division 2 Material Notes for Table 2B (Non-Ferrous Materials)

(a)	The following abbreviations are used: ann., annealed; fin., finished; rel., relieved; Smls., Seamless; and Wld., Welded.
(b)	An alternative typeface is used for stress values based on successful experience in service (see Notes E1 and E2).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 are used when SB-407M Grade N08800 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures are rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See **Nonmandatory Appendix A** for more information.
E2	For values at 800ºF, the design stress intensity values are based on successful experience in service.
G1	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. **Table Y-2** lists multiplying factors that, when applied to the yield strength values shown in **Table Y-1**, will give allowable stress values that will result in lower levels of permanent strain.

G2	Use of external pressure charts for material in the form of barstock is permitted for stiffening rings only.
G3	SB-163 Supplementary Requirement S2 is met.
G4	Design stress intensity values for 100°F may be used at temperatures down to –325°F without additional specification requirements.
G5	A joint efficiency factor of 0.85 has been applied in arriving at the maximum allowable design stress intensity values for this material.
S1	Thickness <= 0.100 in.
W1	Welding except for seal welds is not permitted.

ASME Section VIII Division 2, Table 5A Notes for Ferrous Materials

(a)	The following abbreviations are used: Smls., Seamless; Temp., Temperature; and Wld., Welded.
(b)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T10).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 shall be used when SA-516M Grade 485 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
G1	For temperatures above 1000ºF, these stress values may be used only if the material is heat treated by heating to a minimum temperature of 2000ºF, and quenching in water or rapidly cooling by other means.

G2	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower levels of permanent strain.
G3	At temperatures over 1000ºF, these stress values apply only when the carbon is 0.04% or higher. This note is applicable only when stresses above 1000ºF are published.
G4	For temperatures above 1000ºF, these stress values may be used only if the material has been heat treated by heating to a minimum temperature of 1900ºF and quenching in water or rapidly cooling by other means. This note is applicable only when stresses above 1000ºF are published.
G5	These stress values at temperatures of 1050ºF and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6. This note is applicable only when stresses above 1000ºF are published.
G6	A quality factor of 0.85 has been applied in arriving at the maximum allowable stress values for this material.
G7	These stress values shall be considered basic values to be used when no effort is made to control or check the grain size of the steel.
G8	This steel may be expected to develop embrittlement after service at moderately elevated temperature; see Appendix A, A-340 and A-360.
G9	The tensile strength shall not be in excess of 20,000 psi above the specified minimum.
G10	All forgings shall have a maximum tensile strength not in excess of 25 ksi above the specified minimum.
G11	SA-723 is exempt from the requirement in Section VIII, Division 2, AF-730.3(b) that the average of the individual Brinell hardness numbers shall not be more than 10% below or 25% above the number corresponding to the tensile strength.
G12	See Section VIII, Division 2, Appendix 26.

G13	Upon prolonged exposure to temperatures above 800ºF, the carbide phase of carbon steel may be converted to graphite. See Appendix A, A-240.
G14	Upon prolonged exposure to temperatures above 875ºF, the carbide phase of carbon-molybdenum steel may be converted to graphite. See Appendix A, A-240.
G15	DELETED
G16	Redesignated as G1
H1	Annealed.
H2	Normalized and tempered.
H3	Quenched and tempered.
H4	Liquid quenched and tempered.
H5	Normalized, normalized and tempered, or quenched and tempered.
H6	For applications involving consideration of heat treatment after forming or welding, see Section VIII, Division 2, Table AF-402.1 for P-No. 10K, Group No. 1 materials.
H7	Normalized.
S1	The maximum thickness of forgings shall not exceed 3-3/4 in. (4 in. as heat treated).
S2	The maximum section thickness shall not exceed 3 in. for double-normalized-and-tempered forgings, or 5 in. for quenched-and-tempered forgings.
S3	Both NPS 8 and larger, and schedule 140 and heavier.
T1	Allowable stresses for temperatures of 650ºF and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 700ºF and above are values obtained from time-dependent properties.
T3	Allowable stresses for temperatures of 750ºF and above are values obtained from time-dependent properties.
T4	Allowable stresses for temperatures of 800ºF and above are values obtained from time-dependent properties.

T5	Allowable stresses for temperatures of 850ºF and above are values obtained from time-dependent properties.
T6	Allowable stresses for temperatures of 900ºF and above are values obtained from time-dependent properties.
T7	Allowable stresses for temperatures of 950ºF and above are values obtained from time-dependent properties.
T8	Allowable stresses for temperatures of 1000ºF and above are values obtained from time-dependent properties.
T9	Allowable stresses for temperatures of 1050ºF and above are values obtained from time-dependent properties.
T10	Allowable stresses for temperatures of 1100ºF and above are values obtained from time-dependent properties.
W1	Not for welded construction.
W2	Welding is not permitted when carbon content exceeds 0.35% by ladle analysis except for limited types of welding, as allowed in Section VIII, Division 2, Part AF.
W3	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 100 ksi.
W4	Welded, with the tensile strength of the Section IX reduced section tension test less than 100 ksi but not less than 95 ksi.
W5	In welded construction, for temperatures above 850ºF, the weld metal shall have a carbon content of greater than 0.05%.
W6	Section IX, QW-250 Variables QW-404.12, QW-406.3, QW-407.2, and QW-409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Section VIII, Division 2, Part AF.

W7	The following, in addition to the variables in Section IX, QW-250, shall be considered as essential variables requiring requalification of the welding procedure. (a) An increase in the maximum or a decrease in the minimum specified preheat or interpass temperatures. The specified range of preheat temperatures shall not exceed 150ºF. (b) A change in the thickness T of the welding procedure qualification test plate as follows: (1) For welded joints that are quenched and tempered after welding, any increase in thickness (the minimum thickness qualified in all cases is ¼ in.). (2) For welded joints that are not quenched and tempered after welding, any change as follows: (a) for T less than 5/8 in., any decrease in thickness (the maximum thickness qualified is 2T); (b) for T equal to 5/8 in. and over, any departure from the range of 5/8 in. to 2T.

ASME Section VIII Division 2, Table 5B Notes for Non-Ferrous Materials

(a)	The following abbreviations are used: ann., annealed; Cond., Condenser; extr., extruded; fin., finished; rel., relieved; Smls., Seamless; and Wld., Welded.
(b)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T14).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 shall be used when SB-407M Grade N08800 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.

(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
G1	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower levels of permanent strain.
G2	Use of external pressure charts for material in the form of barstock is permitted for stiffening rings only.
G3	Maximum allowable stress values for 100ºF may be used at temperatures down to –325ºF without additional specification requirements.
G4	Maximum allowable stress values for 100°F may be used at temperatures down to –452ºF without additional specification requirements.
G5	Maximum temperature for external pressure design not to exceed 350°F.
G6	These alloys are occasionally subject to the hazard of stress corrosion cracking. Even though they are suitable for engineering use under a wide variety of corrosive conditions, with no particular hazard with respect to stress corrosion, the supplier of the material should be consulted before applying them.
G7	A joint efficiency factor of 0.85 has been applied in arriving at the maximum allowable stress values for this material.
G8	For stress relieved tempers (T451, T4510, T4511, T651, T6510, T6511), stress values for materials in the basic temper shall be used.
G9	Copper-silicon alloys are not always suitable when exposed to certain media and high temperature, particularly steam above 212°F. The user should satisfy him/herself that the alloy selected is satisfactory for the service for which it is to be used.
G10	At temperatures over 1000ºF, these stress values apply only when the carbon is 0.04% or higher.

G11	This alloy is subject to severe loss of impact strength at room temperatures after exposure in the range of 1000ºF to 1400ºF.
G12	Alloy N06022 in the solution annealed condition is subject to severe loss of impact strength at room temperatures after exposure in the range of 1000ºF to 1250ºF.
G13	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 1500ºF and shall be considered in the design.
H1	For temperatures above 1000ºF, these stress values may be used only if the material is annealed at a minimum temperature of 1900ºF and has a carbon content of 0.04% or higher.
H2	For temperatures above 1000ºF, these stress values may be used only if the material is heat treated by heating it to a minimum temperature of 1900ºF and quenching in water or rapidly cooling by other means.
T1	Allowable stresses for temperatures of 250ºF and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 300ºF and above are values obtained from time-dependent properties.
T3	Allowable stresses for temperatures of 350ºF and above are values obtained from time-dependent properties.
T4	Allowable stresses for temperatures of 400ºF and above are values obtained from time-dependent properties.
T5	Allowable stresses for temperatures of 500ºF and above are values obtained from time-dependent properties.
T6	Allowable stresses for temperatures of 700ºF and above are values obtained from time-dependent properties.
T7	Allowable stresses for temperatures of 750ºF and above are values obtained from time-dependent properties.
T8	Allowable stresses for temperatures of 800ºF and above are values obtained from time-dependent properties.
T9	Allowable stresses for temperatures of 850ºF and above are values obtained from time-dependent properties.
T10	Allowable stresses for temperatures of 900ºF and above are values obtained from time-dependent properties.

T11	Allowable stresses for temperatures of 950ºF and above are values obtained from time-dependent properties.
T12	Allowable stresses for temperatures of 1000ºF and above are values obtained from time-dependent properties.
T13	Allowable stresses for temperatures of 1050ºF and above are values obtained from time-dependent properties.
T14	Allowable stresses for temperatures of 1100ºF and above are values obtained from time-dependent properties.
W1	Welding except for seal welds is not permitted.
W2	For welded construction, stress values for material at O temper shall be used.
W3	The stress values given for this material are not applicable when either welding or thermal cutting is employed.
W4	Use NFA-12 when welded with 5356 or 5556 filler metal, all thickness, or 4043 or 5554 filler metal, thickness <= 3/8 in. Use NFA-13 when welded with 4043 or 5554 filler metal, thickness > 3/8 in.

Material Properties Dialog Box

Displays properties for the selected material. You can modify some properties. Doing so only changes the properties locally. It does not modify the database.

NOTE Properties available in this dialog box vary depending on the command used.

Material Name - Displays the ASME code material specification for the selected item.

Listing # - Indicates a unique identification number for the material.

Yield Stress, Design - Enter the yield stress for the material at the operating temperature. You can find these values in the ASME Code, Section 2 Part D; they are not stored in the material database. If the yield stress at operating temperature is significantly different than the yield stress at ambient temperature, and if some of the items in the model make use of yield stress, such as vessel legs, then you should carefully check and enter this value.

When you select a material from the material database, the software looks up its operating yield stress in the yield stress database and automatically fills in this value. If there are duplicate entries in the yield stress database, then the software displays a message. You can then select from among the duplicates.

Allowable Stress, Design - Enter the allowable stress for the element material at operating temperature. The operating temperature for most vessels is defined to be the same as the design metal temperature for the internal pressure. You can find this value in the ASME Code, Section II, Part D, Table 1A, 1B, and 3.

If you enter a valid material name in the Material Input field, the software searches its database and determines the allowable stress for the material at ambient temperature, and populates this field.

NOTE The software also determines the allowable stress when you select a material name from the **Material Selection** window.

Allowable Stress, Ambient - Enter the allowable stress for the element material at ambient temperature. The ambient temperature for most vessels will be 70° F or 100° F or 30° C). You can find this value in the ASME Code, Section II, Part D, Table 1A, 1B, and 3.

NOTE The software also determines the allowable stress when you select a material name from the **Material Selection** window.

Nominal Material Density - Enter the nominal density of the material. The software uses this value to calculate component weights for this analysis. The typical density for carbon steel is 0.2830 lbs/in^3.

Nominal Thickness for this P Number - Enter the thickness for the P number.

Table UCS-57 of the ASME Code, Section VIII, Division 1 lists the maximum thickness above which full radiography is required for welded seams. This thickness is based on the P number for the material listed in the allowable stress tables of the Code.

NOTE If a seam is partially radiographed and the required thickness exceeds the P number thickness, **PV Elite** automatically changes the joint efficiency to 1.0 as stated in the Code.

External Pressure Curve Name

UCS-66 Curve

Select one of the following:

- **Curve A - D** - UCS-66 curves
- **Impact Tested** - AM 218.1 impact test exemption curve
- **Not a Carbon Steel**

NOTE By default, the material database selects the non-normalized curve. Select **Is the Material Normalized?** or click **Normalized** to use the normalized curve for ASME material. Adjust the curve if you are using normalized material produced to fine grain practice.

The following is from Section VIII Division 1, Figure UCS-66, ed. 2011a Addenda:

1. **Material Curve A**
   a. All carbon and all low alloy steel plates, structural shapes, and bars not listed in Curves B, C, and D below.
   b. SA-216 Grades WCB and WCC if normalized and tempered, or water-quenched and tempered; SA-217 Grade WC6 if normalized and tempered, or water-quenched and tempered.
2. **Material Curve B**
   a. SA-216 Grade WCA if normalized and tempered or water-quenched and tempered
   SA-216 Grades WCB and WCC for thicknesses not exceeding 2 in. (50 mm), if produced to fine grain practice and water-quenched and tempered.
   SA-217 Grade WC9 if normalized and tempered
   SA 285 Grades A and B
   SA 414 Grade A

SA-515 Grade 60
SA-516 Grades 65 and 70 if not normalized
SA-612 if not normalized
SA/EN Grade B if not normalized
SA/EN 10028-2 Grades P235GH, P265GH, and P295GH as rolled
SA/AS 1548 Grades PT430NR and PT460NR

b. Except for cast steels, all materials of Curve A if produced to fine grain practice and normalized which are not listed in Curves C and D below;

c. All pipe, fittings, forgings, and tubing not listed for Curves C and D below;

d. Parts permitted under UG-11 shall be included in Curve B even when fabricated from plate that otherwise would be assigned to a different curve.

3. **Material Curve C**

a. SA 182 Grades 21 and 22 if normalized and tempered
SA 302 Grades C and D
SA 336 Grades F21 and F22 if normalized and tempered, or liquid-quenched and tempered
SA 387 Grades 21 and 22 if normalized and tempered, or liquid-quenched and tempered
SA 442 Grades 55 <= 1 in. if not to fine grain practice and normalized
SA 516 Grades 55 and 60 if not normalized
SA 533 Grades B and C SA 662 Grade A

b. All materials listed in 2(a) and 2(c) or Curve B if produced to fine grain practice and normalized, normalized and tempered, or liquid-quenched and tempered as permitted in the material specification, and not listed for Curve D below.

4. **Material Curve D**

SA 203

SA 508 Grade 1

SA 516 if normalized, or quenched and tempered

SA 524 Classes 1 and 2

SA 537 Classes 1, 2, and 3

SA 612 if normalized

SA 662 if normalized

SA 738 Grade A

SA 738 Grade A with Cb and V deliberately added in accordance with the provisions of the material specification, not colder than -20ºF (-29ºC)

SA 738 Grade B not colder than -20ºF (-29ºC)

SA/AS 1548 Grades PT430N and PT460N

SA/EN 10028-2 Grades P235GH, P265GH, and P295GH if normalized

SA/EN 10028-3 Grade P275NH

Impact Test Exempted

If you are using an impact tested material when no MDMT calculations are required, select **Impact tested Material**.

Is the Material Normalized? - Click to use the ASME normalized curve for the material. For more information, see **UCS-66 Curve**.

Select one of the following:

- **Curve A - D** - UCS-66 curves
- **Impact Tested** - AM 218.1 impact test exemption curve
- **Not a Carbon Steel**

NOTE By default, the material database selects the non-normalized curve. Select **Is the Material Normalized?** or click **Normalized** to use the normalized curve for ASME material. Adjust the curve if you are using normalized material produced to fine grain practice.

The following is from Section VIII Division 1, Figure UCS-66, ed. 2011a Addenda:

1. **Material Curve A**

    a. All carbon and all low alloy steel plates, structural shapes, and bars not listed in Curves B, C, and D below.

    b. SA-216 Grades WCB and WCC if normalized and tempered, or water-quenched and tempered; SA-217 Grade WC6 if normalized and tempered, or water-quenched and tempered.

2. **Material Curve B**

    a. SA-216 Grade WCA if normalized and tempered or water-quenched and tempered
SA-216 Grades WCB and WCC for thicknesses not exceeding 2 in. (50 mm), if produced to fine grain practice and water-quenched and tempered.
SA-217 Grade WC9 if normalized and tempered
SA 285 Grades A and B
SA 414 Grade A
SA-515 Grade 60
SA-516 Grades 65 and 70 if not normalized
SA-612 if not normalized
SA/EN Grade B if not normalized
SA/EN 10028-2 Grades P235GH, P265GH, and P295GH as rolled
SA/AS 1548 Grades PT430NR and PT460NR

    b. Except for cast steels, all materials of Curve A if produced to fine grain practice and normalized which are not listed in Curves C and D below;

    c. All pipe, fittings, forgings, and tubing not listed for Curves C and D below;

    d. Parts permitted under UG-11 shall be included in Curve B even when fabricated from plate that otherwise would be assigned to a different curve.

3. **Material Curve C**

    a. SA 182 Grades 21 and 22 if normalized and tempered
SA 302 Grades C and D
SA 336 Grades F21 and F22 if normalized and tempered, or liquid-quenched and tempered

SA 387 Grades 21 and 22 if normalized and tempered, or liquid-quenched and tempered
SA 442 Grades 55 <= 1 in. if not to fine grain practice and normalized
SA 516 Grades 55 and 60 if not normalized
SA 533 Grades B and C SA 662 Grade A

b. All materials listed in 2(a) and 2(c) or Curve B if produced to fine grain practice and normalized, normalized and tempered, or liquid-quenched and tempered as permitted in the material specification, and not listed for Curve D below.

4. **Material Curve D**

SA 203

SA 508 Grade 1

SA 516 if normalized, or quenched and tempered

SA 524 Classes 1 and 2

SA 537 Classes 1, 2, and 3

SA 612 if normalized

SA 662 if normalized

SA 738 Grade A

SA 738 Grade A with Cb and V deliberately added in accordance with the provisions of the material specification, not colder than -20ºF (-29ºC)

SA 738 Grade B not colder than -20ºF (-29ºC)

SA/AS 1548 Grades PT430N and PT460N

SA/EN 10028-2 Grades P235GH, P265GH, and P295GH if normalized

SA/EN 10028-3 Grade P275NH

Impact Test Exempted

If you are using an impact tested material when no MDMT calculations are required, select **Impact tested Material**.

Elastic Modulus ID

The elastic modulus reference number is a value that points to or corresponds to a set of data set forth in ASME Section II Part D, tables TM-1, 2 and so on. Unfortunately, many materials have a composition or UNS number that does not match the criteria of what is supplied in the ASME Code. In these cases, the reference number will be brought in as zero. If this happens, you will need to enter in an appropriate value.

Reference Number	**Table**	**Description/UNS Number**
1	TM-1	Carbon Steels with C<= 0.3%
2	TM-1	Carbon Steels with C> 0.3%

Reference Number	Table	Description/UNS Number
3	TM-1	Material Group A
4	TM-1	Material Group B
5	TM-1	Material Group C
6	TM-1	Material Group D
7	TM-1	Material Group E
8	TM-1	Material Group F
9	TM-1	Material Group G
10	TM-1	S13800
11	TM-1	S15500
12	TM-1	S45000
13	TM-1	S17400
14	TM-1	S17700
15	TM-1	S66286
16	TM-2	A03560
17	TM-2	A95083
18	TM-2	A95086
19	TM-2	A95456
20	TM-2	A24430
21	TM-2	A91060
22	TM-2	A91100
23	TM-2	A93003
24	TM-2	A93004
25	TM-2	A96061

Reference Number	Table	Description/UNS Number
26	TM-2	A96063
27	TM-2	A92014
28	TM-2	A92024
29	TM-2	A95052
30	TM-2	A95154
31	TM-2	A95254
32	TM-2	A95454
33	TM-2	A95652
34	TM-3	C93700
35	TM-3	C83600
36	TM-3	C92200
37	TM-3	C92200
38	TM-3	C28000
39	TM-3	C28000
40	TM-3	C65500
41	TM-3	C66100
42	TM-3	C95200
43	TM-3	C95400
44	TM-3	C44300
45	TM-3	C44400
46	TM-3	C44500
47	TM-3	C64200
48	TM-3	C68700

Reference Number	**Table**	**Description/UNS Number**
49	TM-3	C10200
50	TM-3	C10400
51	TM-3	C10500
52	TM-3	C10700
53	TM-3	C11000
54	TM-3	C12000
55	TM-3	C12200
56	TM-3	C12300
57	TM-3	C12500
58	TM-3	C14200
59	TM-3	C23000
60	TM-3	C61000
61	TM-3	C61400
62	TM-3	C65100
63	TM-3	C70400
64	TM-3	C19400
65	TM-3	C60800
66	TM-3	C63000
67	TM-3	C70600
68	TM-3	C97600
69	TM-3	C71000
70	TM-3	C71500
71	TM-4	N02200

Reference Number	Table	Description/UNS Number
72	TM-4	N02201
73	TM-4	N04400
74	TM-4	N04405
75	TM-4	N06002
76	TM-4	N06007
77	TM-4	N06022
78	TM-4	N06030
79	TM-4	N06045
80	TM-4	N06059
81	TM-4	N06230
82	TM-4	N06455
83	TM-4	N06600
84	TM-4	N06617
85	TM-4	N06625
86	TM-4	N06690
87	TM-4	N07718
88	TM-4	N07750
89	TM-4	N08020
90	TM-4	N08031
91	TM-4	N08330
92	TM-4	N08800
93	TM-4	N08801
94	TM-4	N08810

Reference Number	Table	Description/UNS Number
95	TM-4	N08825
96	TM-4	N10001
97	TM-4	N10003
98	TM-4	N10242
99	TM-4	N10276
100	TM-4	N10629
101	TM-4	N10665
102	TM-4	N10675
103	TM-4	N12160
104	TM-4	R20033
105	TM-5	R50250
106	TM-5	R50400
107	TM-5	R50550
108	TM-5	R52400
109	TM-5	R56320
110	TM-5	R52250
111	TM-5	R53400
112	TM-5	R52402
113	TM-5	R52252
114	TM-5	R52404
115	TM-5	R52254
116	TM-5	R60702
117	TM-5	R60705

Reference Number	Table	Description/UNS Number
118	TM-1	12Cr-13Cr Group F
119	TM-1	20+Cr Material Group G
220	TEMA	Ni-Mo Alloy B
221	TEMA	Tantalum
222	TEMA	Tantalum with 2.5% Tungsten
223	TEMA	7 MO (S32900)
224	TEMA	7 MO PLUS (S32950)
225	TEMA	17-19 CR Stn Steel
226	TEMA	AL-6XN Stn Steel (NO8367)
227	TEMA	AL-29-4-2
228	TEMA	SEA-CURE
229	TEMA	2205 (S31803)
230	TEMA	3RE60 (S31500)

Thermal Expansion Coefficient ID

The thermal expansion reference number is a value that points to or corresponds to a set of data set forth in ASME Section II Part D, tables TE-1, 2 and so on. Unfortunately, many materials have a composition or UNS number that does not match the criteria of what is supplied in the ASME Code. In these cases, the reference number will be brought in as zero. If this happens, you will need to enter in an appropriate value. Thermal expansion coefficients are important especially if you are analyzing a heat exchanger.

Reference Number	Table	Description/UNS Number
1	TE-1	Carbon & Low Alloy Steels, Group 1
2	TE-1	Low Alloy Steels, Group 2
3	TE-1	5Cr-1Mo and 29Cr-7Ni-2Mo-N Steels
4	TE-1	9Cr-1Mo
5	TE-1	5Ni-1/4 4Mo

6	TE-1	8Ni and 9Ni
7	TE-1	12Cr, 12cR-1Al, 13Cr, 13Cr-4Ni Steels
8	TE-1	15Cr and 17Cr Steels
9	TE-1	27Cr Steels
10	TE-1	Austentic Group 3 Steels
11	TE-1	Austentic Group 4 Steels
12	TE-1	Ductile Cast Iron
13	TE-1	17Cr-4Ni-4Cu. Condition 1075
14	TE-1	17Cr-4Ni-4Cu, Condition 1150
15	TE-2	Aluminum Alloys
16	TE-3	Copper Alloys C1XXXX Series
17	TE-3	Bronze Alloys
18	TE-3	Brass Alloys
19	TE-3	70Cu-30Ni
20	TE-3	90Cu-10Ni
21	TE-4	N02200 and N02201
22	TE-4	N04400 and N04405
23	TE-4	N06002
24	TE-4	N06007
25	TE-4	N06022
26	TE-4	N06030
27	TE-4	N06045
28	TE-4	N06059, N06686
29	TE-4	N06230
30	TE-4	N06455
31	TE-4	N06600
32	TE-4	N06625

33	TE-4	N06690
34	TE-4	N07718
35	TE-4	N07750
36	TE-4	N08031
37	TE-4	N08330
38	TE-4	N08800, N08801, N08810, N08811
39	TE-4	N08825
40	TE-4	N10001
41	TE-4	N10003
42	TE-4	N10242
43	TE-4	N10276
44	TE-4	N10629
45	TE-4	N10665
46	TE-4	N10675
47	TE-4	N12160
48	TE-4	R20033
49	TE-5	Titanium Gr 1, 2, 2H, 3, 7, 7H, 11, 12, 16, 16H, 17, 26, 26H, 27
50	TE-5	Titanium Gr 9, 28
51	TEMA	5Cr-1/2Mo
52	TEMA	7Cr-1/2Mo & 9Cr-1Mo
53	TEMA	Ni-Mo (Alloy B)
54	TEMA	Nickel (Alloy 200)
55	TEMA	Copper-Silicon
56	TEMA	Admirality
57	TEMA	Zirconium
58	TEMA	Cr-Ni-F3-Mo-Cu-Cb (Alloy 20Cb)

59	TEMA	Tantalum
60	TEMA	Tantulum with 2.5% Tungsten
61	TEMA	17-19 CR (TP 439)
62	TEMA	AL-6XN
63	TEMA	2205 (S31803)
65	TEMA	3RE60 (S31500)
66	TEMA	7 MO PLUS (S32950)
67	TEMA	AL 29-4-2
68	TEMA	SEA-CURE
69	TEMA	880-20 Cu-Ni (C71000)

Yield Stress - Opens the **Yield Stress Record** dialog box, which displays yield stress details of the selected material.

APPENDIX A

Vessel Example Problems

The example problems illustrating these principles are located in the program installation directory/Examples directory.

A P P E N D I X   B

Keyboard and Mouse Commands

Keyboard

The following software actions are defined for the keyboard:

Begin line	<Home>
Begin list	<Home>
Delete character	<Del.>
Delete prev. char	<Backspace>
Delete window	<Alt+F4>
End line	<End>
End list	<End>
Exit	<Shift+F3>
Help	<F1>
Hot key	<Alt+char(with '_')>
Insert toggle	<Ins>
Left word	<Ctrl+left-arrow>
Mark	<Ctrl+F5>
Maximize	<Alt +>
Menu control	<Alt>
Minimize	<Alt ->
Move window	<Alt+F7>
New model	Ctrl+N
Next cell	<Down_arrow>
Next Character	<Right_arrow>
Next field	<Tab>
Next window	<Alt+F6>

Page down	<Page Down>
Page up	<Page Up>
Previous cell	<Up_arrow>
Previous character	<Left_arrow>
Previous field	<Shift+Tab>
Refresh	<F5>
Right word	<Ctrl+right_arrow>
Select	<Enter>
Size window	<Alt+F8>
System button	<Alt .>

Mouse

The following software actions are defined for the mouse:

In Window Objects:	
Choose	<Left-down-click>
Select	<Left-release>

In Vessel Graphics:	
Select element	<Left-release>
Select detail	<Right-down-click>

Glossary

remaining strength factor

FCA

future corrosion allowance

gl

length of groove-like flaw

gw

width of groove-like flaw

LTA

local thin area

MAPnc

Maximum Allowable Pressure in a new and cold condition

MAWP

Maximum Allowable Working Pressure

Slen

Maximum length between ring stiffeners

wmax

maximum pit depth

Index

E

F

G

H

I

J

K

L

M

U

V

W