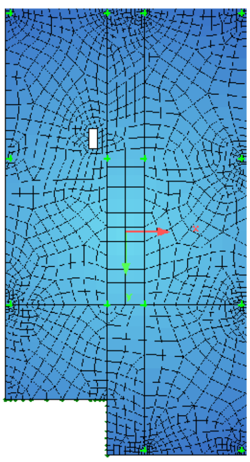
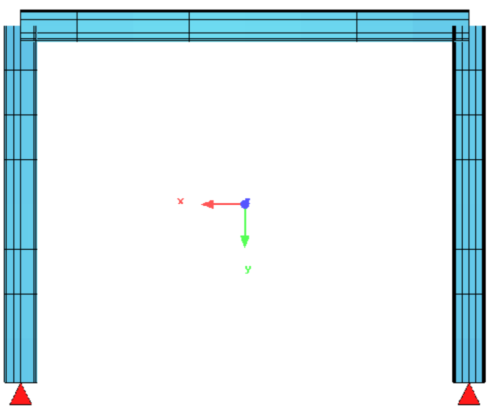
SSD

SOFiSTiK Structural Desktop

User Interface of SOFiSTiK Software

Getting Started





Contents

1 Preface

Dear SOFiSTiK Customer,

With the FEA Version 23 and the SOFiSTiK Structural Desktop (SSD), you have the latest development. The SOFiSTiK Structural desktop (SSD) represents a uniform user interface for the entire range of SOFiSTiK-Software. The SSD module controls Pre-processing, Processing and Post processing.

This Tutorial provides you with a short overview and by means of simple examples a fast entry to the SSD.

In the chapter Basic Workflow we introduce the SSD’s new user interface and explain the fundamental functionality of the individual areas.

The following chapters execute several examples with the SSD. All essential steps are described descriptively so they can simply be reproduced by you. We chose standard examples from literature so that you can compare the results directly.

In the last chapter the essential functions of the SSD are arranged and expounded.

We now wish you much success with the execution of the software.

Your SOFiSTiK Team

2 Basics SSD

2.1 Overview

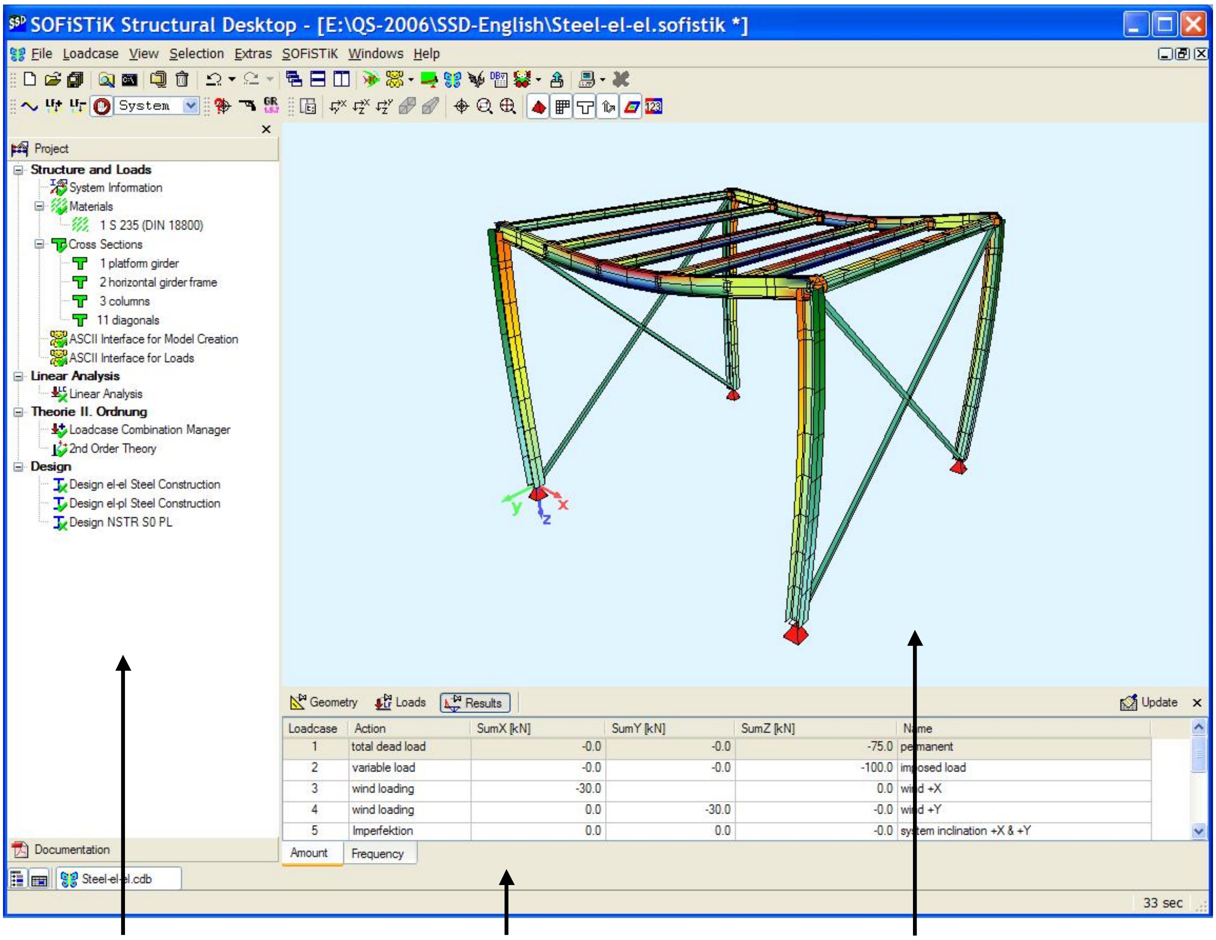
In order to understand the operation of the SOFiSTiK programs, the basic program structure is represented subsequently. It is imperative, that all data is stored in a **central data base** (SOFiSTiK Database CDB).

The SOFiSTiK **software** has a modular structure. The SOFiSTiK Structural Desktop (SSD) controls the communication between all of the individual application programs.

2.2 User interface SSD

The SOFiSTiK Structural Desktop (SSD) represents a uniform user interface for the total range of SOFiSTiK software. The module controls pre processing, processing and post processing. The system can be entered graphically with SOFiPLUS(-X) or as parameterised text input using TEDDY. The control of the calculation and design process takes place using dialog boxes, which are accessed via the task tree.

The screen is divided into three main areas.



1. task tree **2. table area** **3. work area**

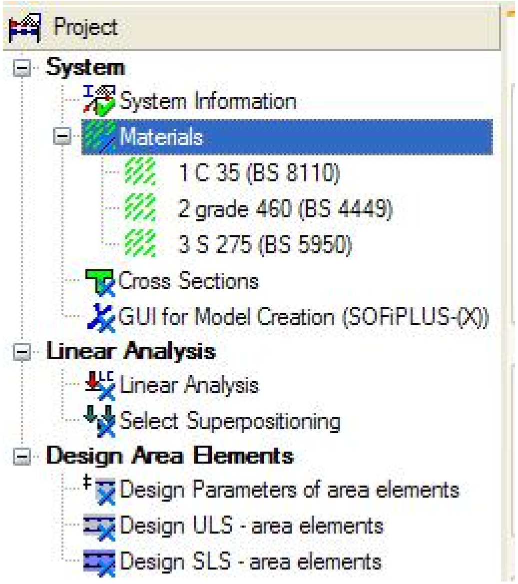
Figure 1: Division of SSD screen

2.3 Basic Workflow

The SSD is task oriented. The tasks are arranged in groups (e.g. the group "System and load" contains the tasks materials, cross sections, geometry and loads).
When creating a new project, the necessary groups and tasks are set by default depending on the chosen problem.

2.3.1 Groups

The computational groups are organized in a tree-structure. This structure can be changed by the user at any time, as the individual tasks can be dragged to the desired place with the mouse. The user can remove or insert additional groups at any time with associated tasks.



Example of a possible group-structure of the SSD:

System

- System and load

Linear Analysis

- Calculation and Superposition

Design Area Elements

- Design ULS and SLS

Figure 2: Group structure SSD

2.3.2 Tasks

The tasks available are accessed via the right-click-menu in the task tree. They can be inserted at any place within the tree. When you select the command “insert task” with the right mouse-button, the following dialog with all available tasks appears.

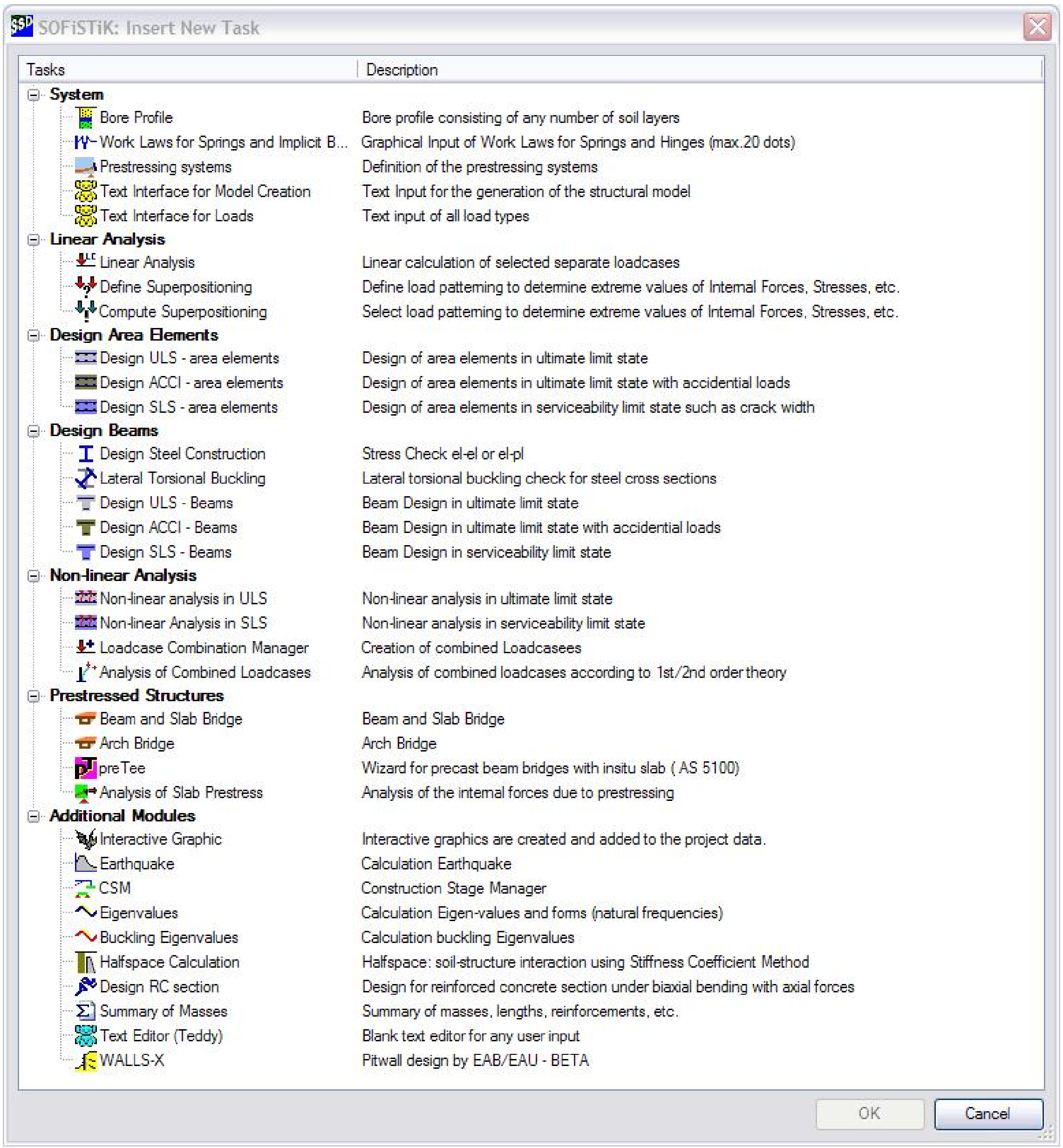
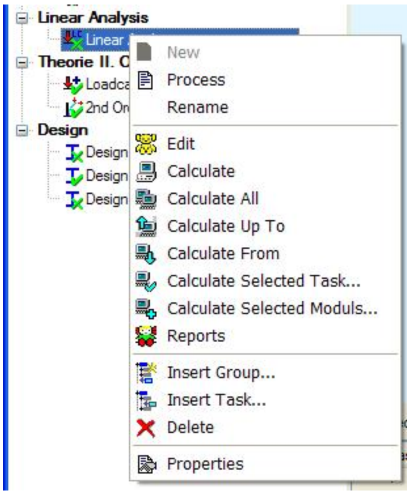


Figure 3: List of the tasks

2.3.2.1 Task Tree

In the task tree the options are accessed via the right-click-menu which automatically adjusts itself to show only those available.



Right click menu in the task tree

The right click menu will provide relevant functions for the selected task.

Examples:

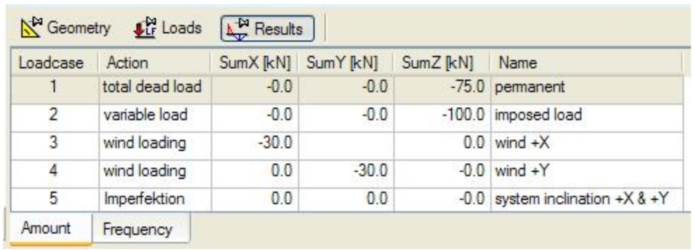
Process à Dialog

Edit à Text-Input (<name>.DAT)

Reports à Result viewer (name.PLB)

Figure 4: Right click menu in the task tree

2.3.2.2 Table Area

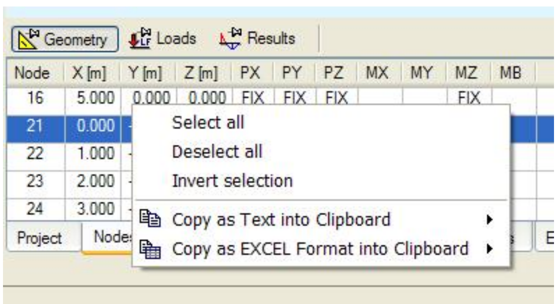


Loadcase	Action	SumX [kN]	SumY [kN]	SumZ [kN]	Name
1	total dead load	-0.0	-0.0	-75.0	permanent
2	variable load	-0.0	-0.0	-100.0	imposed load
3	wind loading	-30.0		0.0	wind +X
4	wind loading	0.0	-30.0	-0.0	wind +Y
5	Imperfektion	0.0	0.0	-0.0	system inclination +X & +Y

Database information is written in the table area.

Possible categories:

- Geometry
- Loads
- Results



These results can be copied with right clicking menu into the clipboard

Possible format:

- Text - Format
- EXCEL- Format

Figure 5: Table area

2.3.2.3 Work Area

The work area displays the ANIMATOR visualisation of the system by default. The work area changes to WinPS during processing to show calculation status and the TEDDY for further text input prior to analysis. The graphical input with SOFiPLUS(-X) operates within its own separate window, making the best possible use of dual monitors.

2.3.3 Template files <name>.SOFiSTiX

For processing of frequently recurrent standard tasks, the Template files of the type <name>.SOFiSTiX are provided. General Templates are saved in a subdirectory of the SOFiSTiK directory, for example C:\Programs\SOFiSTiK\SOFiSTiK.23\ SSD-Templates.

2.3.3.1 Adding User- defined Template directories

For own Templates, the user can define further Template-directories.

⇒ **SOFiSTiK à Options… à SSD-Template Path à (Find-Button) and Add**

In this directory, further subdirectories can be created. These subdirectories appear as tabs and template icons (see Figure 9). There is only one level of subdirectories available.

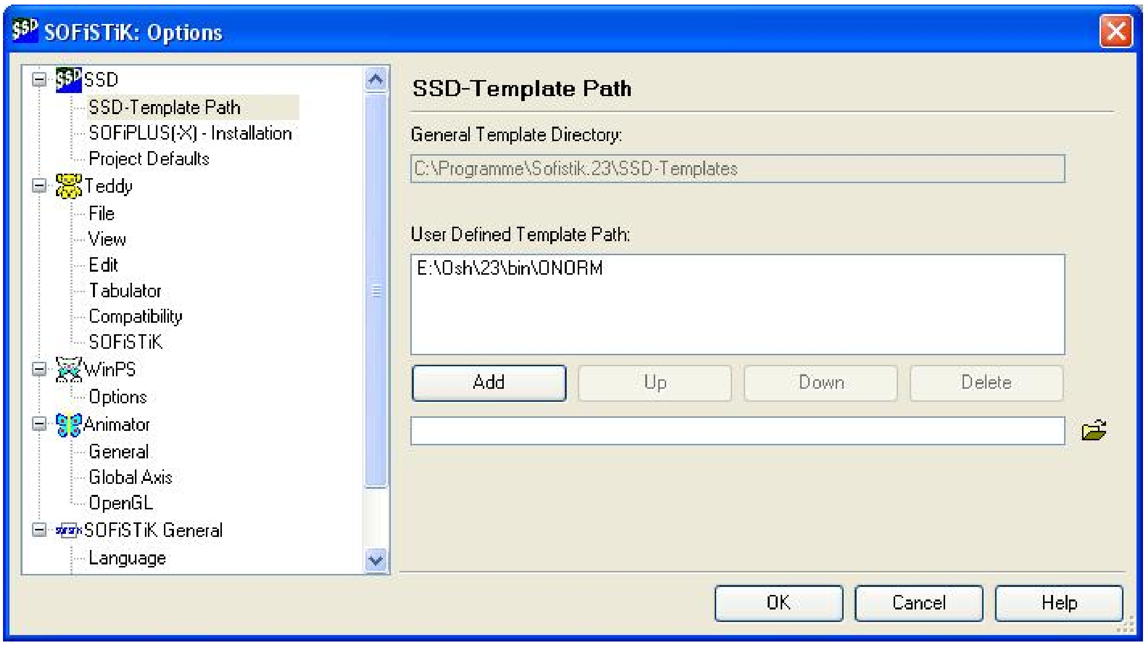


Figure 6: SOFiSTiK à Options à SSD-Template Path

2.3.3.2 *User Defined Template Files*

Any file <name>.SOFiSTiK can be stored into the desired Template-Directory as Template <name>.SOFiSTiX.

All current project settings can be saved as Templates Including the arrangement and sequence of the tasks. The materials and cross sections are dependent on the chosen design code. A fixed design code cannot be changed within the project. ***File à Save As Template***

Figure 7: Dialog Save Project as Template

A later **changing of the code** is possible if the Template is stored “without design code".

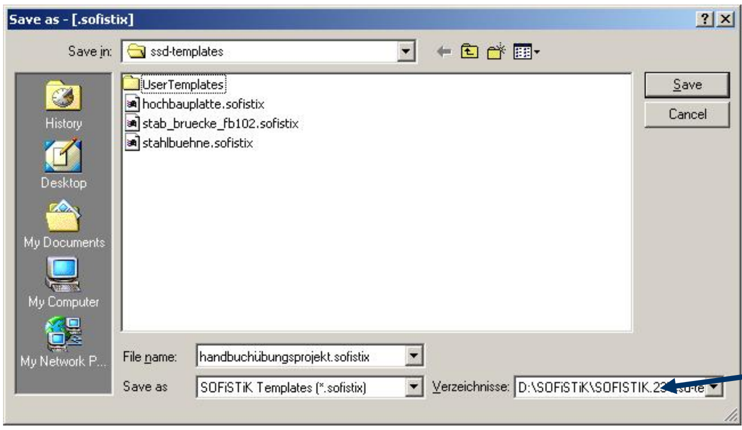


Figure 8: File à Save as Template

The existing Template directories are shown under Directories.

File à New Project from Template

The saved file <name>.SOFiSTiX is now available as a further Template.

2.3.3.3 *Usage of Template Files <name>.SOFiSTiX*

⇒ ***File à New Project from Template…***

The existing Templates from the Template path are offered.

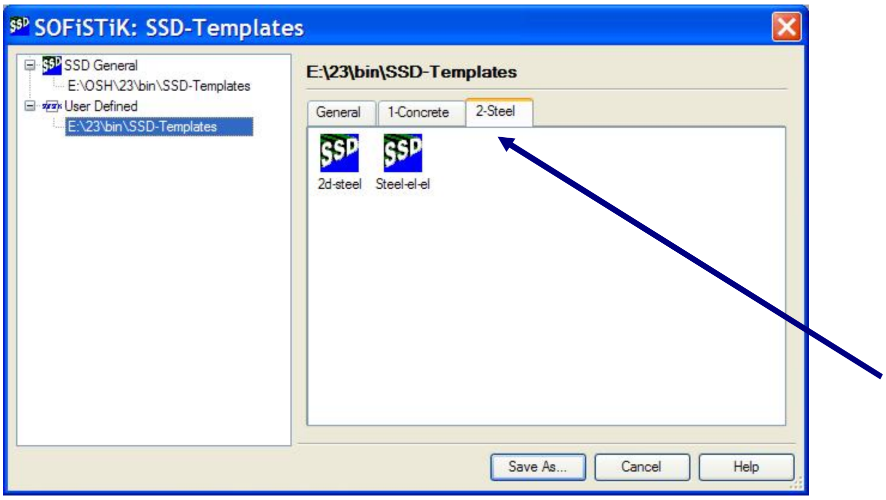


Figure 9: File à New Project from Template ...

The desired file <name>.SOFiSTiX is selected and stored under a new data file name with the button “Save As…” into a project directory.

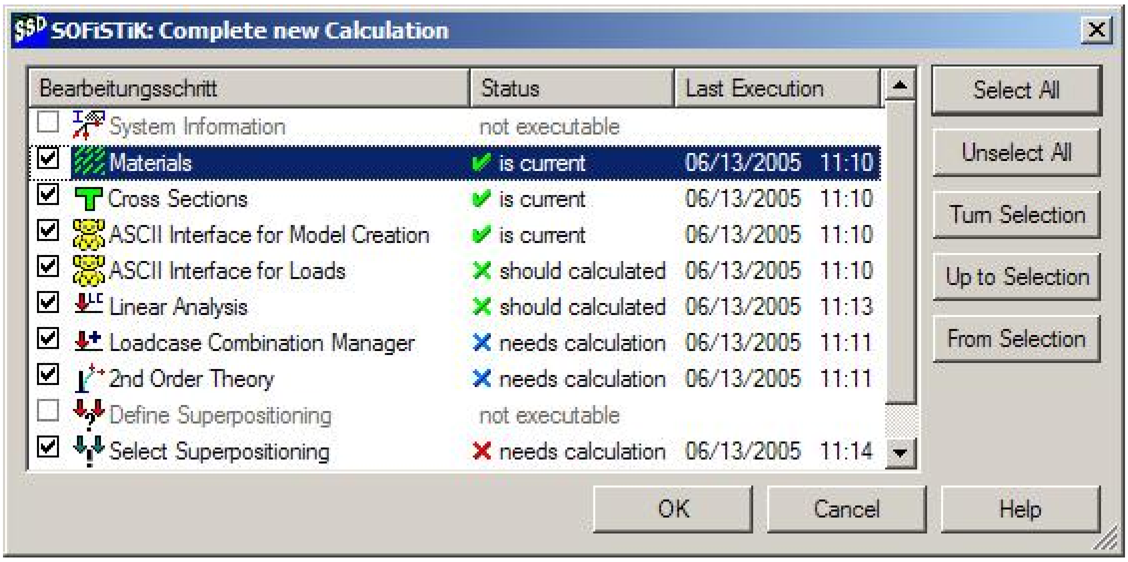
The new file contains all tasks of the Template. In addition, the data (for example cross-sections, geometry... etc.) from the Template are transferred into the new file. The data is then immediately ready for calculation.

With "Templates <u>without</u> Design Code”, the design code can be altered. The materials and cross sections must be checked and amended.

2.4 Structure and Function Mode

2.4.1 Computation status

Every task has its own symbol to show the actual computation status



	Without computation	Input is written directly into the database
	Green check mark	No computation required
	Blue arrow	New input data à computation required
	Blue cross	Old data à computation required
	Red cross	Error message à computation required
	Green cross	Warning message à computation required (possibly)

Figure 10: Computation status

2.4.2 SOFiSTiK Options

Numerous settings within the options dialog are possible of which only the most important are introduced now.

2.4.2.1 Language Settings

There is a difference between the dialog box language and the input/output language. The dialog box language is stored within the REGISTRY of the local computer. The SSD must be started again so that this alteration is activated. The input/output language is stored within the file SOFiSTiK.DEF.

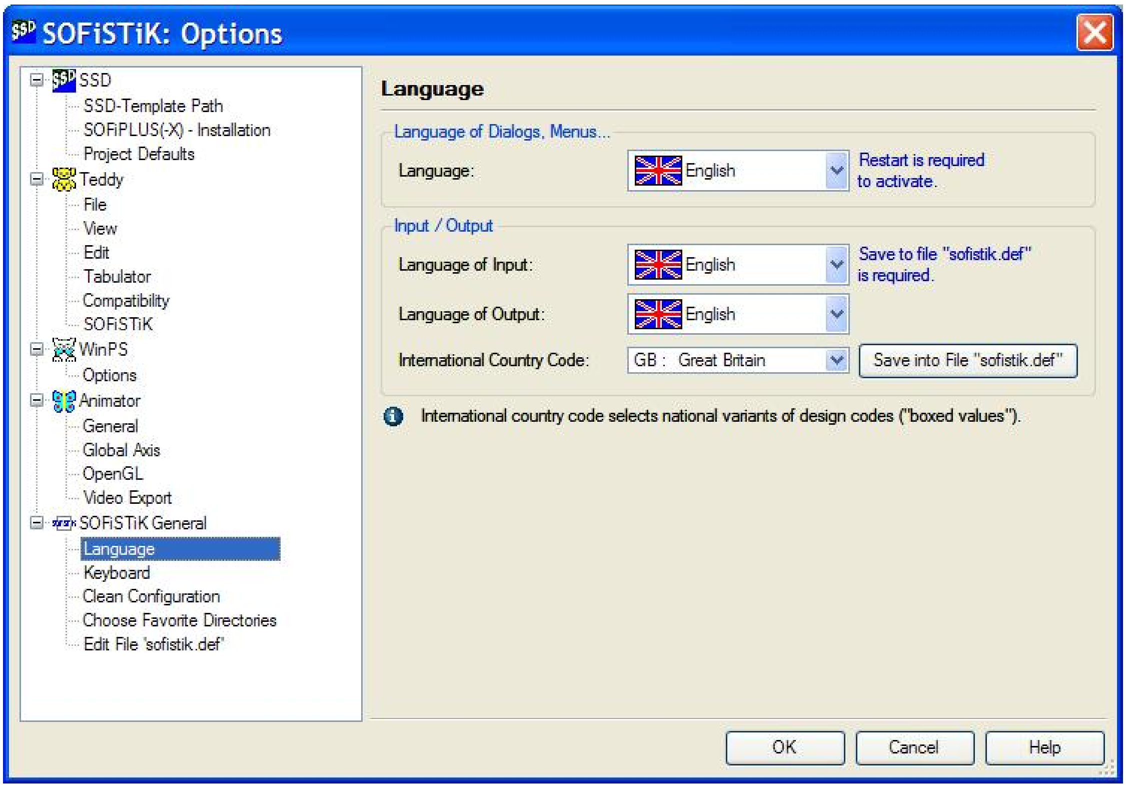


Figure 11: Dialog SOFiSTiK: Options à Language

2.4.2.2 Project defaults

With the option Project Defaults, the user can assign default attributes of new projects. In addition, the user can define if graphical or text is the default input. These attitudes are stored in the Registry.

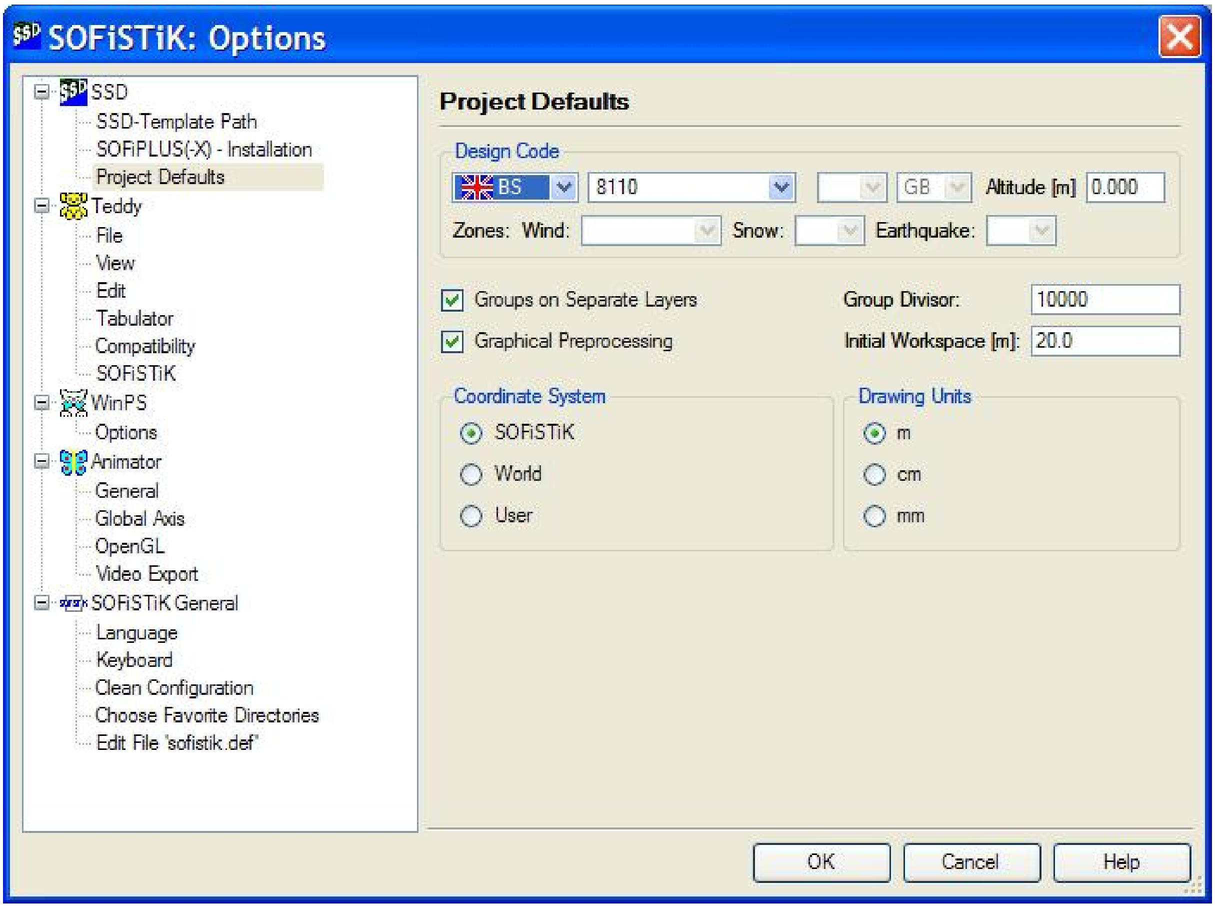


Figure 12: Dialog SOFiSTiK: Options à Project defaults

2.4.2.3 *Process the environment variables in the file SOFiSTiK.DEF*

The SOFiSTiK dialog supports numerous parameters, which are defined as environment variables (in the Environment, in the SOFiSTiK.DEF or in the local <name>.DAT). General defaults are preferably stored in the file SOFiSTiK.DEF. In the following example, the input of a modified company heading takes place in the results listing with the variable SOFiSTiK_NAME.

⇒ ***SOFiSTiK à Options à Edit File "sofistik.def" à Select SOFiSTiK-DEF file with a tick à Edit Button***

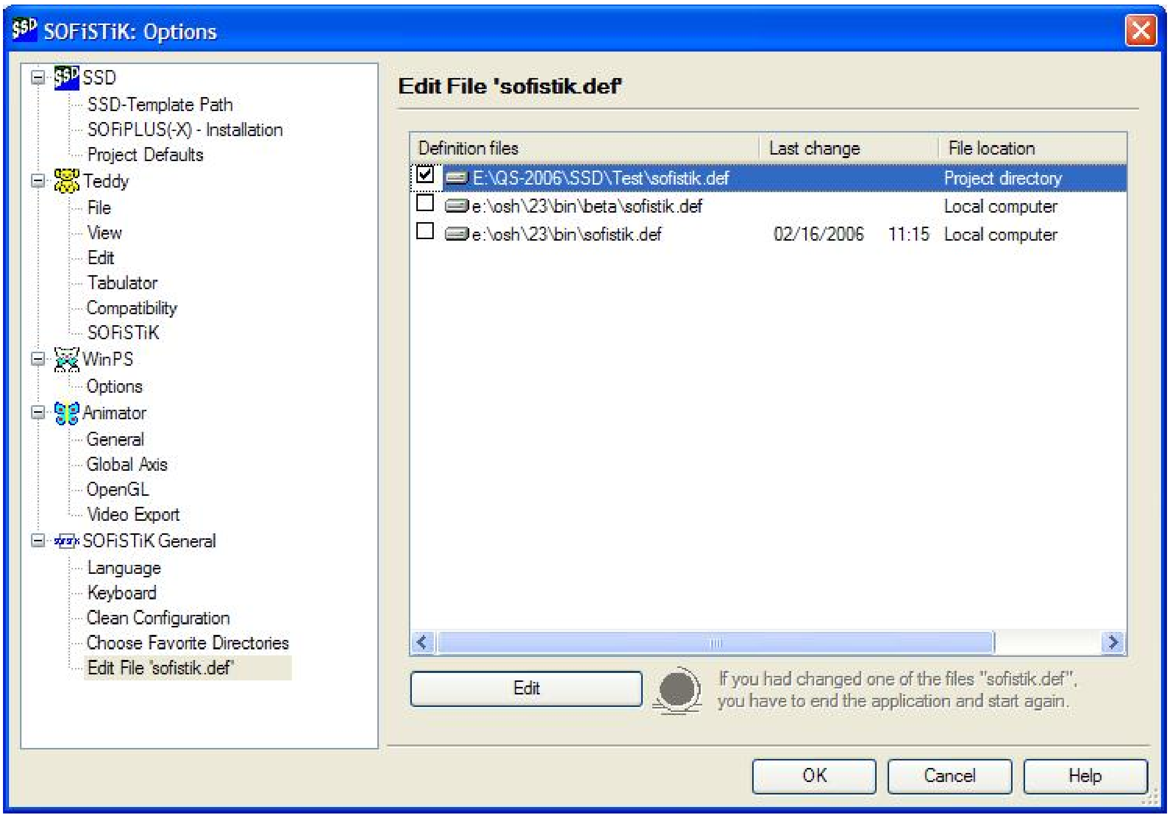


Figure 13: Dialog SOFiSTiK: Options à Edit SOFiSTiK.DEF

The pre-defined parameters are displayed with the button “New”.

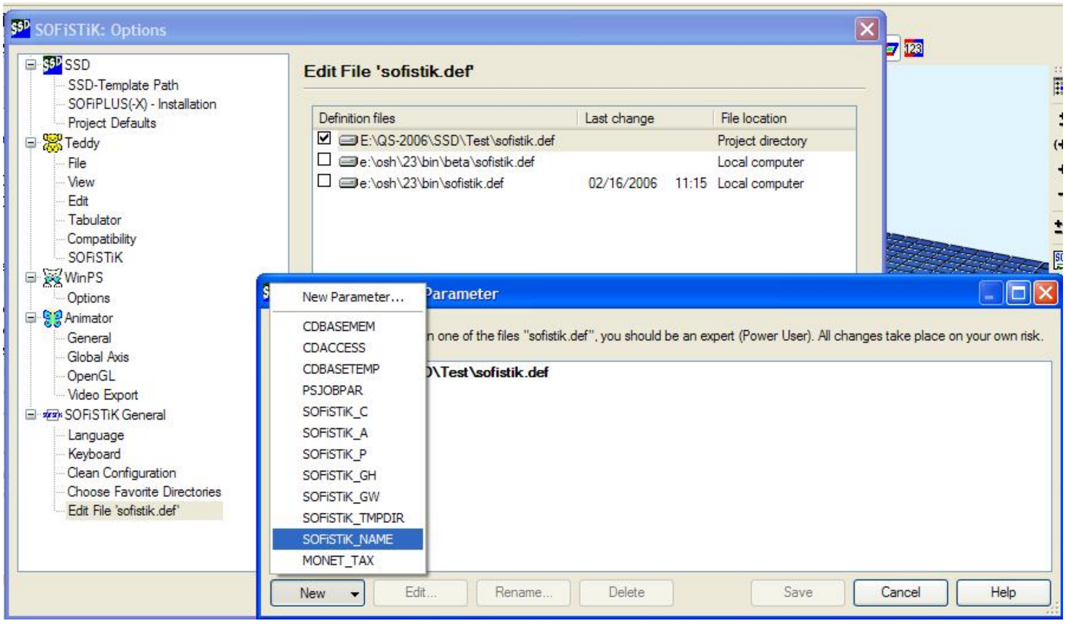
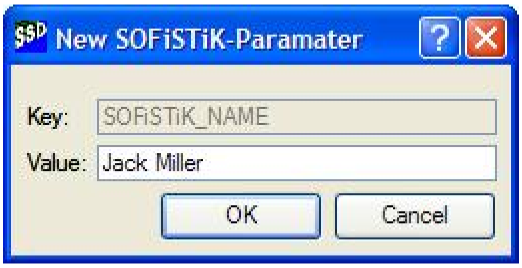
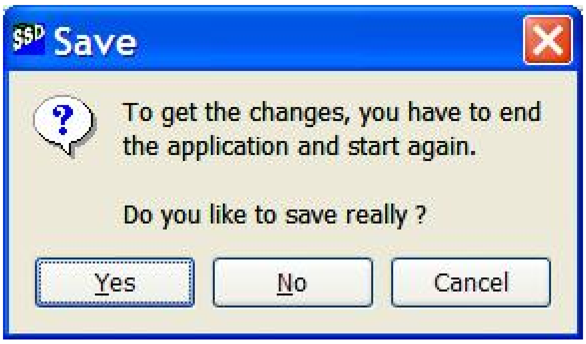


Figure 14: Dialog: Pre-defined parameters



The modified company name now is assigned to the key of SOFiSTiK_NAME as value: "Jack Miller"



Pressing OK stores the new values. However, the program must be closed and re-launched for the new settings to apply.

Figure 15: Allocation of an environment variable

The principle hierarchy of the definition files SOFiSTiK.DEF is shown in Figure 13. The different files SOFiSTiK.DEF are checked in succession one after the other. Highest priority has the setting, which is stored in the SOFiSTiK.DEF file in the project directory. The path of the SOFiSTiK variable is checked off afterwards. If no SOFiSTiK variable is defined in the Environment, the SOFiSTiK directory is used as default.

2.4.3 Files

The basic files are arranged in the following table:

Name.SOFiSTiK	Central XLM- file where all information is saved.
Name.DAT	Control file, ASCII format
Name.PLB	For each task the result is saved in a file task.PLB. This file can be shown and/or printed out individually. Also it can be assigned to the total result using the command "all results".
Name.CDB	Central data base
Name.DWG	Using graphical input all information about system and loading is saved in this drawing.
Name.SOFiSTiX	Template- file in XLM- format: File in Template- directory.

Table 1: Overview of the file- Extensions

3 Basics of SOFiPLUS(-X)

3.1 Basic Input

SOFiPLUS(-X) is a graphic user interface (GUI) for the system generation based on AutoCAD and ADT.

There are two basic generation principles. First you may generate the systems by directly defining the finite elements. Every element and node has to be drawn separately and the system will be generated out of this input. Second you may define the system by structural elements, which are only the wire frame of the structure. After that an automatic mesh generation will create the finite element system for further analysis.

> The important differences are described below. For further information please see the SOFiPLUS(-X) HELP file.

3.1.1 Structural Elements

The definition of Finite Elements starts with the toolbox “SOFiPLUS Classic Create Structure”. Structural Areas, Structural Lines and Structural Points are the main elements for the structure definition.

The basic functionality of Structural Elements are:

- Structural Area
- Curved Area
- Opening
- Structural Line
- Structural Point
- DWG à SOFIMSHB
- The meshing is done externally
- No FE-mesh!
- Copying elements including object attributes is possible

The basic idea with the structural elements is to create a wire frame of the structure. For this all AutoCAD commands can be used. After finishing the wire frame model the automatic mesh generation will create a finite element system and an input data file <name>.DAT which can be modified using TEDDY.

Important is the possibility to define “Free Nodes”, which are independent from mesh and structure. You also may define loads connected to structural elements.

The analysis will be started directly out of SOFiPLUS or with WinPS.

A mixture of Finite Elements and Structural Elements is possible under special conditions. We recommend avoiding any mixture if at all possible.

The Post processing can be done with the normal programs like WinGRAF, DBPRIN and DBView. For the documentation use the URSULA result browser.

The most important attributes are:

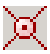
Structure Area

- Defined by Structure Points and Structure Lines
- Plane and curved areas are available
- = geometric area in SOFiMSHB (GAR)

Structural Line

- Area boundary
- Defined by ≥2 Structure Points
- Geometric restraint for SOFiMSHB
- Definition of linear support conditions
- Definition of linear coupling
- Beam, Truss or Cable element
- Pile Element
- Form: straight line, arch, spline
- = geometric line in SOFiMSHB (GLN)

Structure Point



- Geometric restraint for SOFiMSHB
- Part of Structure Line
- Single Support condition
- Column including dimensions for punching design
- Spring
- = geometric point in SOFiMSHB (GPT)

3.1.2 Finite Elements

The definition of Finite Elements starts with the toolbox “SOFiPLUS-Classic Create Elements”. Nodes, constraints, beams, springs, truss elements, cables, boundary elements and area elements are possible elements.

The basic functionality of Finite Elements are:

- FE Elements are produced directly in AutoCAD
- Meshing is done directly in AutoCAD
- CDB export / import
- Copying of elements is possible

Macros available for different mesh generation

4 Example Flat Slab

Example is taken from reference [1], and [2]

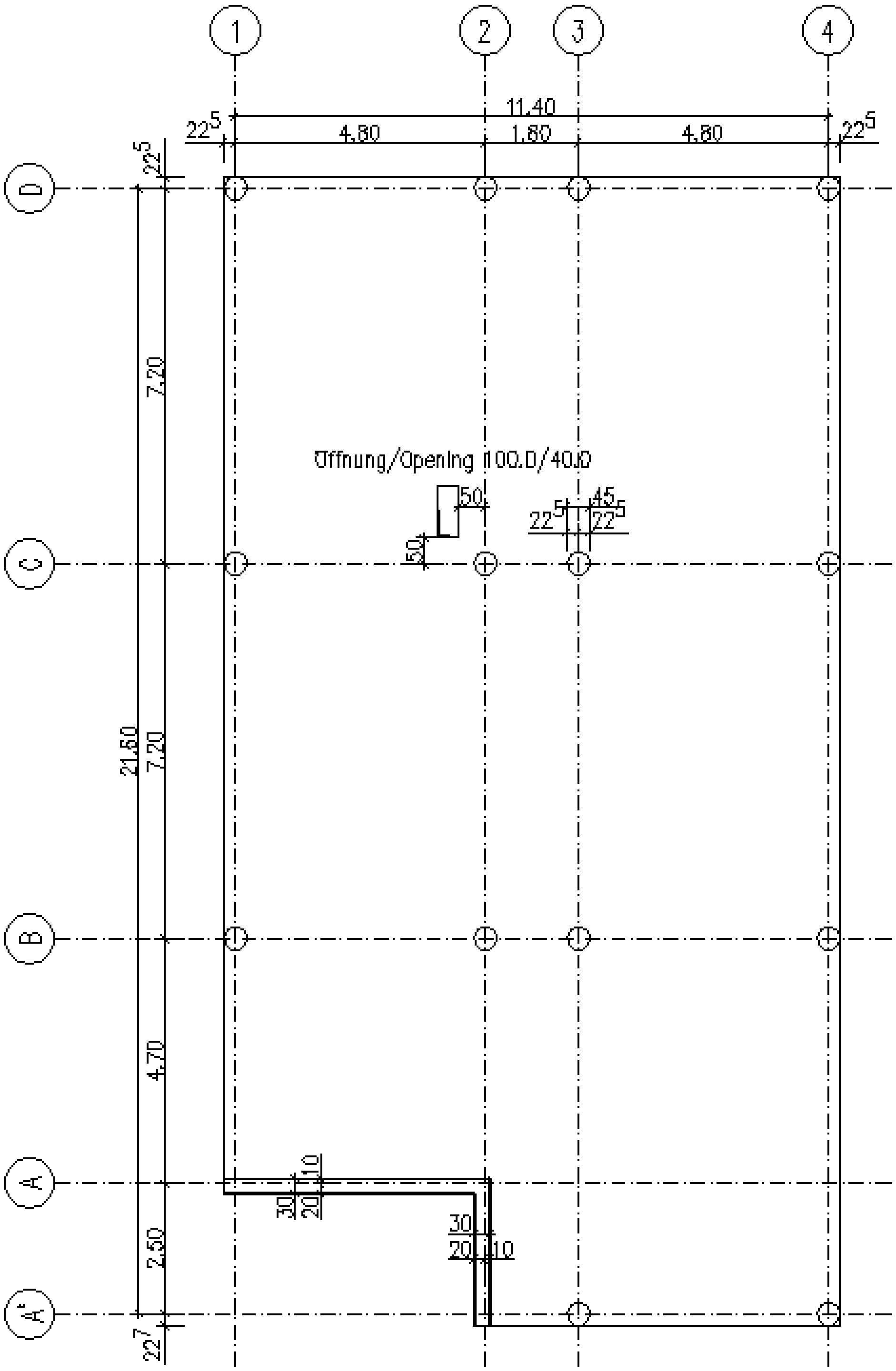


Figure 16: Geometry of Slab

4.1 Problem

A concrete and punching design according to German code DIN 1045-1 for the slab shown in Figure 16 is required.

Materials:	concrete	C 30/37
	reinforcement	BST 500

Exposition class XC1, concrete cover c_{nom} = 20 mm

Columns:	Ø 45 cm	floor height 3,0 m
Walls:	d = 30 cm	floor height 3,0 m
Slab:	d = 24 cm	

Loads:

Dead loads (G)	
Self weight slab 0,24 · 25	= 6,00 kN/m² (automatic calculation)
Dead load 1,50 + 0,50	= 2,00 kN/m²
Variable loads (Q)	
Live load	= 5,00 kN/m²

The following structural design calculations will be done:

Ultimate limit state (ULS)
Serviceability limit state (SLS)
Punching design
Non linear analysis with cracked concrete for long term displacements

The following example shows the basic workflow right from the beginning of a project with the definition of geometry and loads, then analysis and design with the final documentation of results.

4.2 Step 1: Starting SSD

First of all we have to start the SSD with a double click on the button , which should be located on your desktop. To open a new project please use the button / or go via the top menu using: File > New Project…

After that the following dialog will open. Please add title, database name and directory.

Also select design code, system, calculation and choose the option "Graphical Pre-processing".

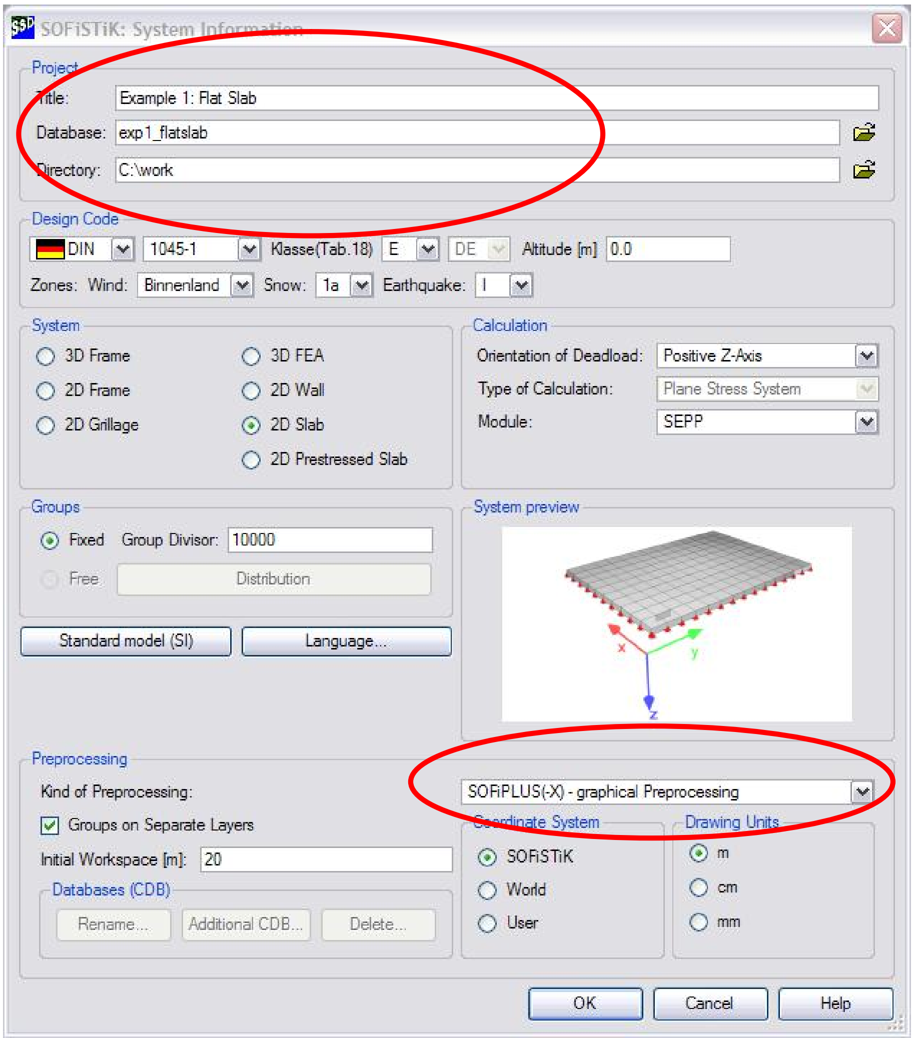


Figure 17: Start dialog – System Information

With selection of "Graphical Pre-processing" additional input is necessary. The default settings are usually OK. Important is the input of the system size. Confirming the input with OK, the program starts a central database exp1-flatslab.cdb.

Changing the design code after saving the project is not possible.

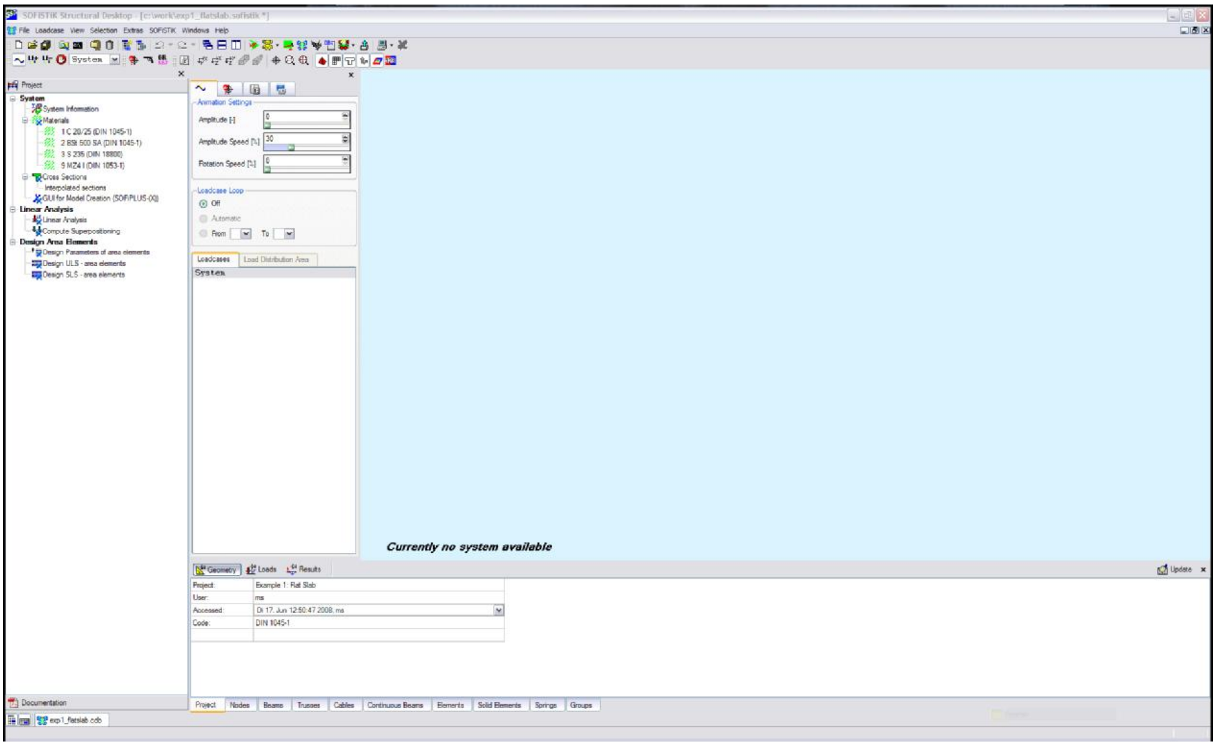


Figure 18: Main window SSD -

As described in chapter 2.3.2.1 the task tree is placed on the left side. With this tree you can access all groups and tasks within the project.

The next step is to define materials and cross sections.

4.3 Step 2: Materials and Cross Sections

Four default materials concrete C 20/25, reinforcement BSt 500, steel S 235 and brick MZ4 I are created. To modify the materials open the dialog with a double click.

The material number 1 concrete C 20/25 has to be changed to a concrete 30/37. Change the classification from 20 to 30. Confirm the change with the OK button.

To delete a material simply use the right mouse click DELETE.

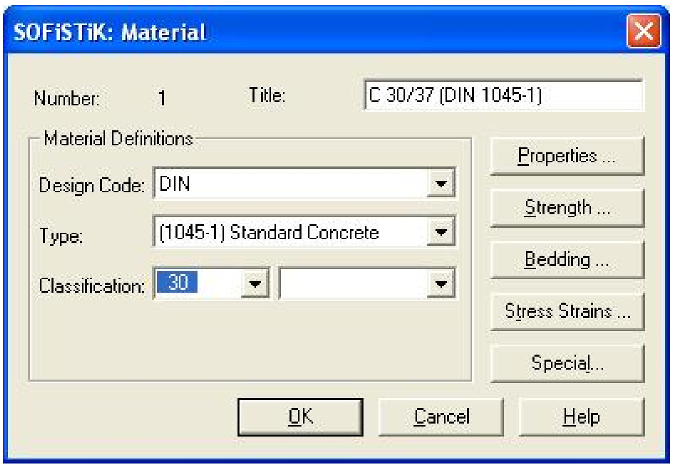


Figure 19: Dialog Material

4.4 Step 3: Graphical Input

For system and load generation the graphical input the program SOFiPLUS(-X) will be used. Now double click on the task “GUI for Model Creation (SOFiPLUS(-X))“ to open it.

_line

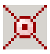
First of all start the drawing of the outline of the slab by using the AutoCAD command ‘line’.. The coordinate origin is located at the top left corner of the slab. Start drawing the outline from the point (0,0) and continue with the following dimensions and directions. .

11,850 horizontal to the right " ,
22.050 vertical down $,
6.825 horizontal to the left ! ,
2.725 vertical up#,
5.025 horizontal to the left ! ,
19.325 vertical up#,
Finish the command with ESC

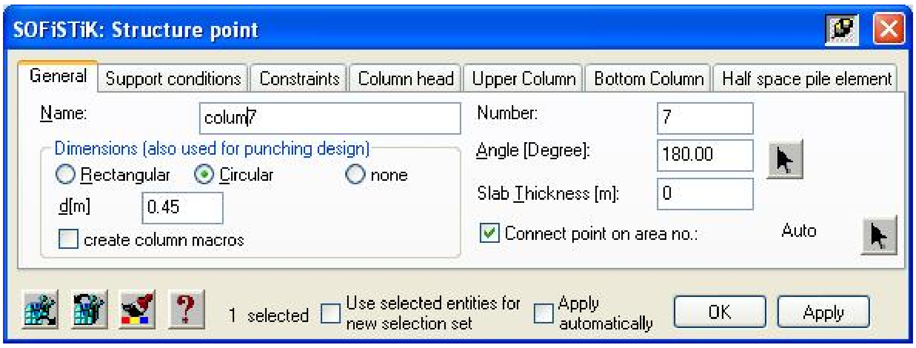
SOF_PM_STEDGE

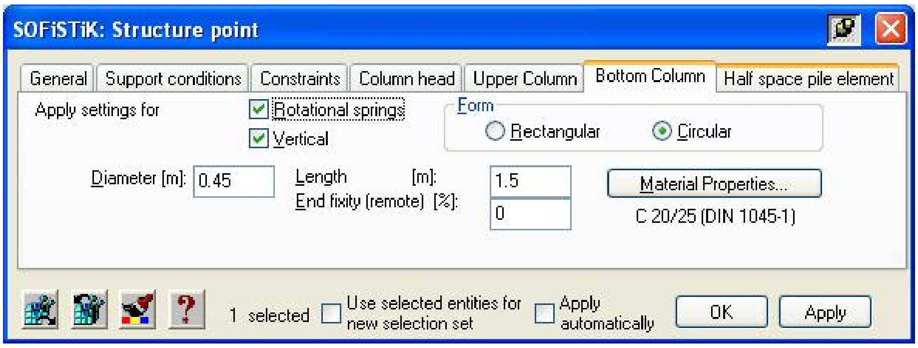
Create structural lines from this outline by opening the command “Structural Line” and select the option “select entity” from the right click menu. Use a selection rectangle starting at the bottom right and moving to the top left side. Confirm the selection by pressing Return.

SOF_PM_STPOINT



Now start the definition of the columns using the command “Structure Point”. The structure points we are going to define now are also the column supports and will be used for punching checks.





Every column has a diameter of 45 cm. For the support conditions select the tab ‘bottom columns’ and apply both the rotational and vertical springs. The spring stiffness is calculated automatically from the cross section, the floor height and the material information provided. The rotational spring stiffness has to be changed to 257000 kNm/rad as written in reference [1] and [2].

The columns are simple to create by using the cursor in the drawing screen. For each column the following coordinates have to be used.

(0.225/0.225)	(5.025/0.225)	(6.825/0.225)	(11.625/0.225)
(0.225/7.425)	(5.025/7.425)	(6.825/7.425)	(11.625/7.425)
(0.225/14.625)	(5.025/14.625)	(6.825/14.625)	(11.625/14.625)
		(6.825/21.825)	(11.625/21.825)

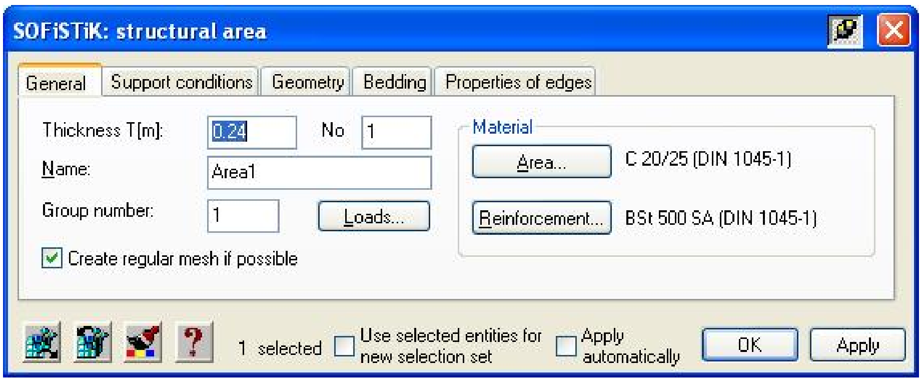
We recommend defining every load area as a structural area. The program automatically creates loadcases for every area.

SOF_PM_STEDGE

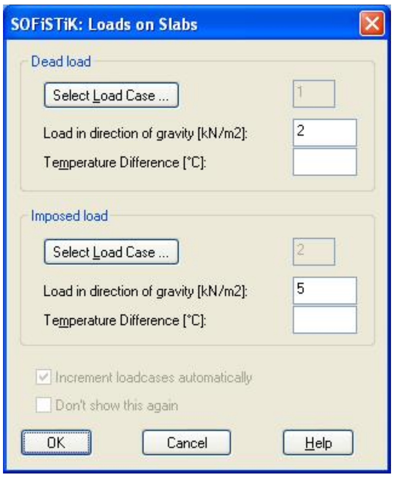
To define structural areas easily, additional structural lines are necessary. Therefore, create new structural lines between the columns and between column and slab outline until the slab is partitioned as shown in Figure 20.

SOF_PM_STAREA

After defining the additional structural lines the structural areas can then be defined. The slab thickness is 0.24 m.



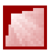
There are four methods to define a structure area. The simplest way is to click into the drawing screen. Please use “select point in area” from the right click menu and click inside the first area.



For load definition open the dialog "Loads on Slabs" and add 2.0 kN/m² for the dead load (LC 1) and 5.0 kN/m² for the following live loads (LC 2, 3 …). Confirm the input with OK.

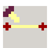
The loadcase number of the live loads will increase by 1 automatically.

Opening

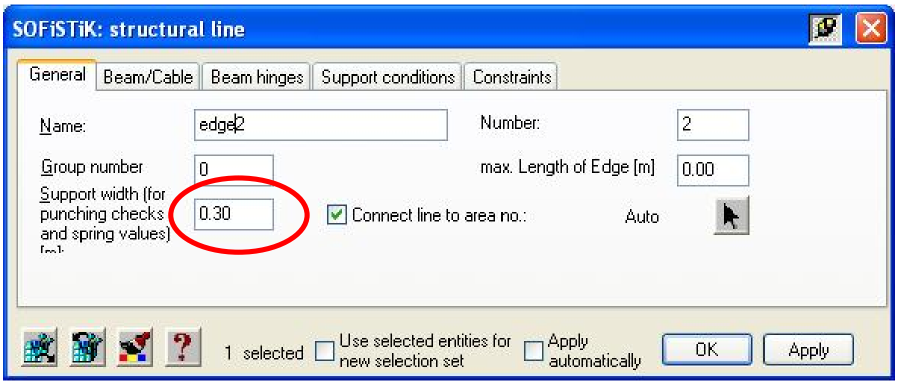


Please add an opening. First we recommend drawing the opening outline using normal AutoCAD Lines. Then start the command "Opening" and chose "PICK point within opening" After clicking inside the opening the definition is finished.

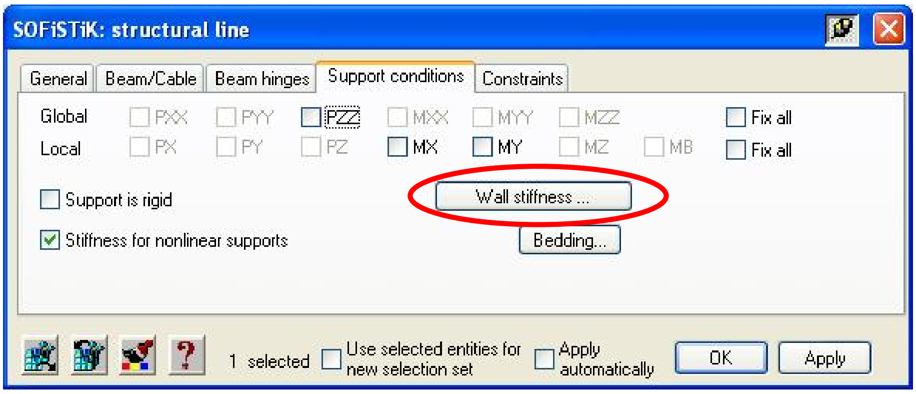
SOF_PM_EDGE_M

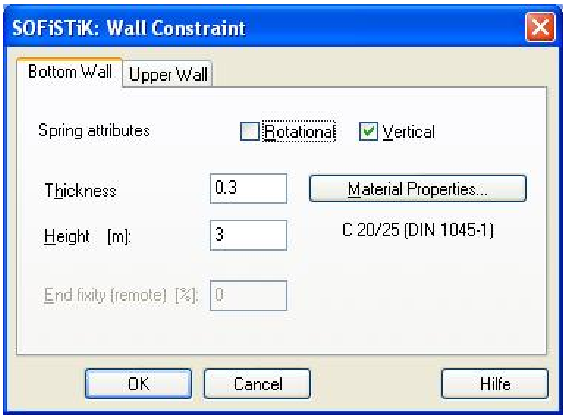


Finally the wall supports (A'/1-2 and 2/a-A') have to be defined. Select the command "structural line" and modify the wall as shown below. The support width of 0.3 m is important for the following punching design.



We recommend using nonlinear supports with a wall stiffness calculated from the wall thickness 0.30 m and the floor height 3.0 m. Use the button "Wall stiffness …". Confirm the input with OK.





Now the graphical input is finished, see Figure 20.

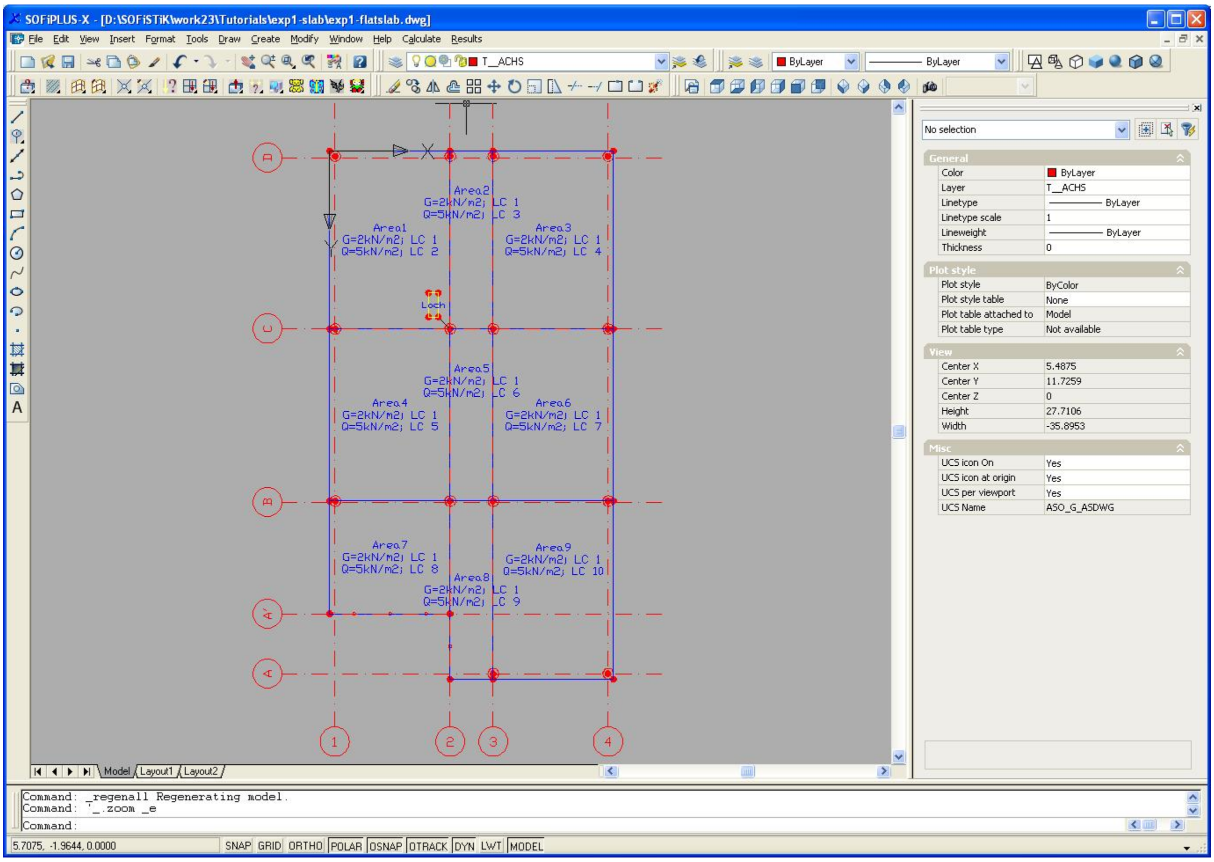


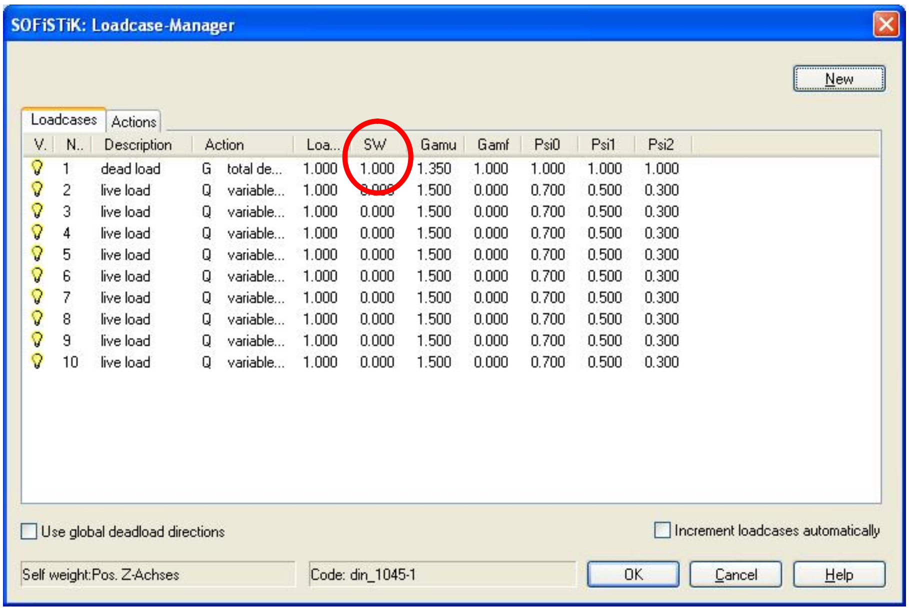
Figure 20: Complete slab system

Before starting the automatic mesh generation we recommend checking the actions and loads.

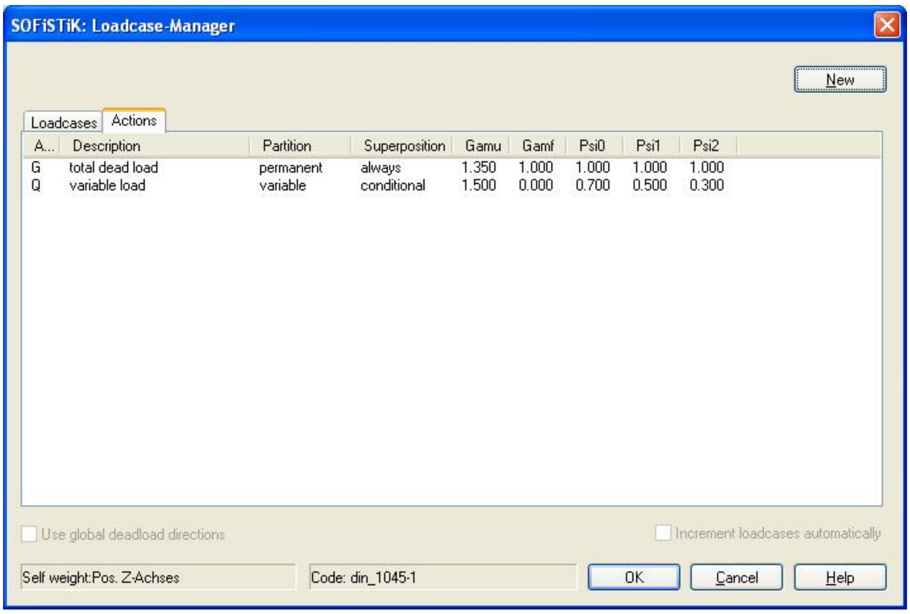
To start the loadcase manager use the button . After the dialog is open you will see the tab loadcases. There are 10 loadcases listed in the table. The 1st loadcase is the dead load. The program is able to calculate automatically the self weight of the construction. To do so the factor in the row SW has to be changed to 1.0 for LC 1.

Change now to the tab actions. There are two actions defined, total dead load and variable load., which is correct for this problem. Beside the action name the correct safety and combination values according to German code DIN 1045-1 are listed.

SOF_glfmod Click on the button to start the loadcase manager



Listing of loadcases



Listing of actions

After these checks the system definition is finished and a system export into the central database is necessary. This will be done by the command button .

Now save your drawing and change back to the main SSD window, where the system will be shown in the ANIMATOR.

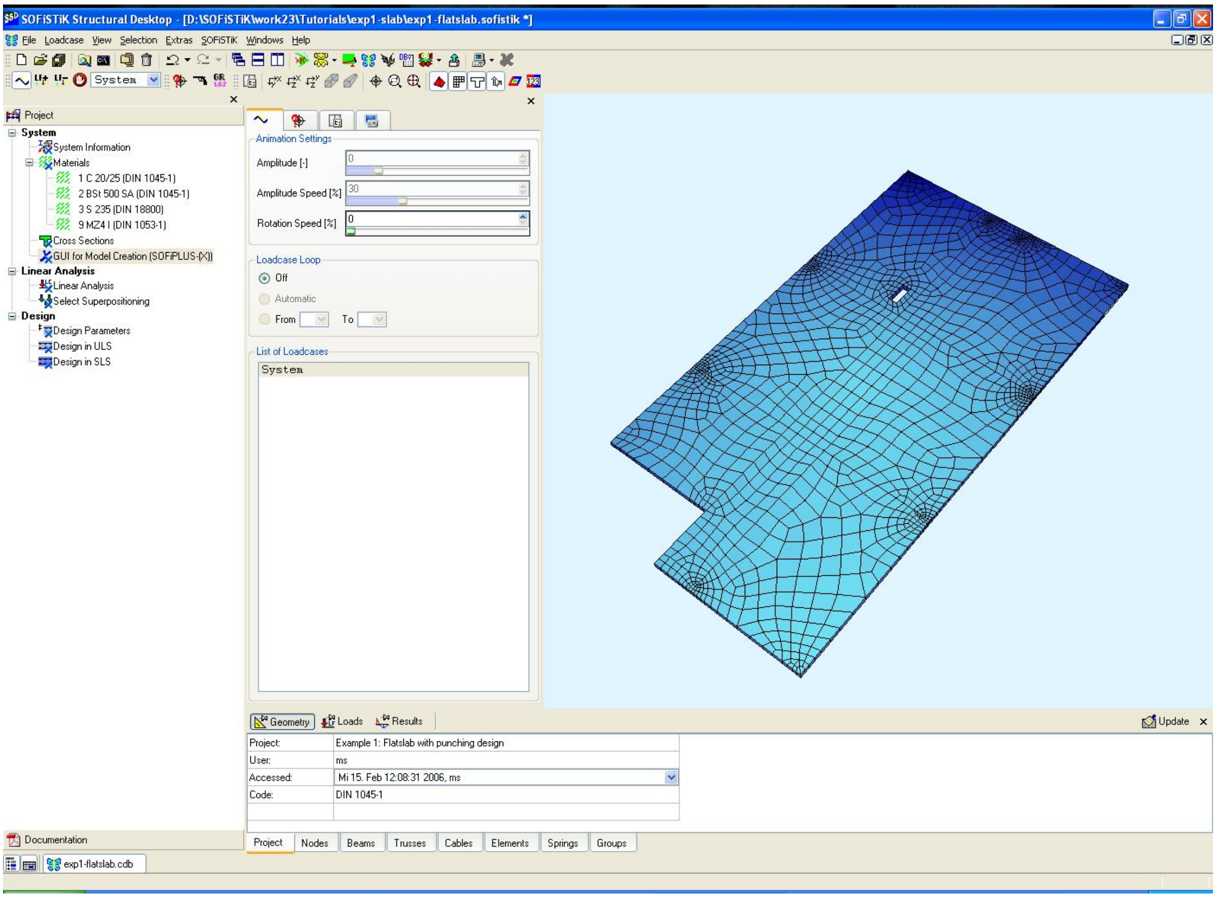


Figure 21: ANIMATOR showing complete slab system

The next step is to do the linear analysis for every single loadcase and calculate the relevant combinations.

SOFiPLUS is able to import any drawing with DWG or DXF format. Therefore architectural drawings can be used for the system generation

4.5 Step 4: Analysis and Combinations

To start the linear analysis, open this task with a double click. By default all existing loadcases are selected. Normally there is no need for a manual selection.

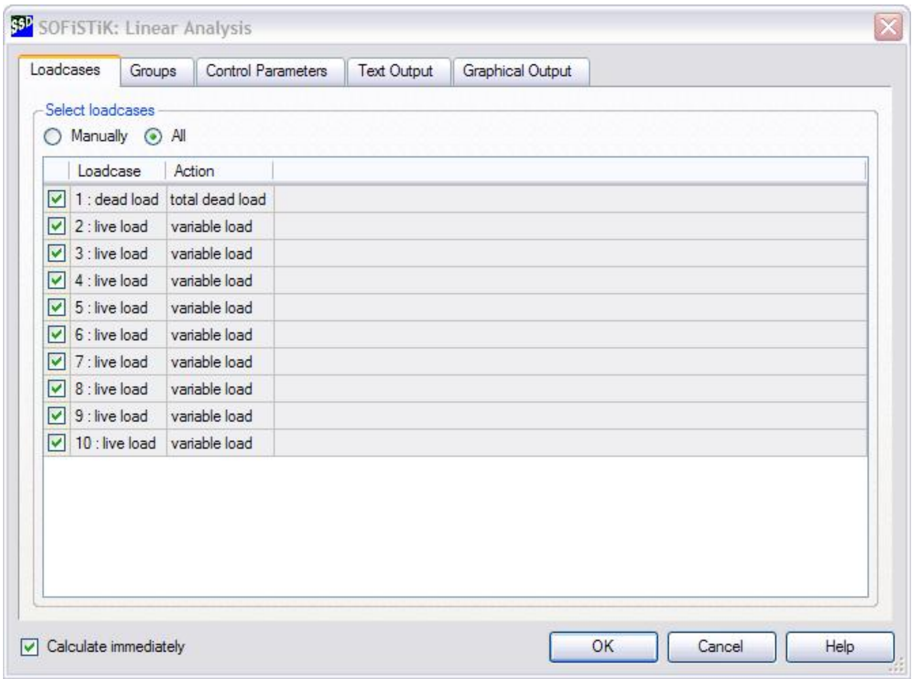


Figure 22: Dialog Linear Analysis – Loadcases Tab

By default all loadcases are selected, which is normally sufficient.

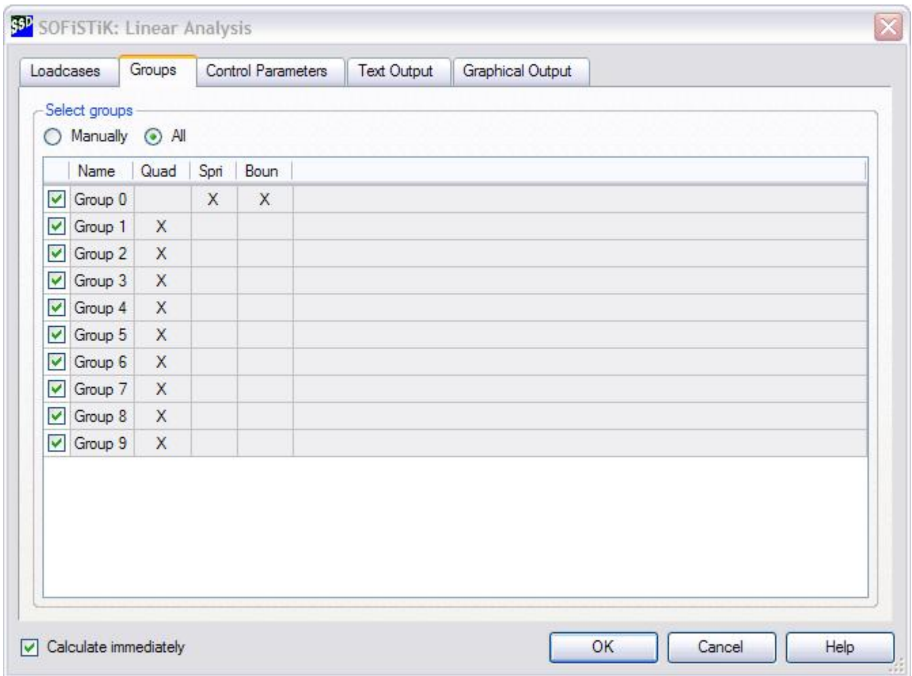


Figure 23: Dialog Linear Analysis – Groups Tab

Group selection is important. By default all groups will be selected for a calculation. Deselected groups are not considered during the calculation. This facility can be used to simulate temporary support conditions during construction. The first loadcases are calculated with the temporary supports. In the last loadcase, deselect the group containing the temporary supports for them to be ignored.

In the case where groups of elements are active and passive for different loadcases it is necessary to define multiple “Linear Analysis” tasks.
It is very important to check the loads because loads on deselected elements will cause an error message during the calculation.

Control Parameters are not necessary. The register “Text Output” is responsible for the output and documentation. The output volume is variable for the different output chapters. Also the default output is sufficient in most cases.

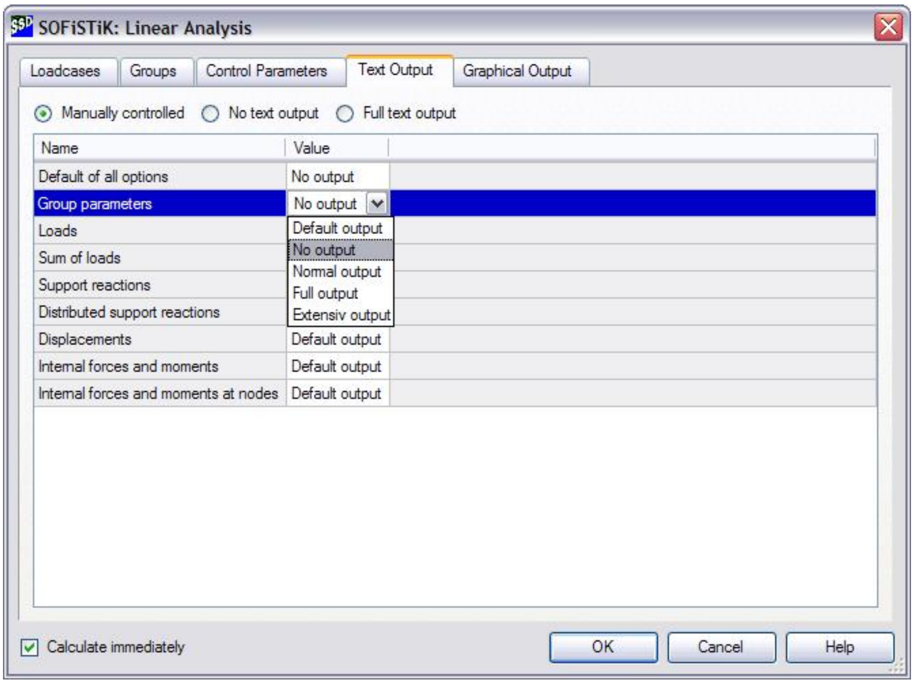


Figure 24: Dialog Linear Analysis – Text Output Tab

Besides the text output a graphical output is also available. There are default settings for standard graphics. To add these graphics to the output just select or deselect the entries from the tree structure.

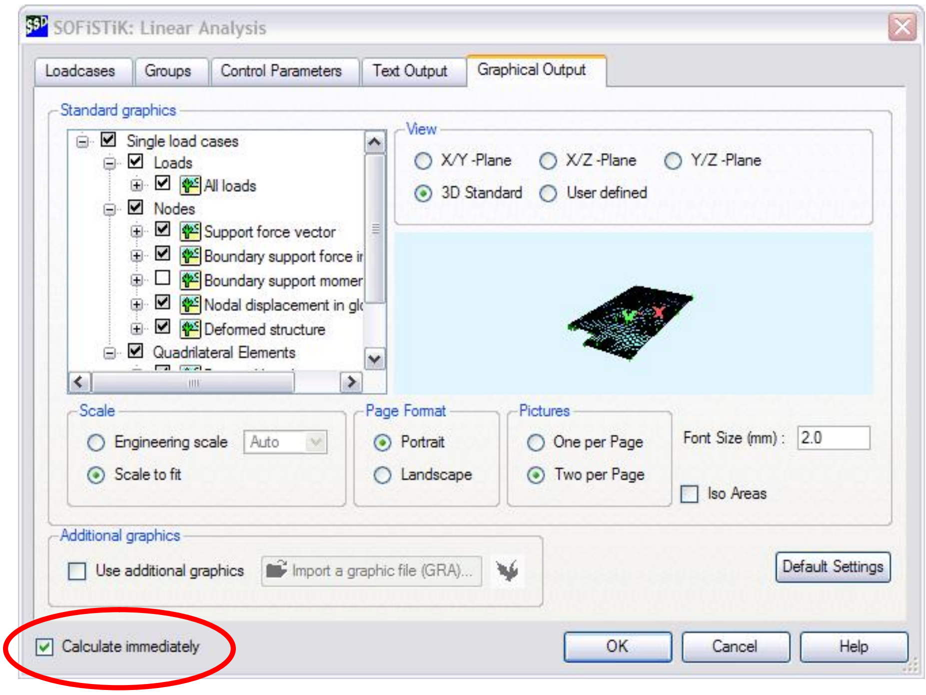


Figure 25: Dialog Linear Analysis – Graphical Output Tab

Important to note is the ease with which a user defined view can be created. Select the view option “User defined” and click into the view window and move the system inside the 3D-orbit.

The import of additional graphic files created with WINGRAF is also possible. Select the option “Use additional graphics” and start browsing . The other way is to start WINGRAF with the button. The graphics you define inside WINGRAF will be transferred to the SSD.

Confirm all settings with the OK button.

In the bottom left-hand corner you will find the preselected option “Calculate immediately”. Confirming with OK button will automatically start the calculation.

With the ‘Calculate immediately’ selected,a new program is opened within the SSD main window. On the left-hand side, there is a tree structure, which shows all tasks. Only the two program blocks SEPP and WING are active for this calculation, which is shown by the signs .

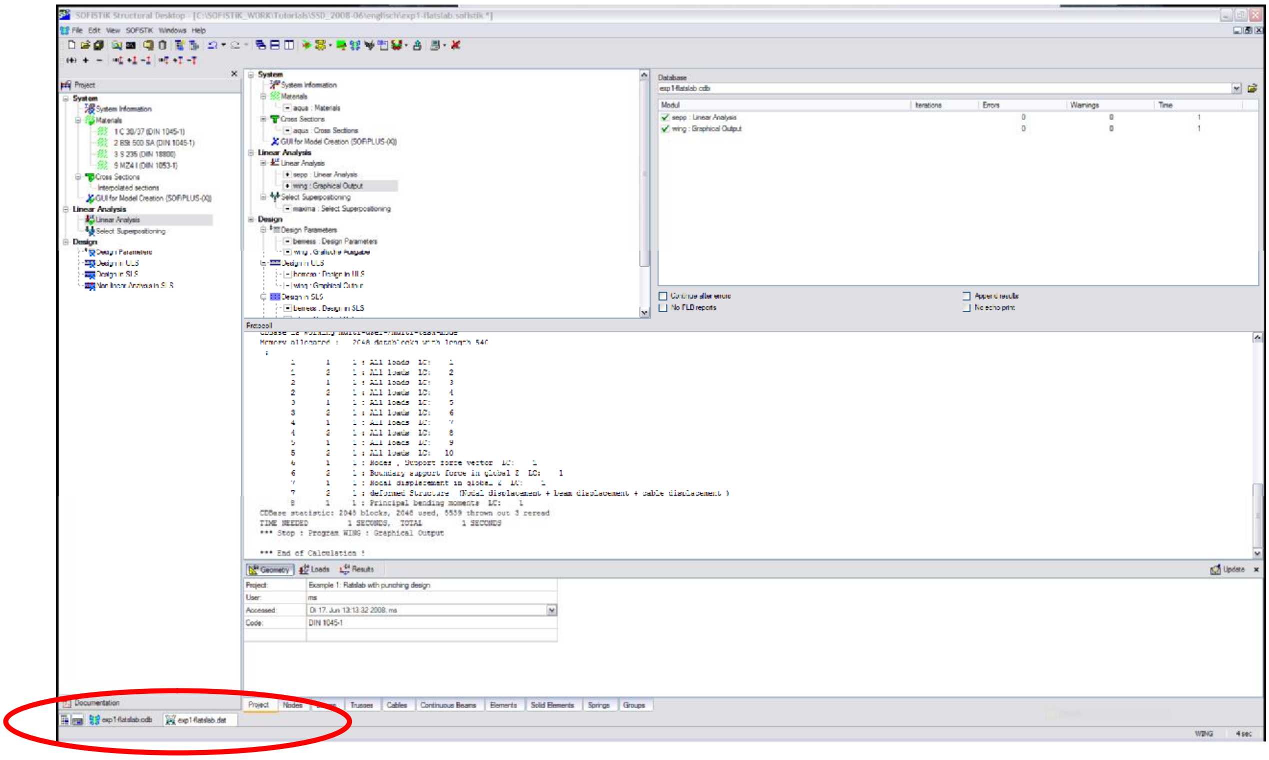


Figure 26: SSD main window - calculation overview

To change the programs use the task bar at the bottom.

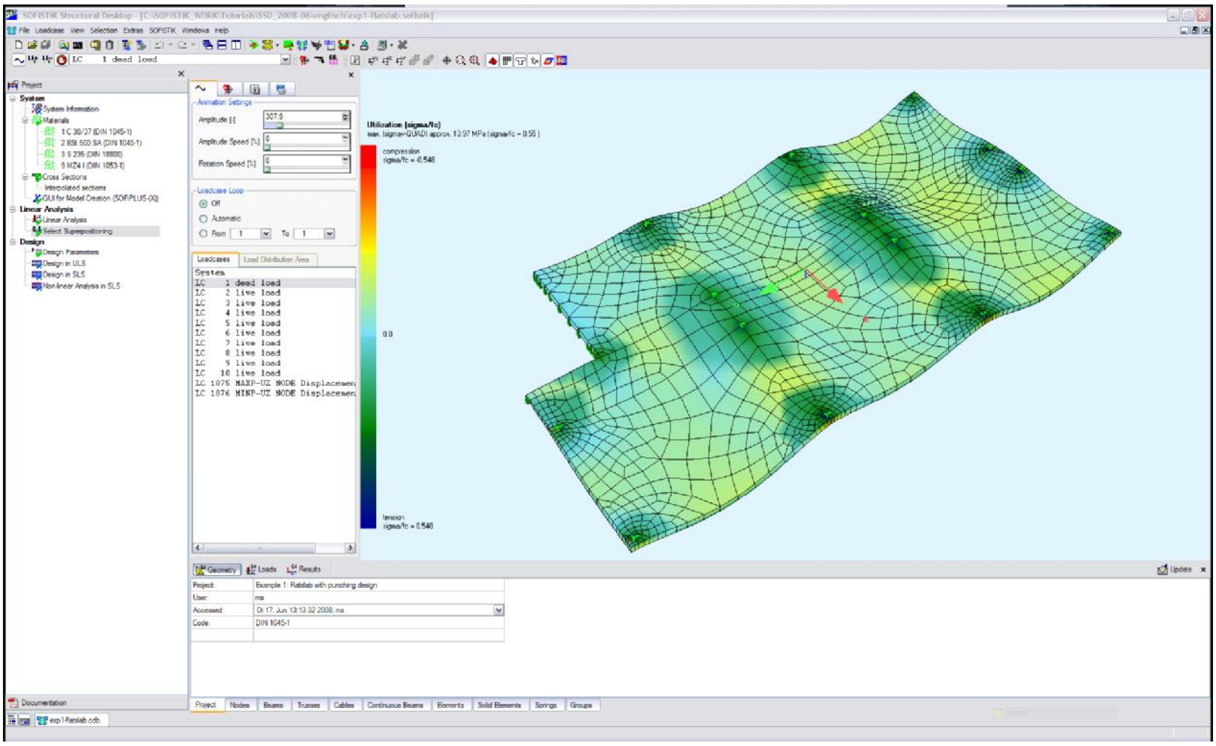


Figure 27: SSD main window - ANIMATOR view of LC 1

These basic proceedings will be done now for the Superpositioning

> The task “Select Superpositioning” is a selection only. To create new superpositions a new task “Define Superpositioning” has to be added.
>
> Always use the task “Select Superpositioning” after the task “Define Superpositioning” to make sure all superpositions are available and can be processed by the program.

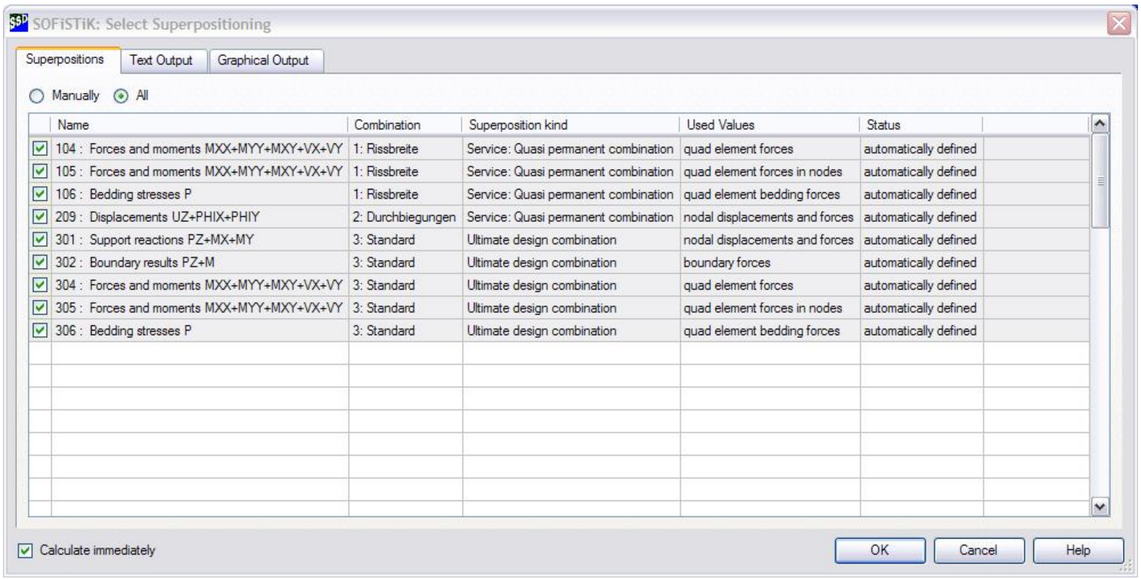


Figure 28: Dialog Select Superpositioning – Superpositions Tab

The text and graphical output can be selected as shown before. In addition, the option “Calculate immediately” works in the same way.

4.6 Step 5: Design

By default three design tasks according to the design code were generated automatically. The tasks are – Design parameters – Design in ULS – Design in SLS -

4.6.1 Design Parameters

The design parameters allow the user to define specific reinforcement for every group of elements. There are four different reinforcement types available: two layers – orthogonal, two layers – non orthogonal, three layers and circular.

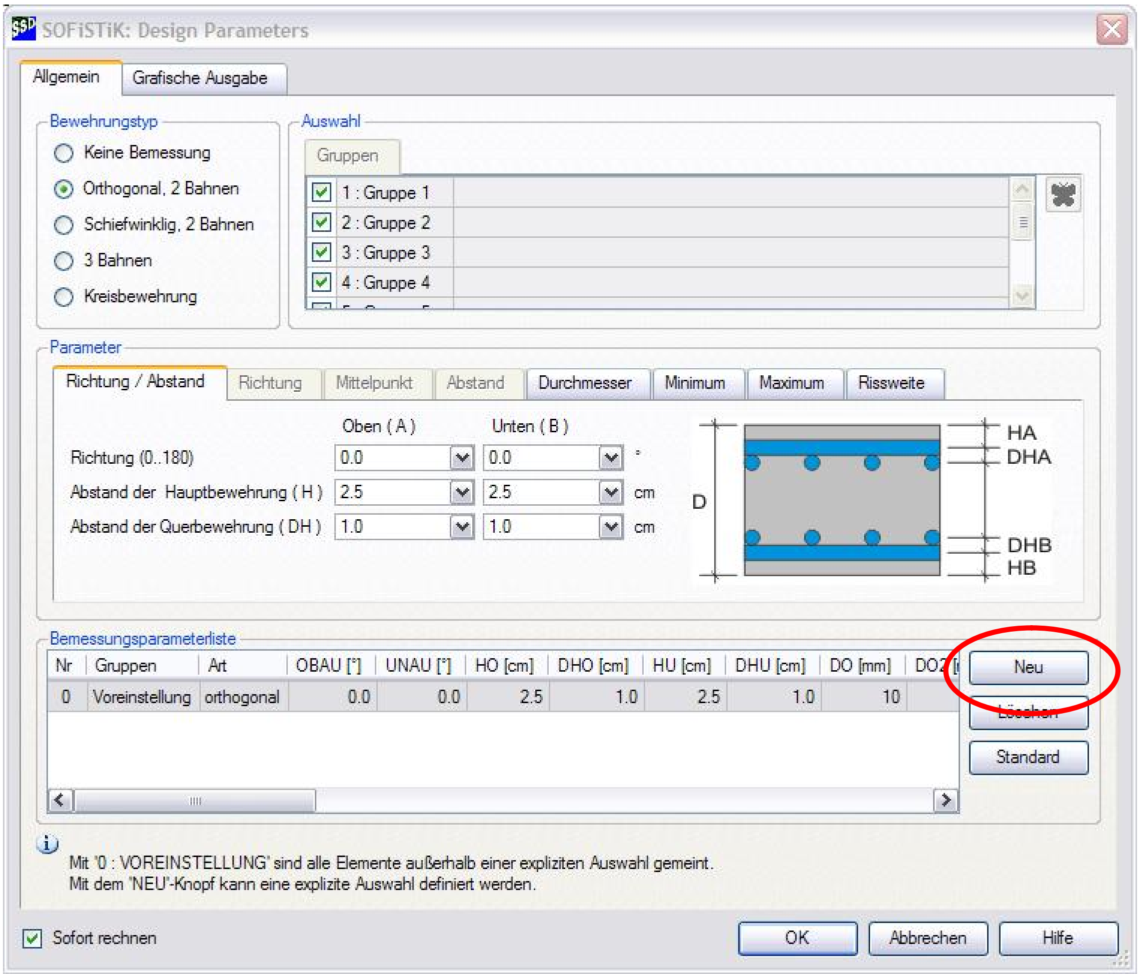


Figure 29: Dialog Design Parameters – Register Common

In most cases, two-layer orthogonal reinforcement is suitable for the design. In this dialog, the main direction and the distances between centreline of reinforcement bars and concrete surfaces can be edited. If reinforcement has to be changed for selected groups, create a new line in the Design parameter list using the button NEW.

Then select the relevant groups from the selection list. For example, select groups 1 to 3 and define a two layer non orthogonal reinforcement with an angle of 45°. The settings for the other groups will remain the same as shown in Figure 30.

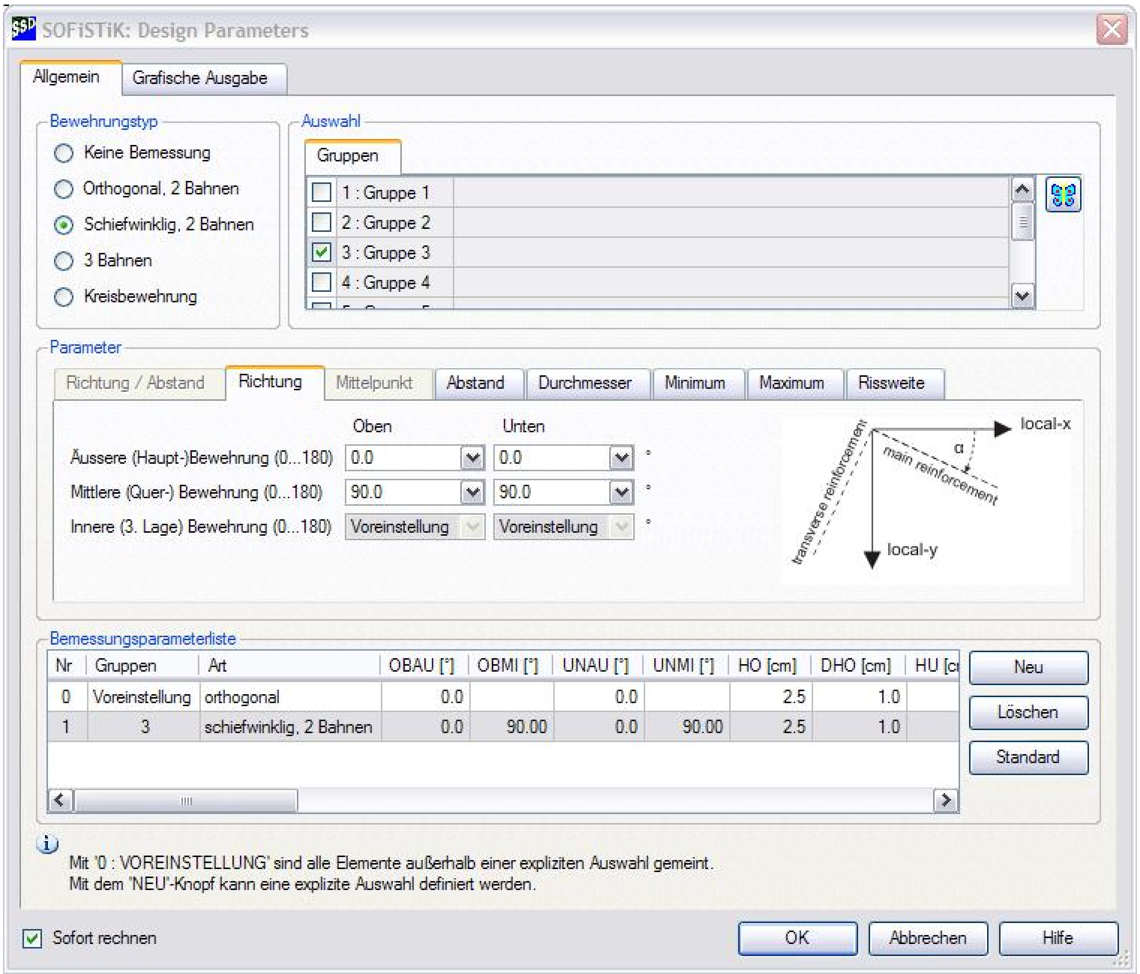


Figure 30: Dialog Design Parameters – Direction Tab

> This example is only to show the input procedure and will not be used in the following calculation. Therefore please delete this input with the button DELETE.

The other parameters are basically used by the serviceability design.

Diameter Tab:	is used for crack width calculation
Minimum Tab:	predefined minimum reinforcement for non-linear analysis
Maximum Tab:	predefined maximum reinforcement for non-linear analysis
Crack Width Tab:	crack width input to determine reinforcement stresses

The definitions made here will be used in all following design tasks.

4.6.2 Design in ULS

The design in ultimate limit state (ULS) is controlled it's own task. Generally, (and for the example) no further input is necessary.
For information, the following input procedures are described as follows.

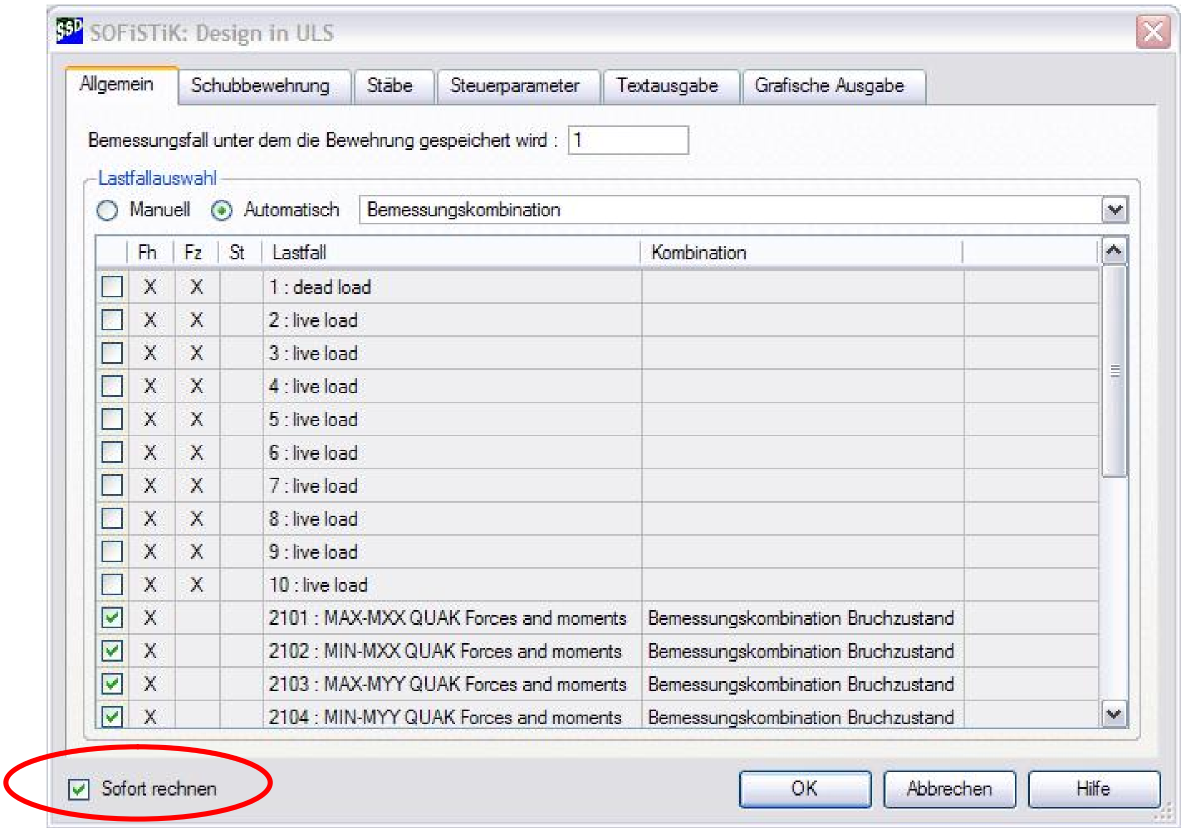


Figure 31: Dialog Design in ULS – General Tab

The superposition results are loadcases with different identifiers. As shown in Figure 31 there are the single loadcases (LC 1 – LC 10) and loadcases 21xx from the ultimate design combination. The program automatically selects all the loadcases for results from the ultimate design combination. The column **Am** indicates loadcases with results for area elements, the column **Aa** indicates loadcases with additional results for area elements and the column **Be** indicates loadcases with results for beam elements. In this example, we don't have any beam elements, so the column is clear, which is correct.

For the shear and punching design go to the "Shear Reinforcement" tab. The program will increase automatically the bending reinforcement to pass the shear and punching checks. You may define a maximum percentage as an upper limitation for the bending reinforcement before shear links are required. There is a control over the shear reinforcement design both inside and outside the punching area. No increase of reinforcement is allowed in this example.

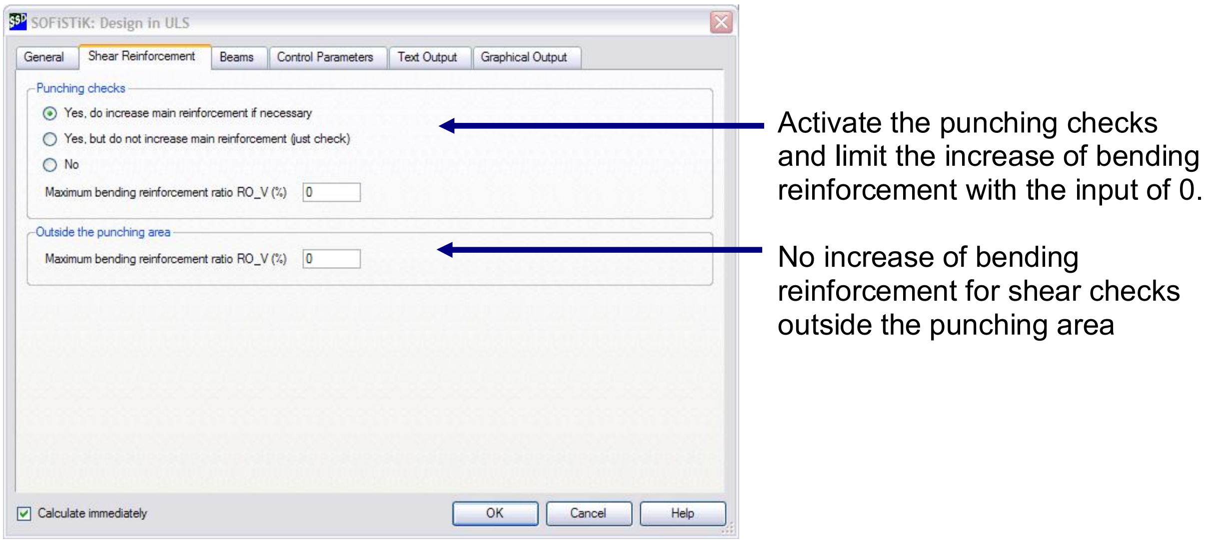


Figure 32: Dialog Design in ULS – Shear Reinforcement Tab

Using the Beams tab you may define the necessary design settings for beam elements. If there are no beams in the system, as in this example, no input is required.

There are various control parameters which can be set depending on the design codes. For further information we refer you to the national codes and the technical literature.

As with all other tasks, the text and graphical output can be selected.

The calculation starts immediately if the ‘Calculate immediately’ check box is ticked. Please deactivate the option “Calculate immediately” and after checking all input, close the input with OK. Now you will see the sign in the task tree. This means, that the task has a new input and is still open for calculation. (see also Figure 10)

Before starting the calculation you may wish to check the CADINP input data, created automatically.

Every task automatically creates a CADINP input file, which is part of the complete input file <name>.dat. To check this input file use the text editor TEDDY. To open the input file click on the task inside the task tree and use the right mouse click – EDIT. Now the TEDDY will be opened.

The TEDDY input file is shown below.

```
$ Automatically generated by DesignULS V(1.00-23) 15.02.2006 12:39:08
$ Attention: Changes will be overwritten if the task is opened again!
+PROG BEMESS urs:11.1 $ Design in ULS
HEAD ULS design

CTRL LCR 1 $ Reinforcement distribution number
CTRL RO_V 0 $ Maximum reinforcement for shear for normal slab region

PUNC YES RO_V 0

LC DESI

END
```

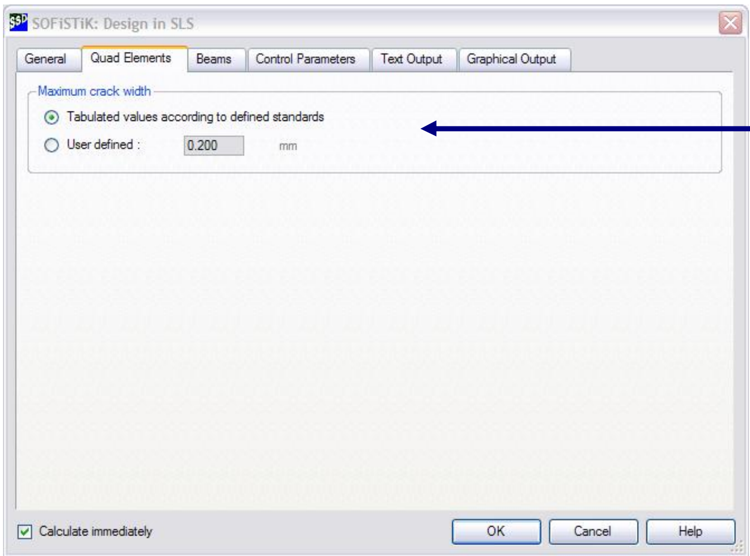
The functionality available within TEDDY is very powerful and all can be access within the SSD. Additional text input can be entered to amend or extend that which has been created with the dialog input. There are numerous possibilities, which are not discussed here. For further information please look in the handbook SOFiSTiK_1.pdf.

Before going on with the “Design in SLS” the calculation “Design in ULS” has to be finished because reinforcement areas are necessary for SLS input.
Also note, that changes within the dialogs will change the CADINP input file.

4.6.3 Design in SLS

According to the German code DIN 1045-1 the design in Serviceability Limit State is more important today. The task “Design in SLS” starts a design for the maximum crack width.

Again the relevant loadcases are selected automatically for the design. A manual selection is not normally necessary.



Crack width is calculated either due to tabulated values or due to user defined input.

Normally the design is done by DIN 1045-1 Table 20. In case there are some limiting stresses added in the task “Design Parameters” the design will be done by Table 21. In case bar diameters and stresses are defined the design follows again Table 21.

Figure 33: Dialog Design in SLS – Quad Elements Tab

4.7 Documentation

The documentation is divided into several URSULA reports. After every calculation a task creates it's own URSULA report. For the final documentation go to the button and select "all reports". URSULA will open automatically and displays the complete report.

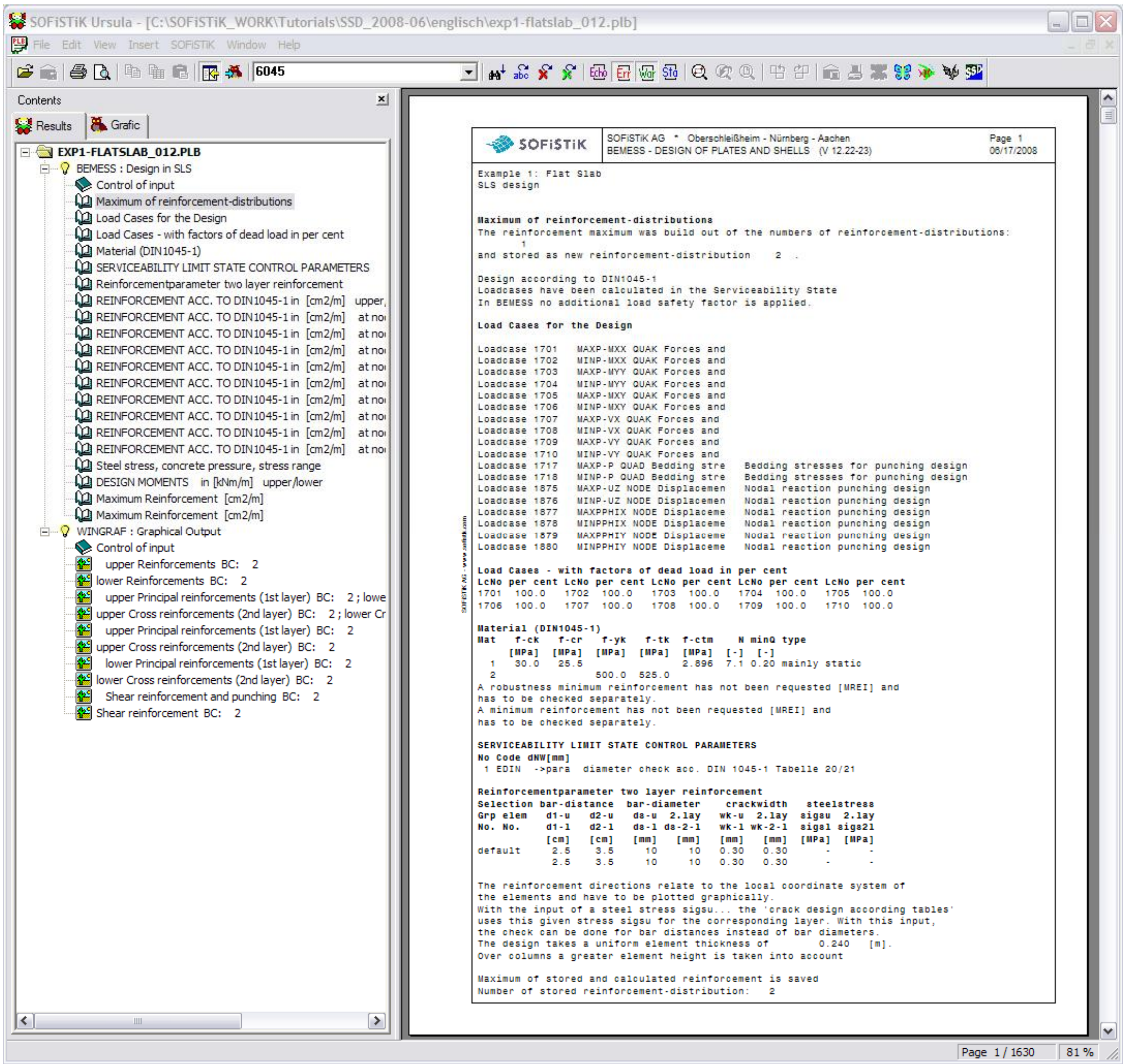


Figure 34: URSULA Result browser

All chapters are printed in a tree structure on the left-hand side of the URSULA screen. Normally every chapter is activated except the "Control of input" chapters. To activate or deactivate every chapter just click on the chapter title within the tree.

There is an easy way to add a complete list of contents, by using the top menu Insert – Insert List of Contents.

4.8 Discussion of results

4.8.1 Structural Analysis

A complete validation of all internal forces is not discussed here. The object here is to compare the relevant support forces and moments for the following punching check.

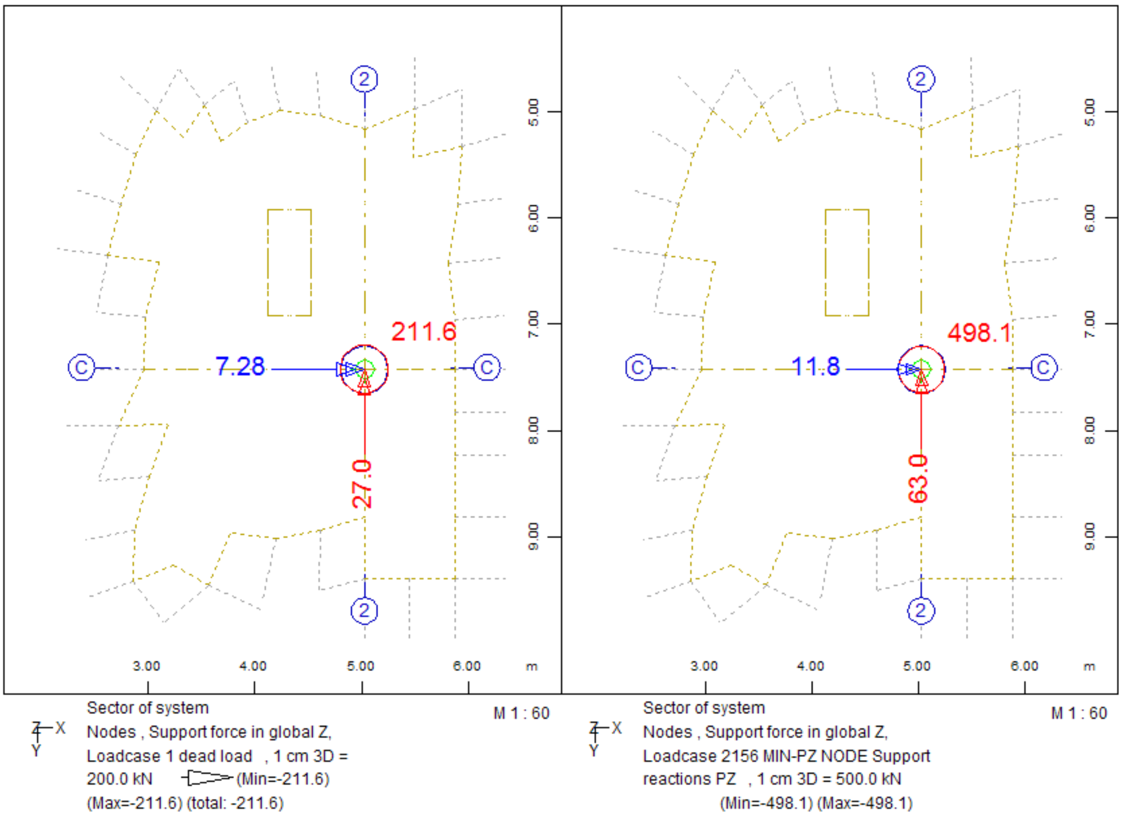


Figure 35: Support forces and moments at node 12 LC 1 and LC 2156

	Betonkalender 2006, page 183, Figure 101			SOFiSTiK SSD	
node12 (S1)	dead load	live load	Max P_{zed} 1)	LC 1: G	Max P_{zed}
R_z [kN]	213	149	511,05	-211,6	-498,1
M_x [kNm]	-7,6	-1,15	-11,99	7,28	11,8
M_y [kNm]	28,8	16,9	64,23	-27,0	-63,0

1) P_{zed} = 1,35 G +1,50 Q

Table 1: Overview results from literature and out of SSD calculation

As shown in Table 1 the results compare very closely with the reference example. The only difference is the signs, which is caused by the different coordinate systems used in literature and SOFiSTiK SSD.

4.8.2 Punching Design

To check the results we recommend using the extended text output for punching design. The output volume can be changed in task “Design in ULS” – Text output tab.

For detailed comparison between the results from the reference literature and from the SOFiSTiK calculation we want to look at the punching checks of node 3 (wall corner) and node 12 (inner column near hole). The detailed text output is printed below:-

```
Punching Design (DIN1045-1)
Node number            =     3           X= 5.025 [m]       Y= 19.33 [m]
Max. shear force V-ULS= 246.8 [kN]      LC=  2102  via QUAD connecting forces
Integrated from boundary reactions over a length of 2*0.294 [m]
Wall corner           a= 0.294 [m]
The active wall length a has been set to 1.4*depth. Wall thickness is not used.
Plate thickness h-slab= 0.240 [m]    depth 0.210 [m]
1. perimeter at  1.5*d= 0.315 [m]                      ucrit= 1.227 [m]
   (u= 39 % of utot due to openings, edges or walls)
Tension reinfor.  as >= 14.89 [cm2/m] mue=  0.71 [o/o]  VRdct 160.9 [kN/m]
mue necessary to satisfy von vRD,max acc. DIN 1045-1 equation 107!
 v-Ed = 1.20*V/ucrit  = 241.4 [kN/m]     > 160.9 [kN/m] =Vrdct
        1.20=sweeping excentricityfaktor beta
Beta value end of walls/corners acc. Normenausschuß Bau [NABAU] Lfd 233 10.5.2
1. design cut of shear reinforcement at point 0.5d -> u= 0.714 [m]
Shear reinforcem.  Ass= (V-Ed*ucrit/u-VRdc)*u/fyd/kappa-s (kappa-s=0.70)
Shear reinforcem.  Ass=  5.96 [cm2]   ass= 52.99 [cm2/m2]
    to be provided in the 1. perimeter up to columnedge    +  0.184 [m]
    2. perimeter   Ass=  2.94 [cm2]   ass= 17.00 [cm2/m2] til 0.341 [m]
                   Ass= (V-Ed(u)-VRdc)*u*sw/d/fyd/kappa-s
    3. perimeter   Ass=  1.42 [cm2]   ass=  6.06 [cm2/m2] til 0.499 [m]
In the critical punching zone at least     14.89 [cm2/m]
tension reinforcement is required
.

Node number            =    12           X= 5.025 [m]       Y= 7.425 [m]
Max. shear force V-ULS= 517.8 [kN]      LC=  2104  via QUAD connecting forces
Circular column      dS= 0.450 [m]
Plate thickness h-slab= 0.240 [m]    depth 0.210 [m]
1. perimeter at  1.5*d= 0.315 [m]    utot= 3.393 [m]   ucrit= 3.016 [m]
   (u= 89 % of utot due to openings, edges or walls-> edge column)
Min.reinforc. as-upper= 16.08 [cm2/m]  (Min.design-moment-> edge column)
Min.reinforc  as-lower=  7.52 [cm2/m]  (Min.design-moment-> edge column)
Tension reinfor.  as >= 16.08 [cm2/m] mue=  0.77 [o/o]  VRdct 165.1 [kN/m]
mue necessary to satisfy von vRD,max acc. DIN 1045-1 equation 107!
 v-Ed = 1.20*V/ucrit  = 206.0 [kN/m]     > 165.1 [kN/m] =Vrdct
        1.20=sweeping excentricityfaktor beta
1. design cut of shear reinforcement at point 0.5d -> u= 1.843 [m]
Shear reinforcem.  Ass= (V-Ed*ucrit/u-VRdc)*u/fyd/kappa-s (kappa-s=0.70)
Shear reinforcem.  Ass= 10.41 [cm2]   ass= 35.87 [cm2/m2]
    to be provided in the 1. perimeter up to columnedge    +  0.184 [m]
    2. perimeter   Ass=  4.23 [cm2]   ass=  9.86 [cm2/m2] til 0.341 [m]
                   Ass= (V-Ed(u)-VRdc)*u*sw/d/fyd/kappa-s
In the critical punching zone at least     16.08 [cm2/m]
tension reinforcement is required
```

In addition to the extended output a short table “Conclusions” is also printed in the output. Important are the additional hints written below this listing. As you can see, the Column at node 12 will be treated as an edge column because of the adjacent opening Therefore, node 12 is marked with Typ E = edge column.

```
Punching Design (DIN1045-1)
CONCLUSION
NodeNo Typ     X       Y  V-ULS  d-col ucrit  =%u0  v-max AssSum     ast nperi
    No        [m]     [m]   [kN]    [m]    [m] [o/o]  [MPa]  [cm2] [cm2/m]
     3 L   5.025 19.325  246.8 0.332 1.227    39   1.15  10.32   14.89     3
    12 E   5.025  7.425  517.8 0.450 3.016    89   0.98  14.64   16.08     2
  Typ  I=inner column, E=edge column, C=corner column, F=foundation,
       W=end of wall,  L=wall corner, G=end_of_girder
  ucrit =effective length of 1. perimeter, reduced due to openings and edges
  %u0   =reduktionfactor due to openings and free edges = u0/u0-tot in %
  AssSum=shear reinforcement - total sum of all nperi perimeters
  ast   = min. required tension reinforcement in the punching zone
  nperi =up to this perimeter, shear reinforcement is required
  Both pressure and tension results are taken into account.
Minimim design moments and collapse reinforcement are taken into account.
```

4.8.2.1 *Punching inner column, node 12*

	Literature	SOFiSTiK
Effective height [cm]	20	21
Diameter critical perimeter d_{krit} [m]	1,05	1,08
Reduction due to opening δ [-]	0,8819	0,89
Critical perimeter u_{krit} [m]	2,91	3,016
factor β	1,29	1,20
Shear force v_{ed} [kN/m]	227	206,0
shear capacity without reinforcement $v_{Rd,ct}$ [kN/m]	190,0	165,1
Shear reinforcement 1. perimeter [cm²] distance 0,5d=10,5 cm up to 0,5d+sw/2=18,4 cm	**10,40**	**10.41**
Design cut u_1 [m]	1,80	1,843
Shear reinforcement 2. perimeter [cm²] distance 18,4 cm up to $0{,}5d + 3 \cdot sw/2 = 10{,}5 + 3 \cdot 15{,}75 / 2 = 34{,}1$ cm	**3,95**	**4,23**
Design cut u_2 [m]	2,63	2,72

Table 2: Comparison of results from literature and from SOFiSTiK calculation at node 12

The results compare very well. The slight difference is caused by different effective heights, different bending reinforcement and different factor β which takes into account the opening.

Reducing the critical perimeter near inner columns by up to 80%, the factor β is considered with 1,20 instead of 1,05 (DIN 1045-1, Figure 44). The column will be treated as type E = edge column.

4.8.2.2 *Punching check at wall corner, node 3*

The punching design on wall corners must be treated in a special way.

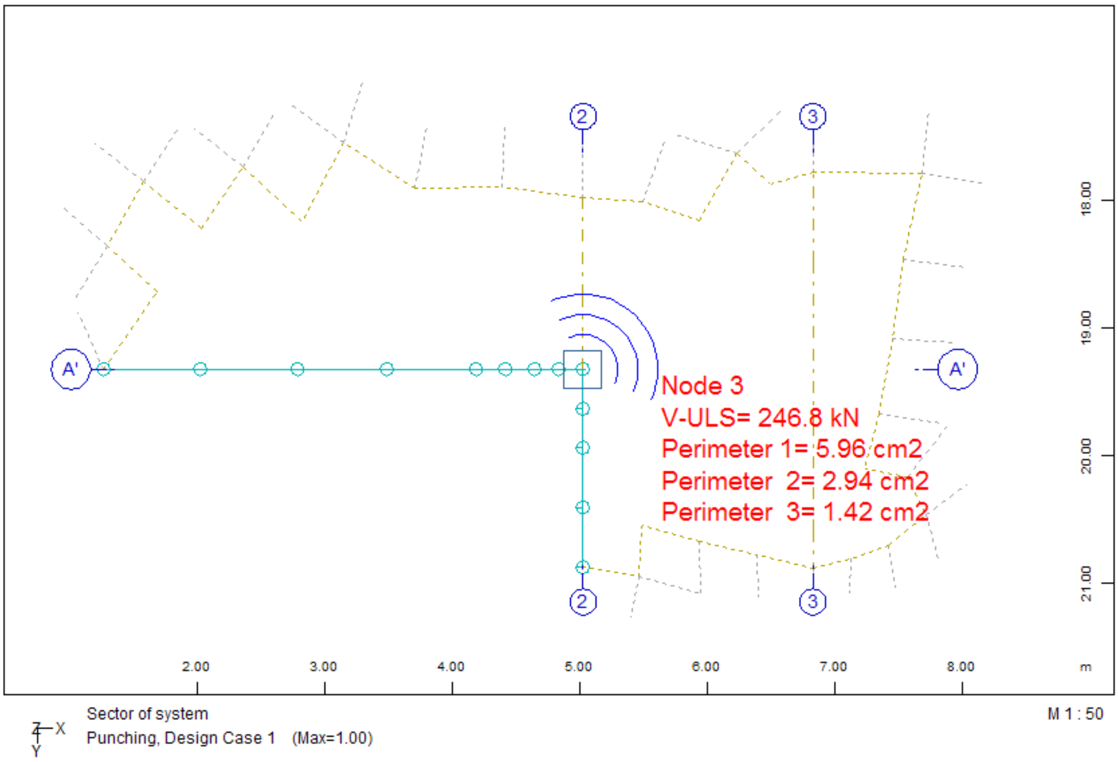


Figure 36: WINGRAF Figure- punching check at node 3

For further description we refer to our BEMESS manual Chapter 2.4.6.

4.8.3 Serviceability

The results of the serviceability check are also printed in the URSULA output. The output is organized in several chapters. The reinforcement from the ultimate limit state design is saved in a reinforcement loadcase, which will be used now. Next is a list of loadcases which will be used for the design.

```
Maximum of reinforcement-distributions
The reinforcement maximum was build out of the numbers of reinforcement-
distributions:
       1
and stored as new reinforcement-distribution    2  .

Design according to DIN1045-1
Loadcases have been calculated in the Serviceability State
In BEMESS no additional load safety factor is applied.

Load Cases for the Design

Loadcase 1701   MAXP-MXX Forces and mome
Loadcase 1702   MINP-MXX Forces and mome
Loadcase 1703   MAXP-MYY Forces and mome
Loadcase 1704   MINP-MYY Forces and mome
Loadcase 1705   MAXP-MXY Forces and mome
Loadcase 1706   MINP-MXY Forces and mome
Loadcase 1707   MAXP-VX  Forces and mome
Loadcase 1708   MINP-VX  Forces and mome
Loadcase 1709   MAXP-VY  Forces and mome
Loadcase 1710   MINP-VY  Forces and mome
Loadcase 1717   MAXP-P   Bedding stresse   Bedding stresses for punching design
Loadcase 1718   MINP-P   Bedding stresse   Bedding stresses for punching design
Loadcase 1875   MAXP-UZ  Displacements     Nodal reaction punching design
Loadcase 1876   MINP-UZ  Displacements     Nodal reaction punching design
Loadcase 1877   MAXPPHIX Displacements     Nodal reaction punching design
Loadcase 1878   MINPPHIX Displacements     Nodal reaction punching design
Loadcase 1879   MAXPPHIY Displacements     Nodal reaction punching design
Loadcase 1880   MINPPHIY Displacements     Nodal reaction punching design

Load Cases - with factors of dead load in per cent
LcNo per cent LcNo per cent LcNo per cent LcNo per cent LcNo per cent
1701  100.0   1702  100.0   1703  100.0   1704  100.0   1705  100.0
1706  100.0   1707  100.0   1708  100.0   1709  100.0   1710  100.0
```

Important are the material values and the safety factors.

```
Material (DIN1045-1)
Mat   f-ck   f-cr   f-yk    f-tk Param.  f-ctm    N minQ type
     [MPa]  [MPa]  [MPa]  [MPa]  [MPa]  [-]  [-]
  1   30.0   25.5                 2.896  6.3 0.20 mainly static
  2                500.0  525.0
A robustness minimum reinforcement has not been requested [MREI] and
has to be checked separately.
A minimum reinforcement has not been requested [MREI] and
has to be checked separately.
```

The serviceability limit state control parameters are very important to check. The report shows the used design code and the maximum allowed crack width.

```
SERVICEABILITY LIMIT STATE CONTROL PARAMETERS
No Code dNW[mm]
 1 EDIN  ->para  diameter check acc. DIN 1045-1 Tabelle 20/21

Reinforcement parameter two layer reinforcement
Selection bar-distance  bar-diameter    crackwidth   steelstress
Grp elem   d1-u   d2-u   ds-u  2.lay   wk-u  2.lay  sigsu  2.lay
No. No.    d1-l   d2-l   ds-l ds-2-l   wk-l wk-2-l  sigsl sigs2l
           [cm]   [cm]   [mm]   [mm]   [mm]   [mm]  [MPa]  [MPa]
default     2.5    3.5     10     10   0.30   0.30      -      -
            2.5    3.5     10     10   0.30   0.30      -      -

The reinforcement directions relate to the local coordinate system of
the elements and have to be plotted graphically.
With the input of a steel stress sigsu... the 'crack design according tables'
uses this given stress sigsu for the corresponding layer. With this input,
the check can be done for bar distances instead of bar diameters.
The design takes a uniform element thickness of        24.0  [cm].
Over columns a greater element height is taken into account
Maximum of stored and calculated reinforcement is saved
Number of stored reinforcement-distribution:   2
```

After this basic information the design output is listed. For example the element 10001 passed the crack check for loadcase 1701. The output GRZD means that the maximum bar diameter is smaller than allowed according to DIN 1045-1, Table 20.

```
REINFORCEMENT ACC. TO DIN1045-1 in  [cm2/m]   upper/lower
General load safety factor - as defined in BEMESS:  Gamma-f = 1.00
Shear: stresses VEd/d and VRd,ct/d  with d=effective depth = h-hm
Shear index 2m = minimum shear reinforcement
  ELEM    LC MAT GEO    h  Reinforcement    dphi Df Load Crack-   Shr  VEd/d
    No    No  No  No  [m]  main cross  dir   deg No fact check    zon  [MPa]
 10001 1701   1   1 0.24                 0        1 1.00            0
              2          1.18  2.81      0    86         GRZD            0.020
```

A complete listing for every element and every node is part of the output.

```
Maximum Reinforcement  [cm2/m]
(stored in data base file with reinforcement-distribution-no. 2)
 Element upper:As   Ast   dir  lower:As   Ast   dir Ass[cm2/m2] AssE[cm2]
   10001                    0      2.41  6.04     0
   10002                    0      2.79  6.27     0
   10003                    0      2.72  6.42     0
```

At the end of the report a reinforcement index is provided. The index is the sum of reinforcement required for every element. It does not include the additional reinforcement required, for example, lap lengths.

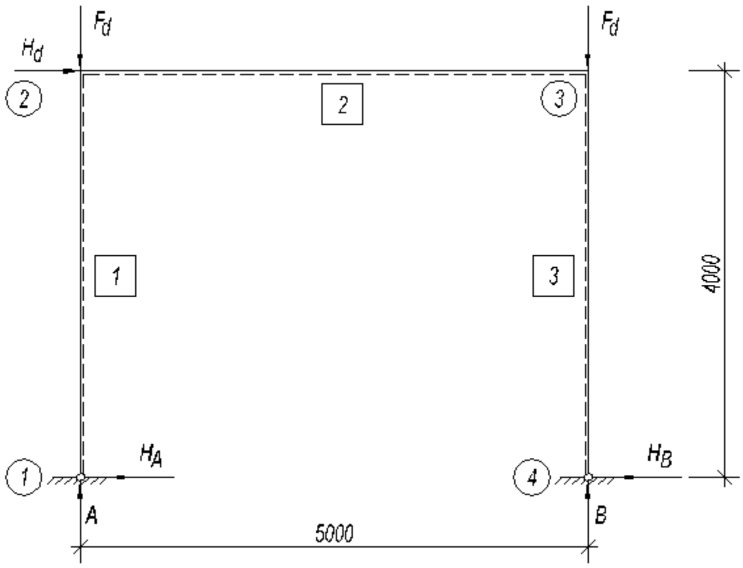
```
REINFORCEMENT INDEX [kg netto]:  1020.8 (Upper)
                                 1249.5 (Lower)
                                  328.3 (Shear)
```

The reinforcement index is the sum of the calculated reinforcement required from the finite elements in the system and does not include additional reinforcement such as lap lengths.

5 Example Steel Design

5.1 Problem

A portal frame construction from reference [3] chapter 13.4.2 will be analysed using 2nd Order Theory. In addition to the nodal loads a displacement is applied for the vertical members.



Cross section:
vertical member IPE 360
horizontal member IPE 360

Material:
S 235

Load:
vertical load F_d = 490 KN
horizontal load H_d = 30 kN
without self weight

Figure 37: portal frame

Using this example the numeric input of the system and loads will be demonstrated. The possibility of generating non-linear load combinations for 2nd Order Theory calculation and steel design also will be shown.

5.2 Step 1: Input System

Directly after starting the SSD the dialog System Information will be displayed with all the necessary input you will require to define a new project.

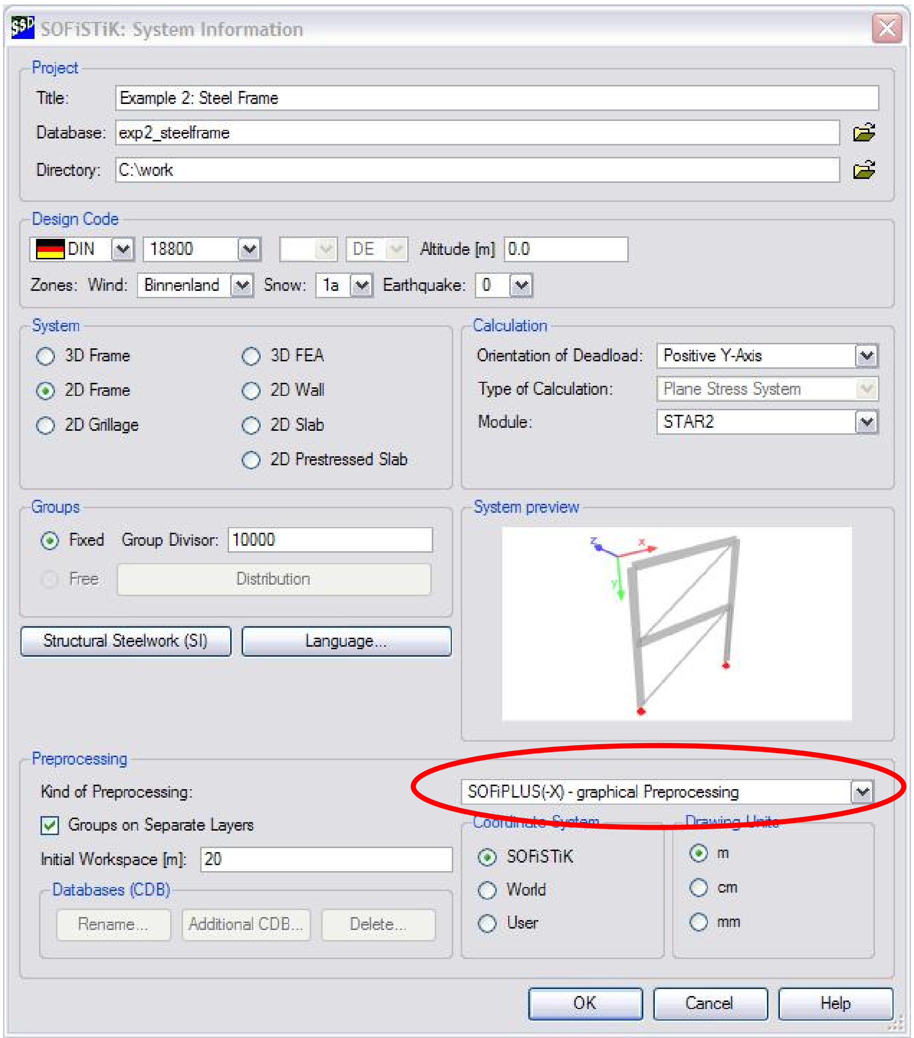


Figure 38: Starting dialog

As shown before the target is to do the analysis for a plane frame, using the German steel code DIN 18800. The program will choose the module STAR2 for this analysis.

For system and load definition the Graphical Pre-processing will be used. Confirm all input with OK. Now the main SSD desktop will open.

In case you want to know more about the numeric input please go to chapter 5.5.

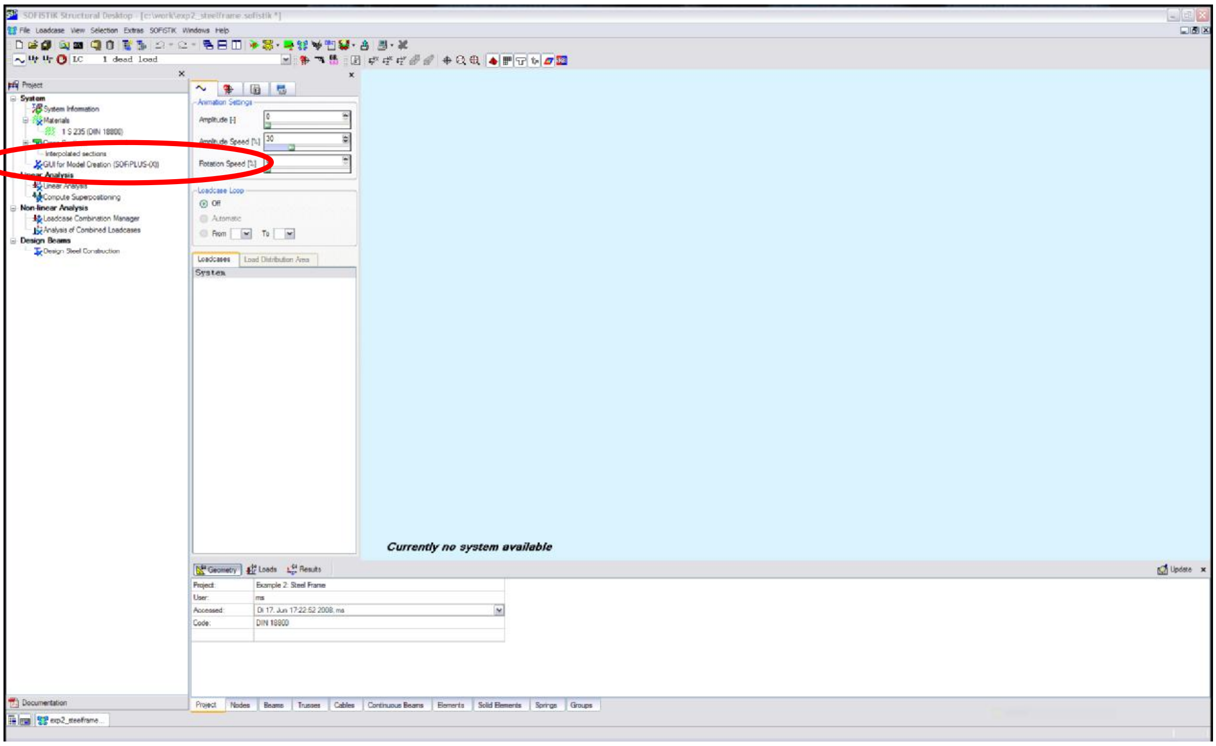


Figure 39: SSD- main desktop

As you see above a new task “GUI for Model Creation (SOFiPLUS(-X))” was added.

First you have to define materials and cross sections. When choosing the German code DIN 18800 a material S 235 is automatically defined. To modify the material just open the task with a double click.

5.3 Step 2: Cross sections

The necessary cross sections are defined by using the task “Cross Sections”. Please start this task with a double click.

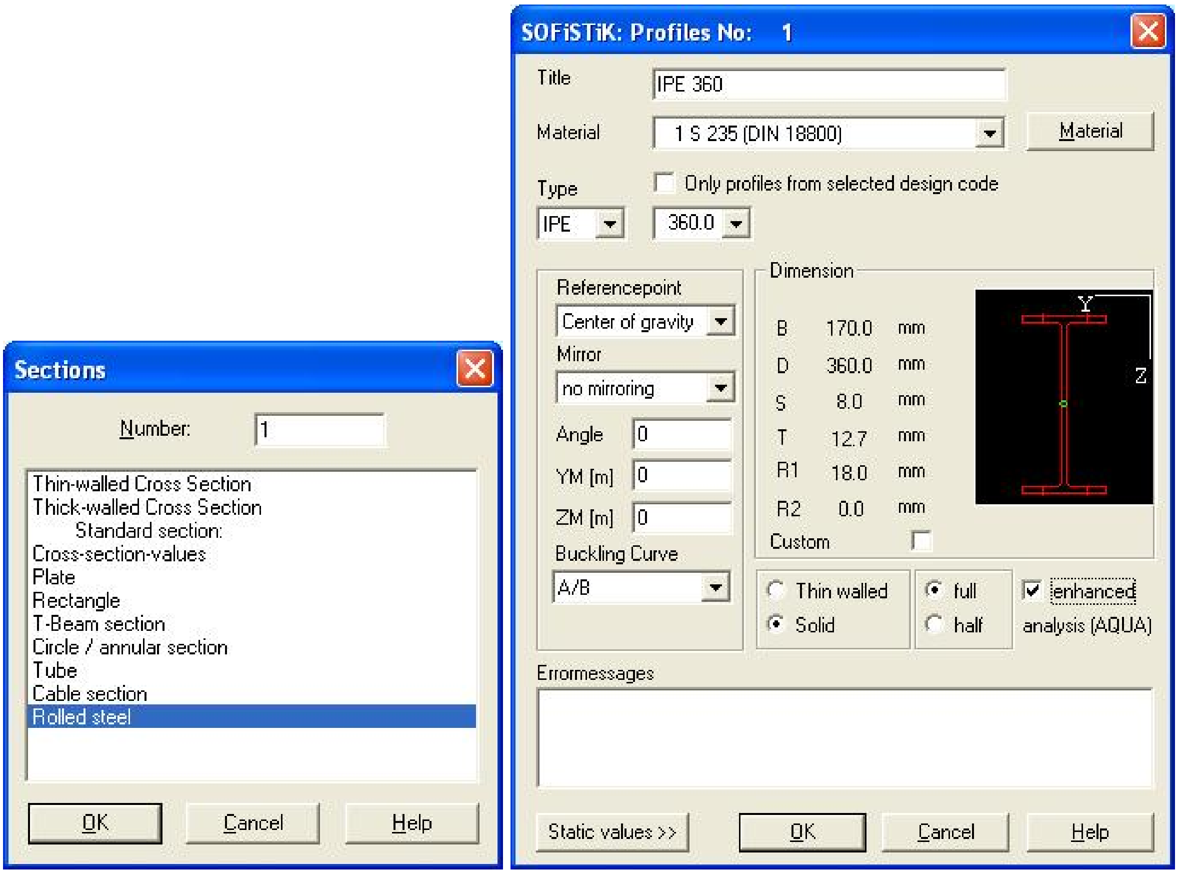


Figure 40: Dialog cross section and rolled steel

Different cross section types are available and are displayed in the dialog. Please select the type “Rolled Steel”, confirm with the OK button and the dialog Profiles will open. Please select a profile and confirm the selection with OK.

- rolled steel profiles are predefined as solid cross sections
- there are full and half profiles possible
- different reference points are possible, for example, to define eccentric beams.
- Rotation of a cross section is possible
- The option “enhanced analysis” is necessary for the calculation of equivalent stresses.
- Using “custom” the profile dimensions can be altered.

5.4 Graphic System - and Load Definition

Double click on the task to open SOFiPLUS(-X) for further input.

5.4.1 Step 3: Graphic Model Creation

For system generation we will use real Finite Elements to define the three beams. Select the

command "create beam element" from the toolbox and the dialog "Beam Element" will be displayed.

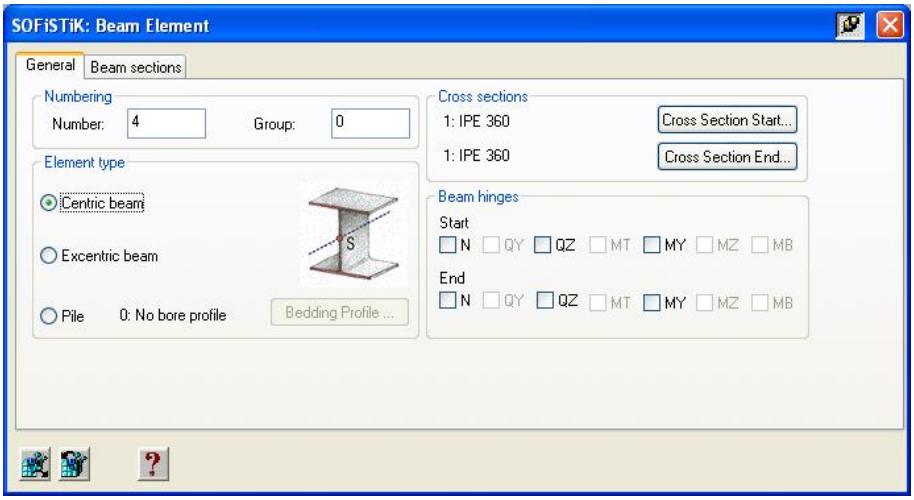
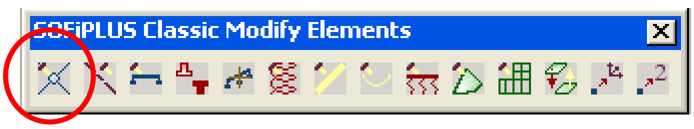


Figure 41: Dialog Beam Element – General Tab

The cross sections defined before are pre-selected for the new beam elements. For plane frame systems only centric beams can be used. To add beam hinges just use the options within this dialog. In this example no hinges are necessary.

After defining the beam attributes please click into the SOFiPLUS drawing window to activate it. After that start defining the first beam with the input of the starting point (0,0), RETURN and the end point (0,-4). The next beam elements are very easy to define. Just define the direction with the mouse and insert the beam length 5 m horizontal and again 4 m vertical downwards. End the input with ESC and this completes the finite element input.



Next step is to define the support conditions. For this select the command "Modify Node"  and select the two bottom nodes. The dialog "Node Element" appears within which the selected nodes should be fixed in x and y directions. Confirm the input with OK.

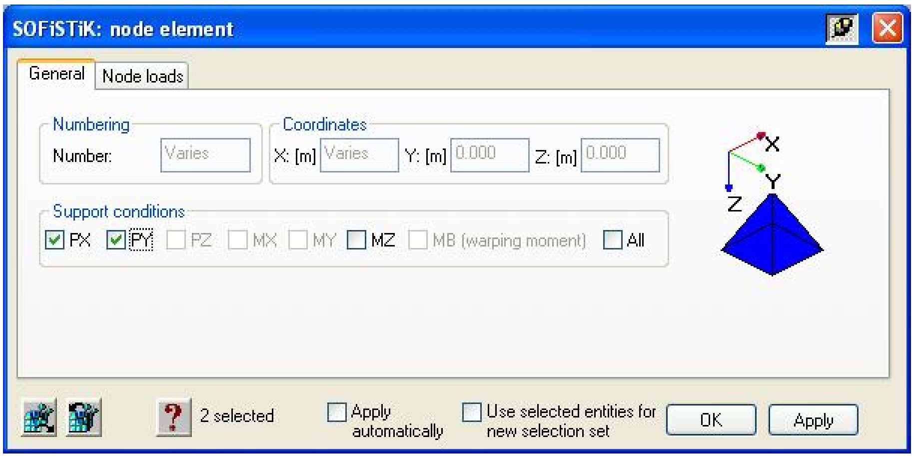


Figure 42: Dialog Node Element – General Tab

The main system generation is finished.

5.4.2 Step 4: Graphic Load Definition

To define the loads start the Loadcase Manger with the command . First it is necessary to define the actions after which you can define the loadcases. In this example, we use the action “total dead load” and the action “wind”. Based on these actions we need three loadcases, LC 1 dead load, LC 2 Wind and LC 3 Imperfection.

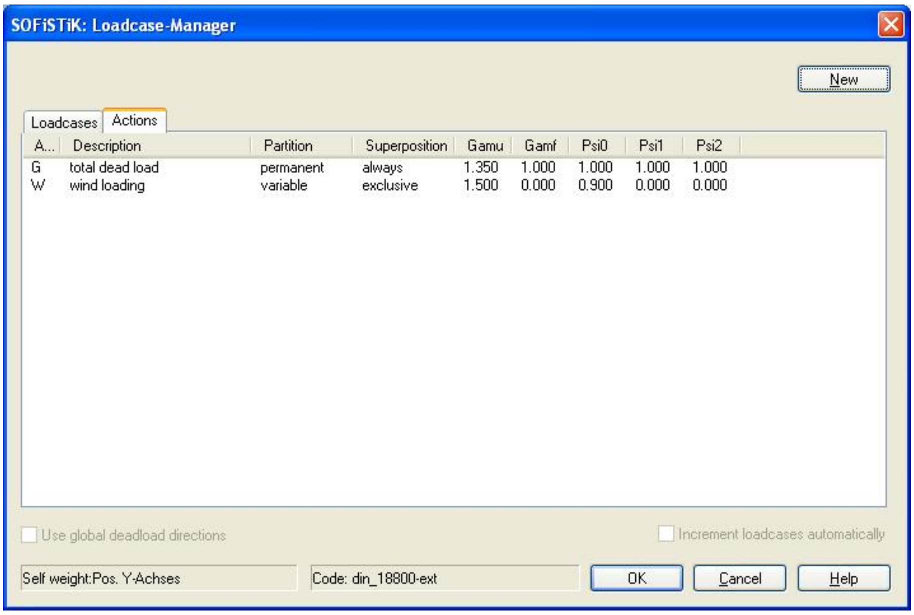
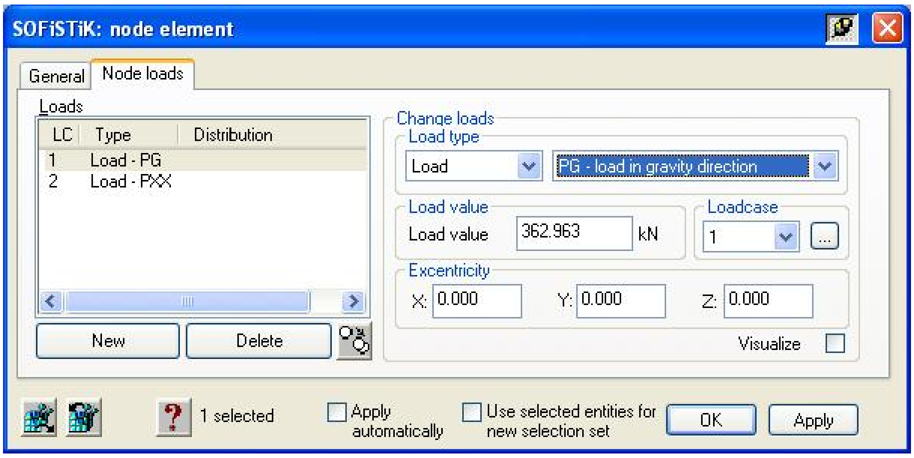


Figure 43: Dialog Loadcase Manager – Actions Tab

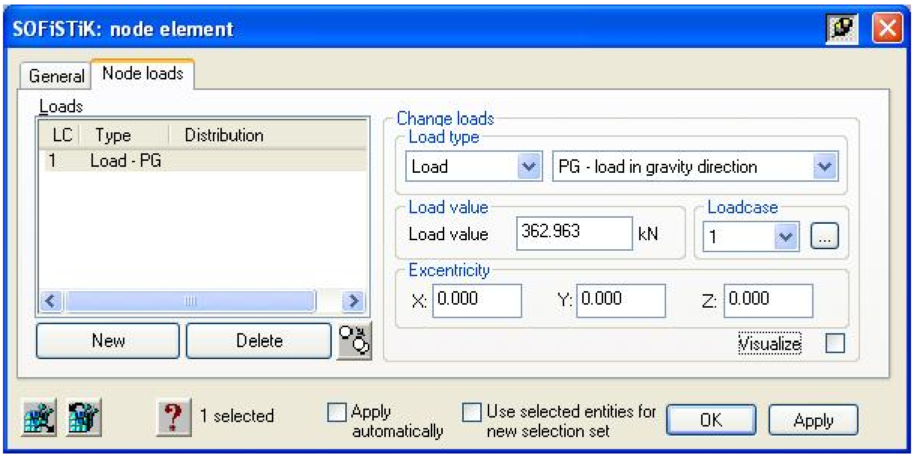
The loads will be defined as nodal loads. For the input choose the command “modify node” and select first the upper left-hand node. In LC 1 create a vertical load 490/1.35 = 362.963 kN and in LC 2 define an horizontal load 30/1.50 = 20 kN.

The loads printied in Figure 37 are design loads. The input must be done with characteristic loads. Therefore the values must be divided by the safety factors of the relevant actions.

Use the Button NEW to creat new loads. The input of every load will be done in the dialog on the left-hand side.



upper left-hand node



upper right-hand node

Figure 44: Dialog Node Element – Input of Node loads

For calculations with 2nd Order Theory it is necessary to create a displacement loadcase. Based on the German design code DIN 18800 an initial sway of φ_0=1/235 but no initial bow is necessary. For the input select the command “modify beam element”  and select first the left-hand frame column. Again use the button NEW and create a new beam load for LC 3. Select the load type imperfection and input the start load value 0.00 mm and the end load value 4000/1.35 = 17.00 mm. De-select the option U.D.L.

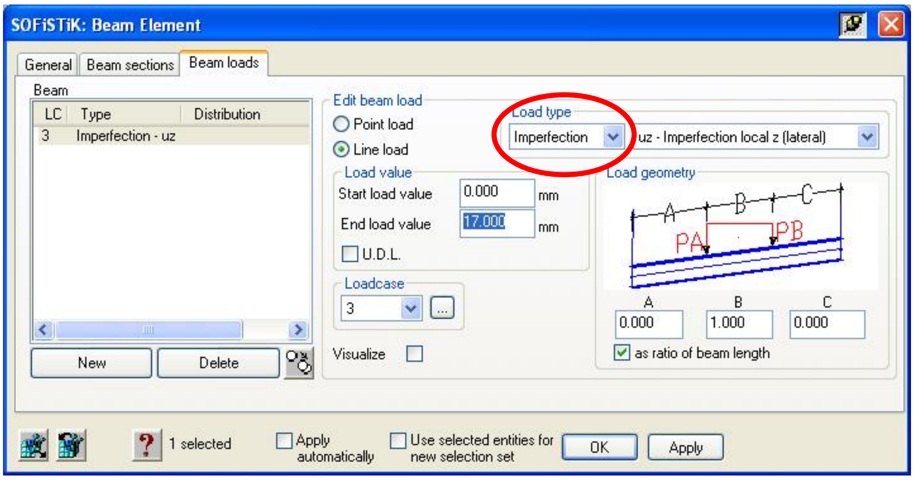


Figure 45: Dialog Beam Element – Beam loads Tab

Go again through the same prodcedure for the right-hand frame column. In this case the start load value is –17.00 mm and the end load value is 0.00 mm. Confirm the settings every time with OK.

The direction of the imperfection displacement depends on the local beam coordinate system. The left-hand frame column was defined upwards, the right-hand frame column was defined downwards. Therefore, the local z-axis points to the left. To make sure both imperfections move to the same side the different input is necessary.

Now the load definition is also finished.

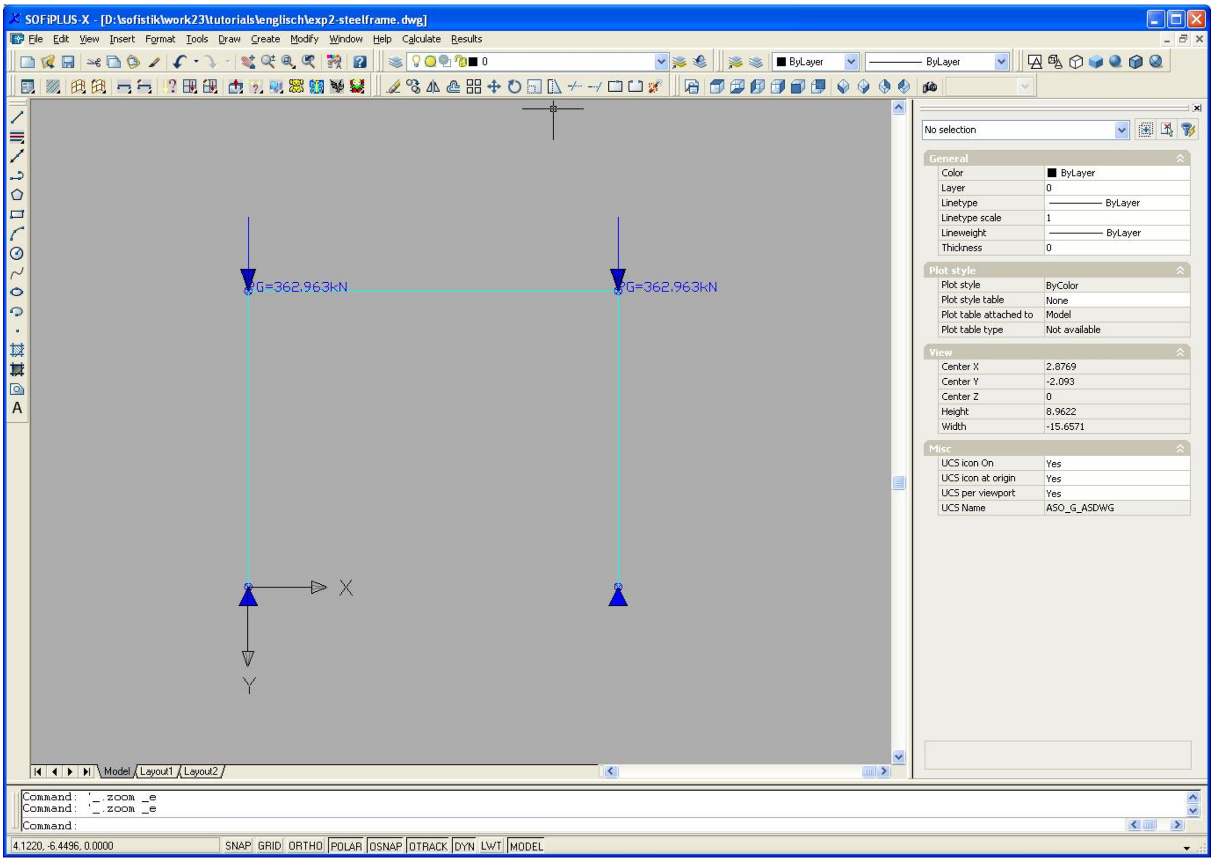


Figure 46: SOFiPLUS - finished System with LC 1

5.4.3 Export to Central Data Base

The graphic definitions are now finished need to be exported into the central data base for further calculation . To export the system, use the command “Export” .

Now save your drawing in SOFiPLUS(-X) and go back to the main SSD window.

5.5 Numeric System- and Load Definition

In this chapter the numeric system and load definition will be presented.

The text input of the system and loads differs from the “Graphic Preprocessing” in that the SSD automatically creates two new tasks “ Text Interface for Model Creation“ and „ Text Interface for Loads“.

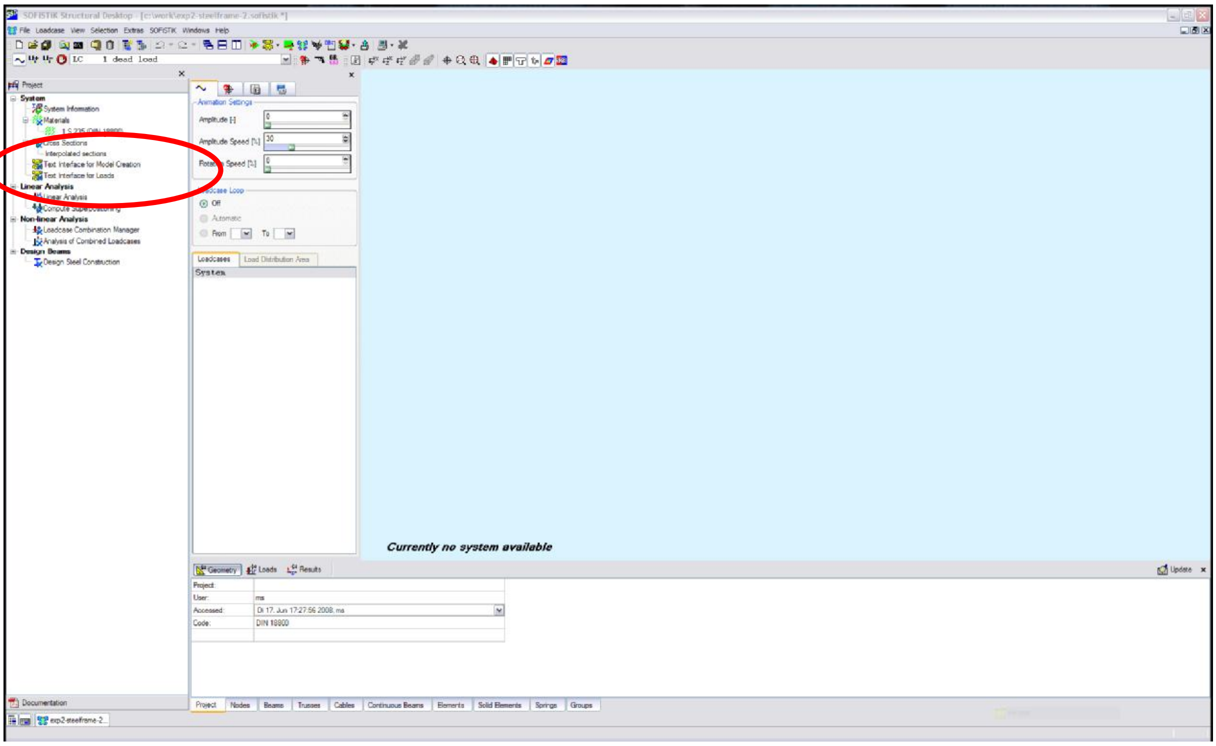


Figure 47: SSD Main view

5.5.1 Step 3: Numeric System Definition

You start the text editor with a double-click on „Text Interface for Model Creation“. The program automatically creates an input block PROG SOFIMSHA for system generation. All necessary inputs will be done within this. SOFiMSHA is also a program for system generation similar to GENF.

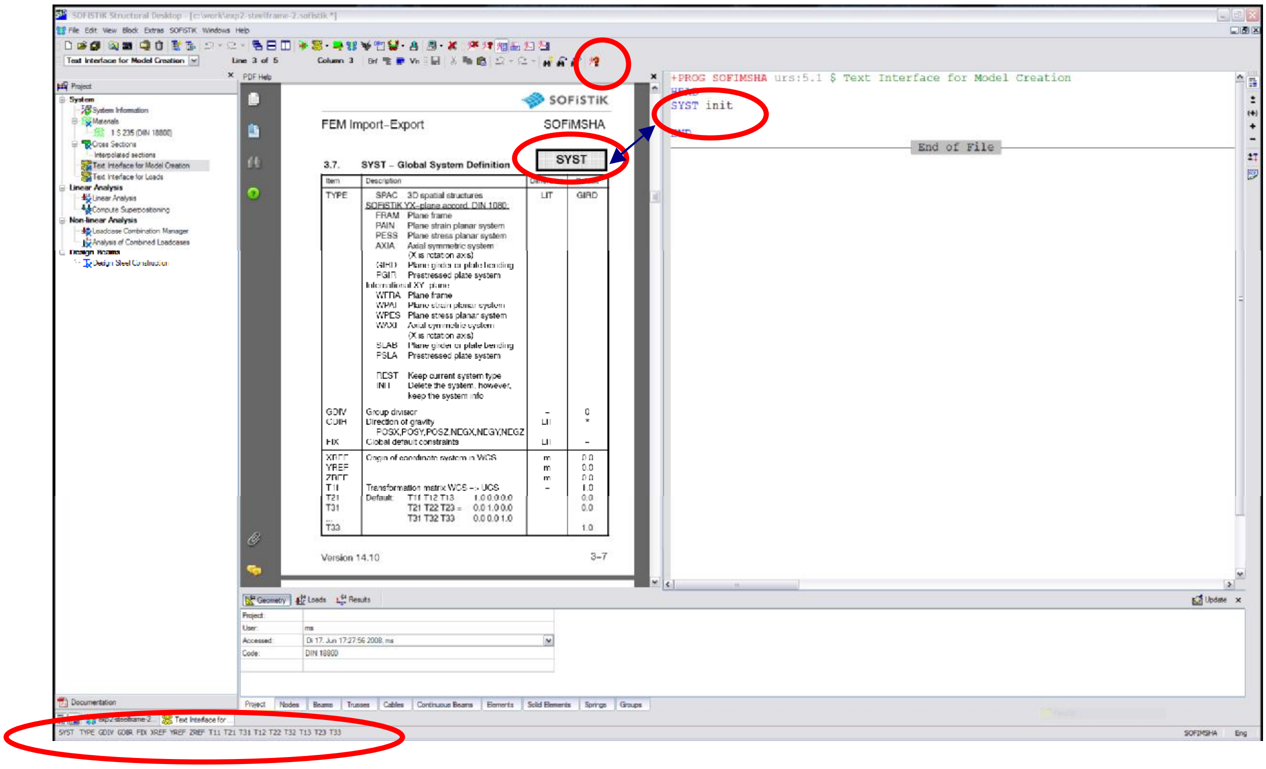


Figure 48: Text editor TEDDY within SSD Desktop

The work is supported by our online help, which can be started by the button . The help function will guide you through every line of input command .

- The complete calculation is done using a main input data file <name>.dat. The text editor TEDDY has to be used for working on this file.
- The input language CADINP is the basis of TEDDY
- The complete input file contains several program blocks. Every block starts with the command PROG NAME and ends with the command END.
- The basic input is a list of commands.
- Every command line contains several Literals. These literals are shown in the SSD status bar.
- Command names and literals are shown in different colours from the normal input values.
- For further explanations please see the SOFiSTiK handbook.

Within the TEDDY you have to generate the static system, by defining nodes, supports and beams. The positive x-direction faces to the right-hand side the positive y-direction faces downwards. The system contains 4 nodes and 3 beam elements.

```
+PROG SOFIMSHA urs:5.1 $ Text Interface for Model Creation
HEAD Steel Frame 2. Order Theory
SYST init
NODE NO 1 X 0 Y  0 FIX PP
NODE NO 2 X 0 Y -4
NODE NO 3 X 5 Y -4
NODE NO 4 X 5 Y  0 FIX PP


BEAM NO 1 NA 1 NE 2 NCS 1 DIV 4
BEAM NO 2 NA 2 NE 3 NCS 1 DIV 5
BEAM NO 3 NA 3 NE 4 NCS 1 DIV 4
END
```

The literal DIV divides the beam element in equal parts, which is important for the analysis results. After creating these commands, start the calculation to generate the system and save it to the central database ,<name>.cdb. To start the calculation use the button . The system will be displayed in the ANIMATOR if the calculation completes without error. Please change the program using the bottom register bsp2-stahlbau.cdb. The system generation is finished.

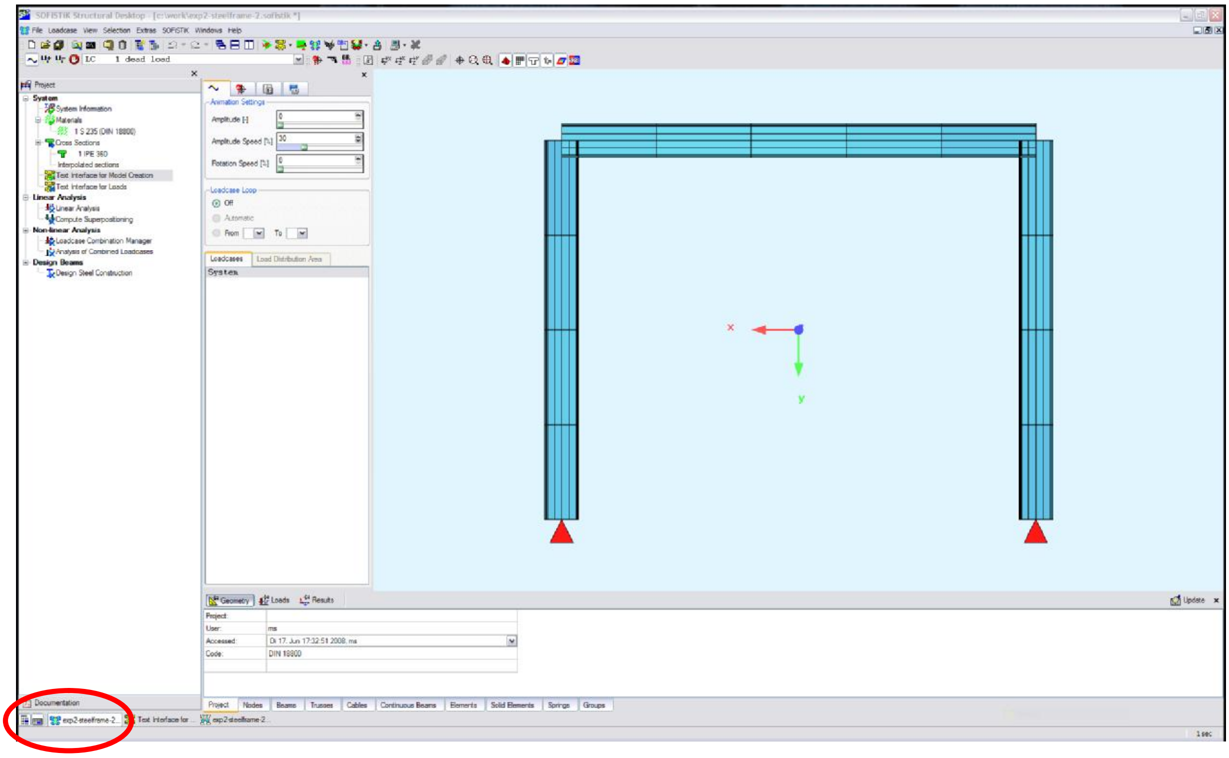


Figure 49: System visualization with ANIMATOR

5.5.2 Step 4: Numeric Load Definition

To define the loads double-click the predefined task „Text Interface for Loads“ and TEDDY will be displayed. We recommend starting again the online-help with .

The load definition, containing actions and loadcases, will be done by the program PROG SOFiLOAD. All input is available for further program blocks.

Some basic work is done automatically by the program. The basic actions dead load, variable load, snow load and wind loading including 4 associated loadcases are predefined. Safety and combinations factors are taken automatically from the pre-defined design code. There is no need for any further input.

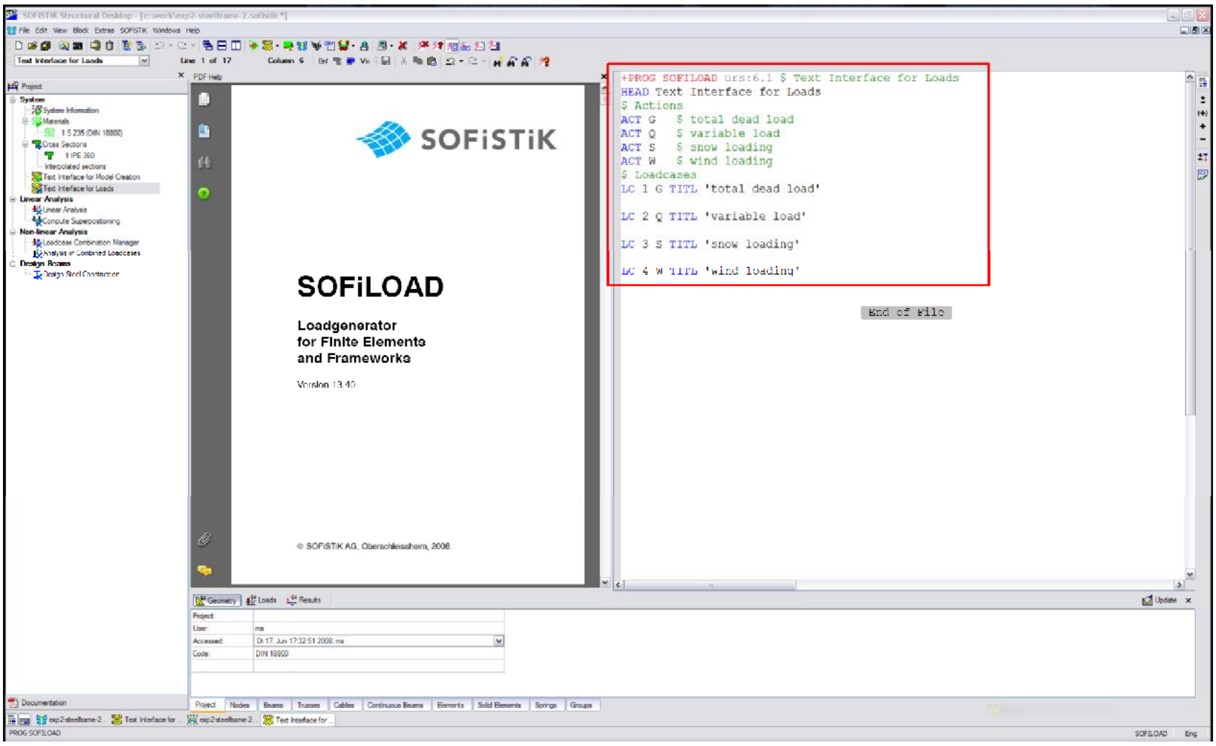


Figure 50: Text editor TEDDY with online help function.

To deal with our problem we only need the actions G…dead load and W…wind load. For calculations with 2nd Order Theory it is necessary to create a displacement loadcase. Therefore we need to define an action I…Imperfection.

Based on the design code DIN 18800 an initial sway of φ_0=1/235 but no initial bow is necessary. The correct input data is shown below.

```
+PROG SOFILOAD urs:6.1 $ Text Interface for Loads
HEAD Text Interface for Loads
$ Actions
ACT G   $ total dead load
ACT W   $ wind loading
$ Loadcases
LC 1 G TITL 'total dead load'
 NODE 2,3 TYPE PY 490/1.35

LC 4 W TITL 'wind loading'
 NODE 2 TYPE PX 30/1.5

LC 5 G TITL 'Imperfection'
 BEAM 1 TYPE UZS PA  0      PE  1/235 REF S
 BEAM 3 TYPE UZS PA -1/235 PE  0      REF S
END
```

Defining the imperfection loadcase LC 5, initial sway, we will use the command BEAM and the literals TYPE UZS and REF S. Please note the CADINP input language is able to use arithmetical expressions.

Every loadcase must be assigned to one specific action. This will be done by the literal TYPE inside the command LF.

The imperfection loadcase is assigned to the action G. Please note that the loadcase factor must set to 1.00 for the combination loadcase.

Since the loads from the problem description include safety factors, the necessary characteristic loads are defined simply by dividing the load value by the relevant safety factor.

- TEDDY is able to use arithmetical expressions.
- The comment character $ turns the rest of the command line into a comment

After finishing the input you can run a linear analysis for the single loadcases. We recommend doing this every time to check the system and loads. Before starting the analysis the loadcases must be saved in the central database by starting the calculation using the button .

Starting the linear analysis is similar to chapter 4.5.

5.6 Step 5: Loadcase Combination Manager

The task “Select Superpositioning” is found inside the group Linear analysis. Because there is no requirement for this task, you can delete it simply by making a right click and Delete.
Important for the 2nd Order Theory is the Loadcase Combination Manager, which you can open with a double-click.
Inside the dialog on the left-hand side all available loadcases are listed. To generate a new combination select the button NEW and a loadcase 1001 will be created. Now select the loadcases on the left-hand side and copy them with the >> button into the new combination. The relevant loadcase factors were copied as well.
If you select the option “Calculate immediately” and confirm the input with OK button, the program starts the calculation and adds a new program block PROG SOFiLOAD into the input file.

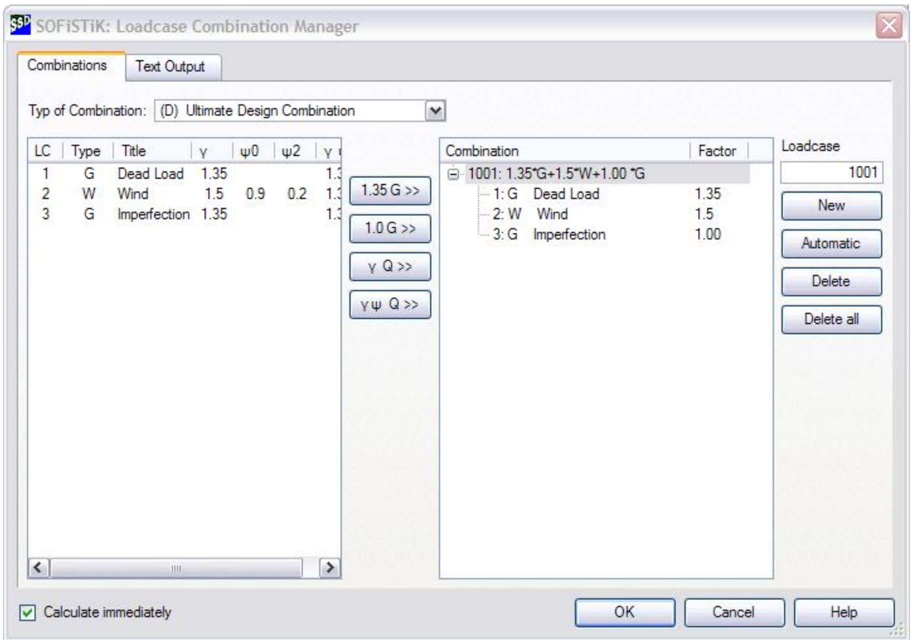


Figure 51: Dialog Loadcase Combination Manager

> In case you use several loadcases of variable actions the total factor of every loadcase is multiplied automatically by gamma*0.9.

5.7 Step 6: Nonlinear Analysis

The nonlinear analysis 2nd Order Theory is now ready to start after all loadcases are defined and saved in the central database. To start the analysis, please open the task “2nd Order Theory”. There is only one single loadcase to select. The non linear analysis should do a maximum of 20 iterations.

The overall stiffness is reduced by 1/1,10 according to DIN 18800.

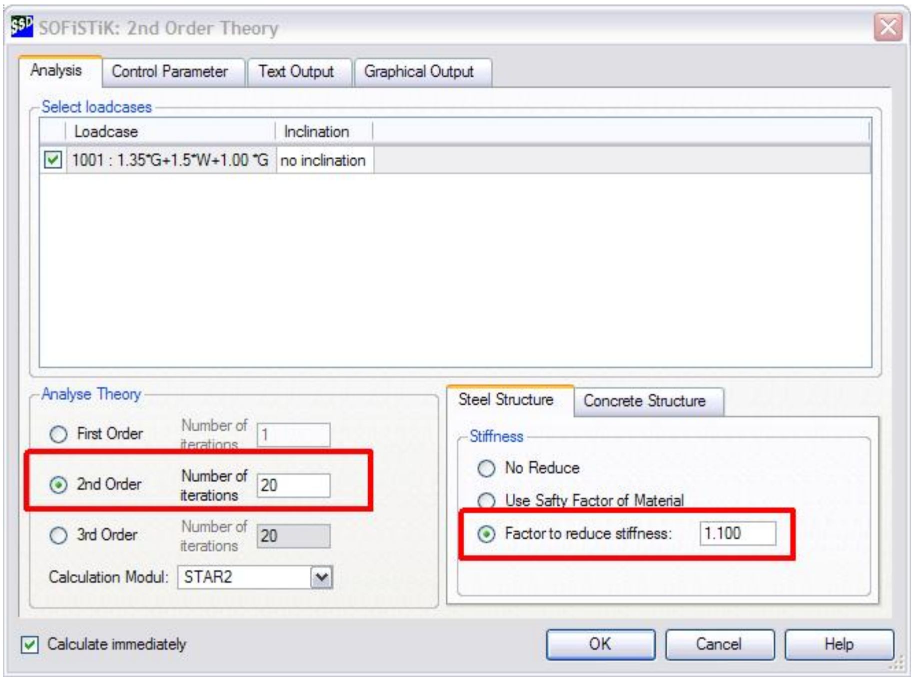


Figure 52: Dialog 2nd Order Theory – Calculation tab

There is no further input necessary in the Control Parameters tab. If the option “Calculate immediately” is selected, the calculations starts immediately after confirming the input with the OK button.

The selection of text and graphical output is identical to all other tasks.

A nonlinear calculation always starts with a linear analysis. Both results will be printed in the output documentation.

5.8 Step 7: Design Steel Construction

Now after the non linear analysis is finished we have to start the steel design. For that please open the task “Design Steel Construction”.

First of all select design loadcases and design elements. By default, all loadcases and all elements will be selected automatically.

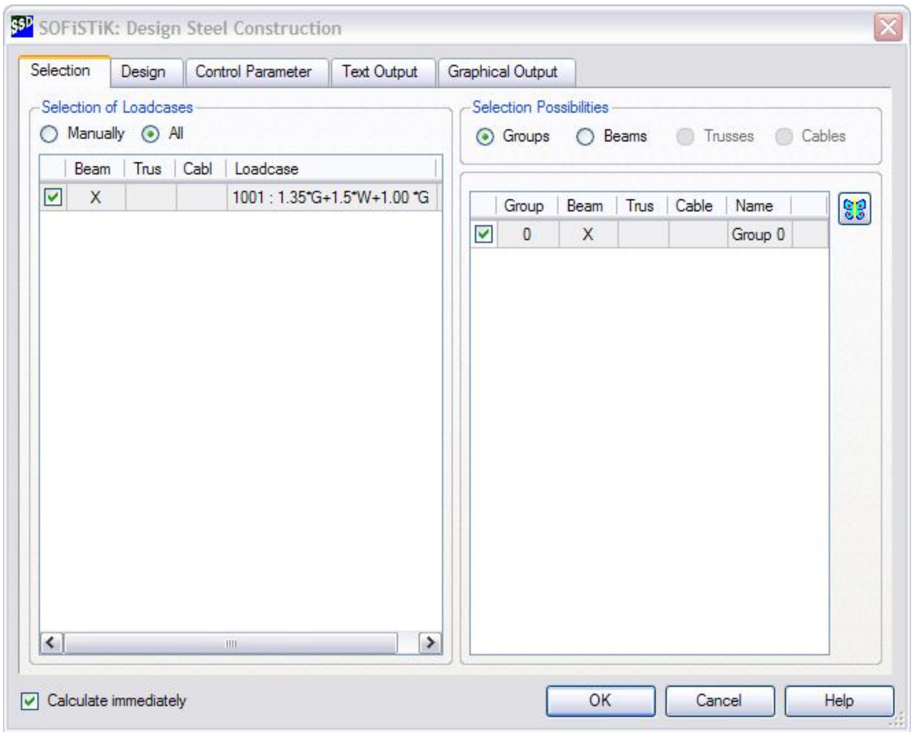


Figure 53: Dialog steel design – Selection tab

You may choose the elements by group or by element number. An interactive graphical selection is also possible which can be started with the button. To select the necessary design calculations go to the Design tab.

There are different design methods available including the limit values b/t.

The design results are saved in special result loadcases.

When using a simple stress analysis, it is also possible to get dimension suggestions automatically by the program.

The selection of text and graphical output is identical to all other tasks.

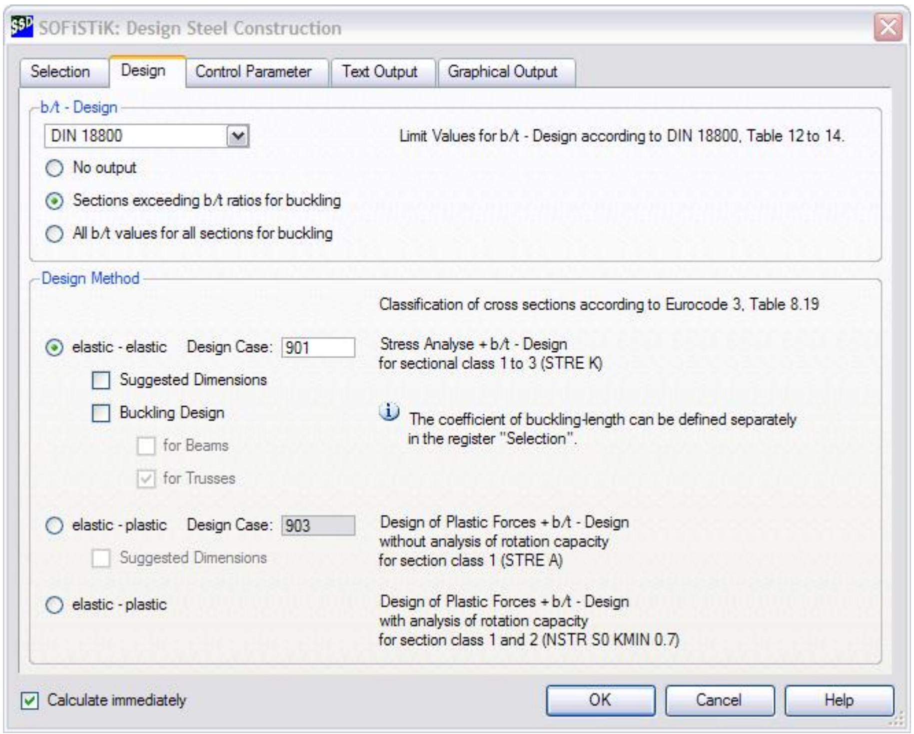


Figure 54:Dialog Design Steel Construction

If several design methods are required simply add a new task “Design Steel Construction”.

5.9 Appraisal of Results

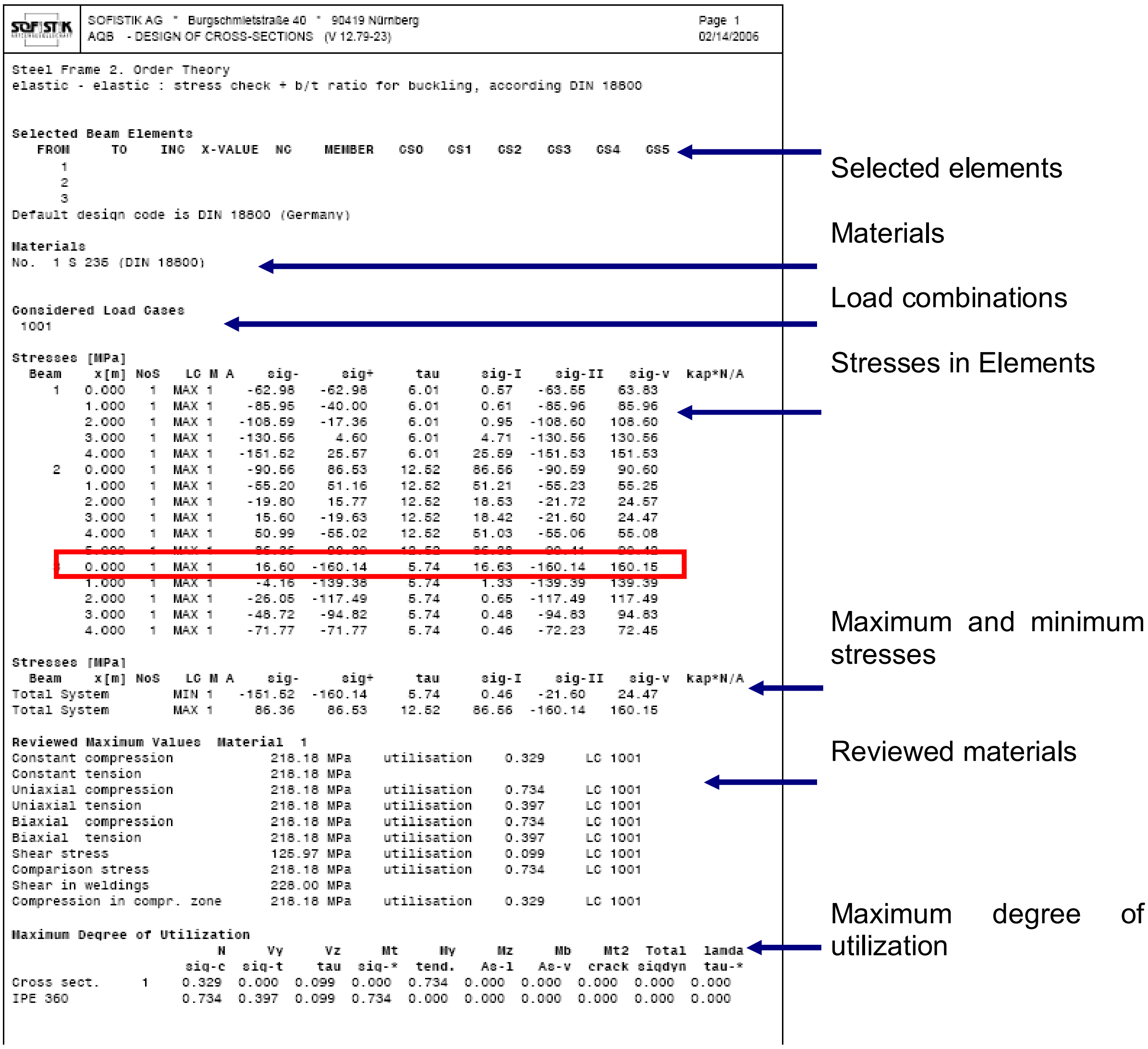
The stress analysis is printed out with the URSULA result browser. (see Figure 55)

The results from the SOFiSTiK calculation and from the literature are listed in Table 3.

	Literature	SOFiSTiK
Normal force N beam 3 [kN]	-521,67	-522,0
Bending moment My beam 3. node 3 [kNm]	-78,93	-79,86
Maximum stress el-el $\|\sigma\|$ [kN/cm²]	16,6	16,02

Table 3: Appraisal results Literature and SOFiSTiK calculation

The results are very similar.


```
SOFISTIK AG * Burgschmietstraße 40 * 90419 Nürnberg                         Page 1
AQB - DESIGN OF CROSS-SECTIONS (V 12.79-23)                                 02/14/2006

Steel Frame 2. Order Theory
elastic - elastic : stress check + b/t ratio for buckling, according DIN 18800

Selected Beam Elements
   FROM      TO     INC  X-VALUE  NC    MEMBER   CS0   CS1   CS2   CS3   CS4   CS5
      1
      2
      3
Default design code is DIN 18800 (Germany)

Materials
No.  1 S 235 (DIN 18800)

Considered Load Cases
 1001

Stresses [MPa]
  Beam     x[m] NoS   LC M A    sig-      sig+      tau      sig-I    sig-II   sig-v  kap*N/A
     1    0.000   1  MAX 1    -62.98    -62.98     6.01     0.57    -63.55    63.83
          1.000   1  MAX 1    -85.95    -40.00     6.01     0.61    -85.96    85.96
          2.000   1  MAX 1   -108.59    -17.36     6.01     0.95   -108.60   108.60
          3.000   1  MAX 1   -130.56      4.60     6.01     4.71   -130.56   130.56
          4.000   1  MAX 1   -151.52     25.57     6.01    25.59   -151.53   151.53
     2    0.000   1  MAX 1    -90.56     86.53    12.52    86.56    -90.59    90.60
          1.000   1  MAX 1    -55.20     51.16    12.52    51.21    -55.23    55.25
          2.000   1  MAX 1    -19.80     15.77    12.52    18.53    -21.72    24.57
          3.000   1  MAX 1     15.60    -19.63    12.52    18.42    -21.60    24.47
          4.000   1  MAX 1     50.99    -55.02    12.52    51.03    -55.06    55.08
          5.000   1  MAX 1     86.36  [illegible] 12.52 [illegible] [illegible] [illegible]
     3    0.000   1  MAX 1     16.60   -160.14     5.74    16.63   -160.14   160.15
          1.000   1  MAX 1     -4.16   -139.38     5.74     1.33   -139.39   139.39
          2.000   1  MAX 1    -26.05   -117.49     5.74     0.65   -117.49   117.49
          3.000   1  MAX 1    -48.72    -94.82     5.74     0.48    -94.83    94.83
          4.000   1  MAX 1    -71.77    -71.77     5.74     0.46    -72.23    72.45

Stresses [MPa]
  Beam     x[m] NoS   LC M A    sig-      sig+      tau      sig-I    sig-II   sig-v  kap*N/A
Total System         MIN 1   -151.52   -160.14     5.74     0.46    -21.60    24.47
Total System         MAX 1     86.36     86.53    12.52    86.56   -160.14   160.15

Reviewed Maximum Values  Material  1
Constant compression           218.18 MPa   utilisation   0.329    LC 1001
Constant tension               218.18 MPa
Uniaxial compression           218.18 MPa   utilisation   0.734    LC 1001
Uniaxial tension               218.18 MPa   utilisation   0.397    LC 1001
Biaxial  compression           218.18 MPa   utilisation   0.734    LC 1001
Biaxial  tension               218.18 MPa   utilisation   0.397    LC 1001
Shear stress                   125.97 MPa   utilisation   0.099    LC 1001
Comparison stress              218.18 MPa   utilisation   0.734    LC 1001
Shear in weldings              228.00 MPa
Compression in compr. zone     218.18 MPa   utilisation   0.329    LC 1001

Maximum Degree of Utilization
                          N      Vy      Vz      Mt      My      Mz      Mb     Mt2   Total   lamda
                      sig-c   sig-t     tau   sig-*   tend.    As-l    As-v   crack  sigdyn   tau-*
Cross sect.    1      0.329   0.000   0.099   0.000   0.734   0.000   0.000   0.000   0.000   0.000
IPE 360               0.734   0.397   0.099   0.734   0.000   0.000   0.000   0.000   0.000   0.000
```


Figure 55: Output URSULA result browser

6 SSD Functionality

6.1 Description TASKS

All the following tasks are provided by the SSD (see chapter Figure 3). The basic contents and functionality are described in the following chapters.

System		
	Bore profile	(see chapter 6.2)
	Work Laws for Springs	(see chapter 6.3)
	Prestressing	(see chapter 6.4)
	Text Interface for Model Creation	(see chapter 5.4)
	Text Interface for Loads	(see chapter 5.5.2)
Linear Analysis		
	Linear Analysis	(see chapter 4.5)
	Define Superpositioning	(see Manual MAXIMA Chapter 4)
	Select Superpositioning	(see chapter 4.5)
Design Area Elements		
	Design in ULS	(see chapter 4.6.2)
	Design in SLS	(see chapter 4.6.3)
Design Beam Elements		
	Design Steel Construction	(see chapter 5.8)
	Lateral Torsional Buckling	
	Design in ULS -Beams	(see chapter 6.5
	Design in ACCI -Beams	(see chapter 6.6
	Design in SLS -Beams	(see chapter 6.7
Non-linear Analysis		
	Non-linear Analysis in ULS	(see chapter 6.8)
	Non-linear Analysis in ULS	(see chapter 6.9)
	Loadcase Combination Manager	(see chapter 5.6)
	2nd Order Theory	(see chapter 5.7)

Prestressed Structure

Beam and Slab Bridge	(see separate Tutorial Bridge Design)
Arch Bridge	
Pre Tee	
Analysis of Slab Prestress	

Additional Modules

Interactive Graphic	(see Manual WINGRAF)
Earthquake	(see separate Tutorial Earthquake)
CSM Construktion Stage Manager	(see separate Tutorial Bridge design)
Eigenvalues	(see chapter 6.11)
Buckling Eigenvalues	(see chapter)
Half space Calculation	(see separate Tutorial Half Space)
Design RC section	(see chapter)
Summary of Masses	(see chapter)
Text Editor (TEDDY)	(see chapter 5.5.2)
WALLS-X	(see Manual WALLS)

6.2 Bore Profile

Using the task Bore Profile there are two basic possibilities. To define a pile bedding constant or use a constrained soil modulus. For each pile, a geometric start coordinate and direction is required.

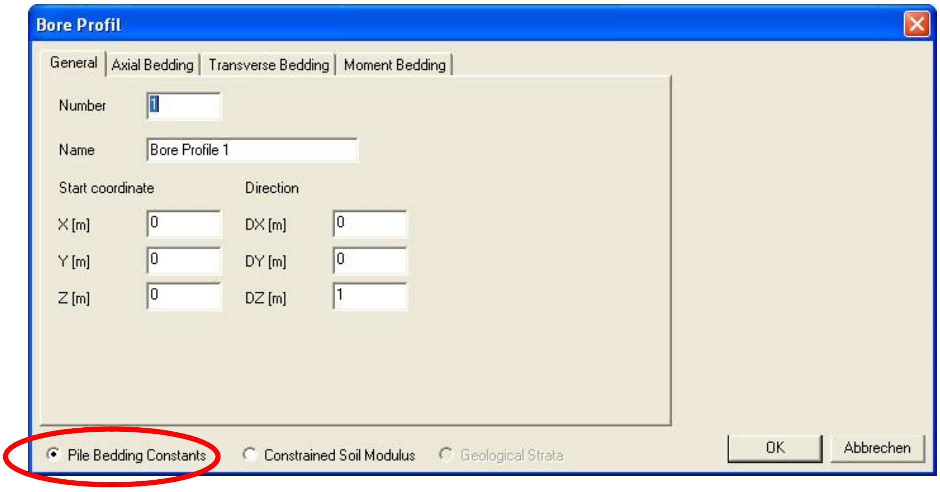


Figure 56: Dialog Bore Profile – General tab

Soil bedding modulus for the pile elements.
These values are derived from the soil modulus above by a multiplication with a form factor with typical values between 0.5 and 2.0. More precisely the soil modulus is transferred to a Winkler bedding constant [kN/m3] by a division with some structural dimension and is then integrated by a multiplication with the width of the pile section.

Properties of the constrained soil modulus for the analysis of settlements or a half space modelling with HASE.

In Figure 56 the basic input for a bore profile is shown. In addition to that, the axial bedding, the transverse bedding and the moment bedding have to be defined. The input follows the same principle in all three cases. The input may contain several soil layers, which are defined by the input of depths from the top of the pile. To create a new depth, select the button NEW. Default distance is 1 m. The different modulus values may be defined for each depth separately. On the right-hand side a graphic displays the actual input. For further information please look at the handbook AQUA.

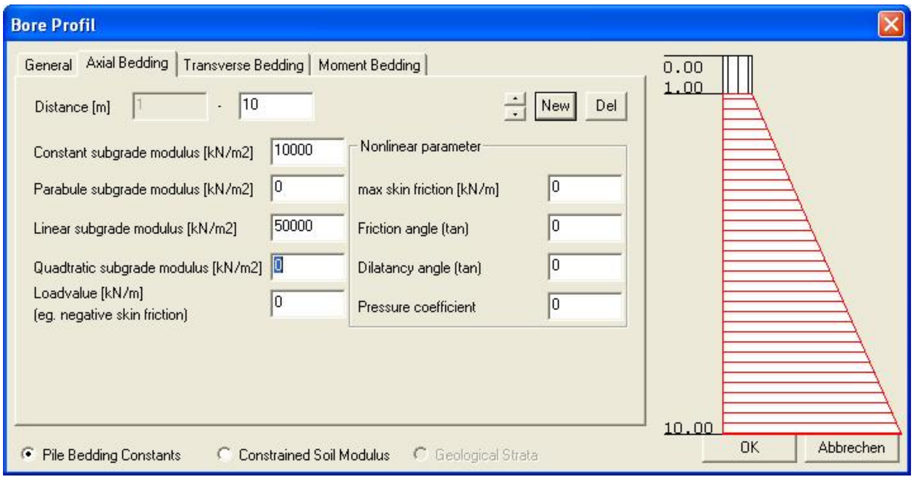


Figure 57: Dialog Bore Profile – Axial Bedding tab

6.3 Work Laws for Springs an Implicit Beam Hinges

Using this task, you may define special work laws for springs and for implicit beam hinges. A worklaw can be defined for every single degree of freedom. Please use the predefined worklaws out of the column TYPE.

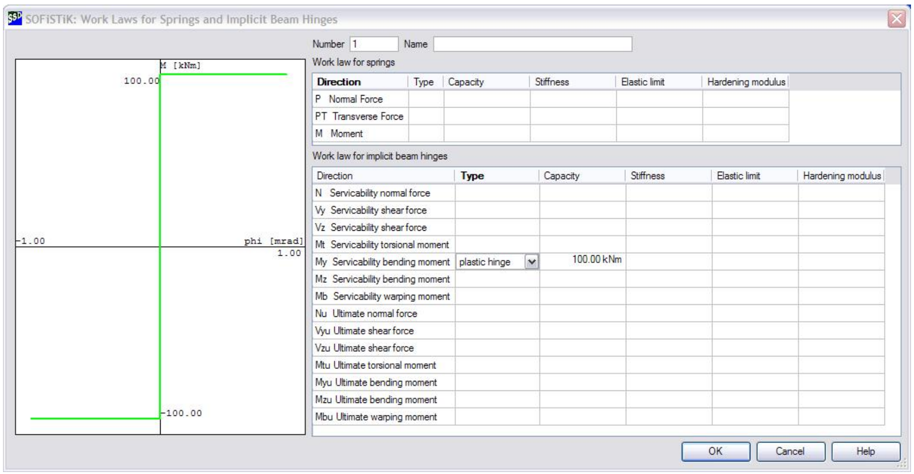


Figure 58: Dialog Stress Strain Curves for Springs

6.4 Prestressing Systems

In particualr for posttensioned slabs this task makes it very easy to define the prestressing systems.

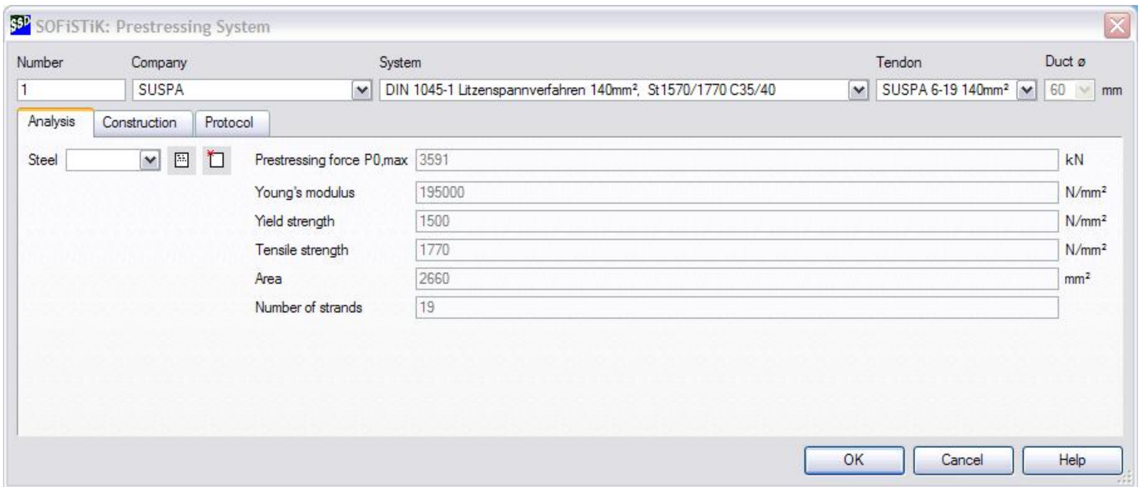


Figure 59: Task Prestressing Systems

6.5 Design ULS –Beams

Use this task to design beams to ultimate limit state.

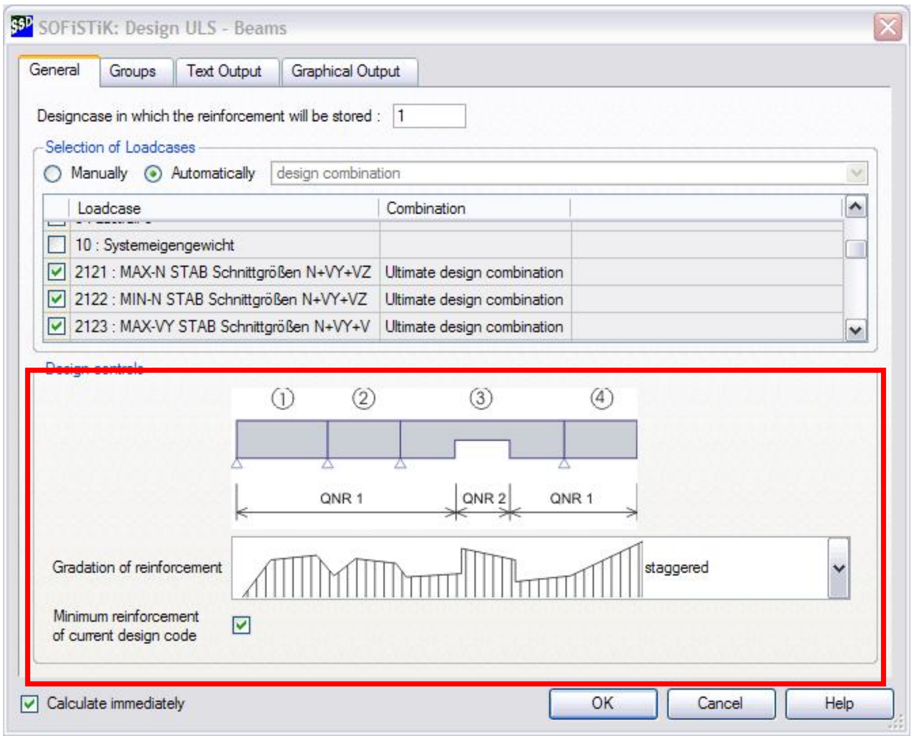


Figure 60: Dialog Design ULS – Beam

As in all design tasks, first select the design loadcase. The selection can be either manual or automatic. The next step is to select the reinforcement distribution. There are variable distributions available. For further information see the AQB handbook, command REIN.

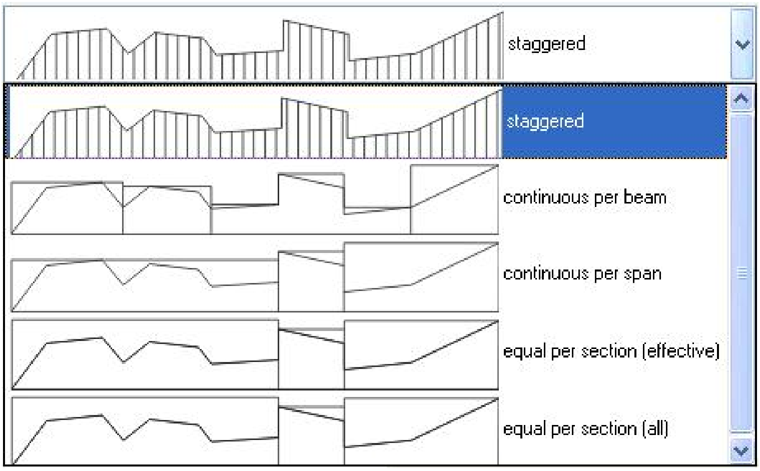


Figure 61: Distribution of reinforcement

6.6 Design ACCI –Beams

For accidental design, start a new task. The complete input is similar to the design in ULS.

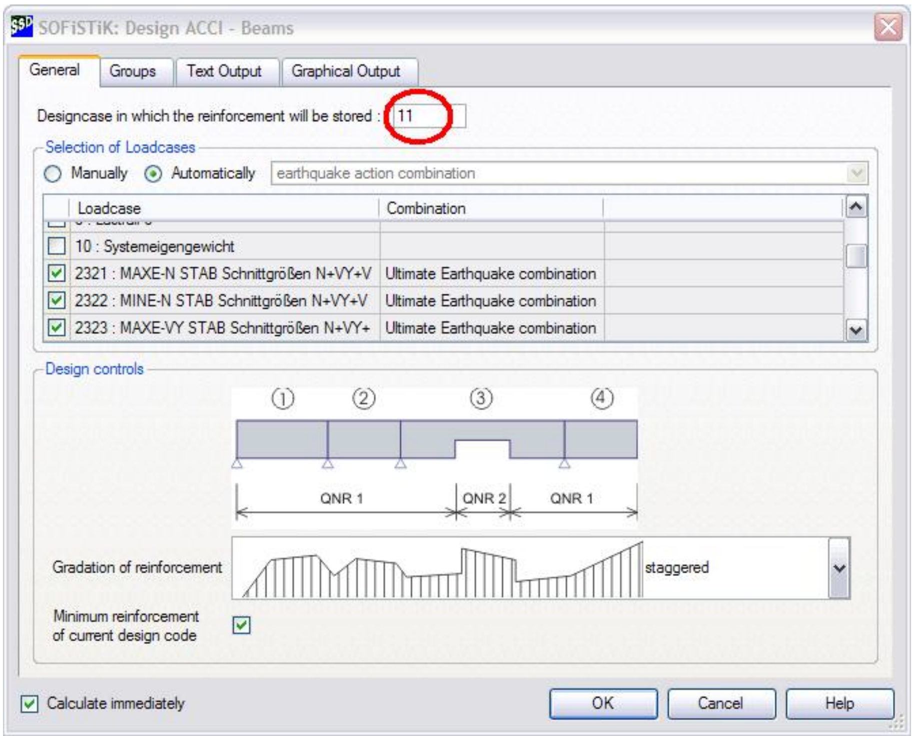


Figure 62: Dialog Design ACCI –Beams

The reinforcement distribution from the accidental design will be saved in a separate loadcase number 11. The reinforcement results will not overwrite the reinforcement from the ULS or SLS design.

6.7 Design SLS –Beams

Again the task Design SLS is similar to the other design tasks. Important is the design control where the crack width is defined. The crack width can be defined manually instead of the default settings which will use the values according to the selected design code. .

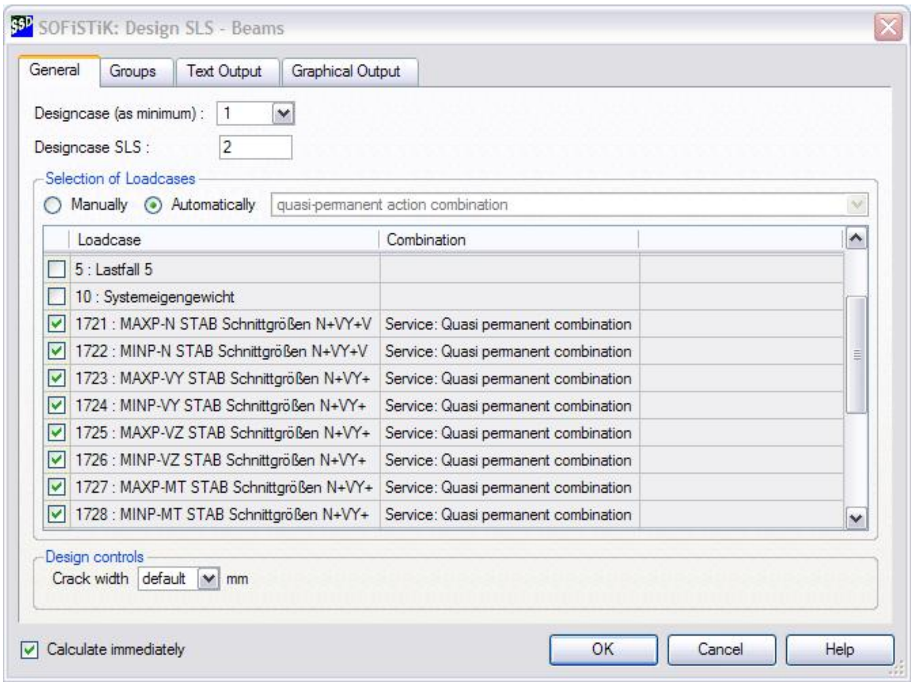


Figure 63: Dialog Design SLS – Beams

6.8 Non-linear Analysis in ULS

Using this task you start a nonlinear calculation with cracked concrete.

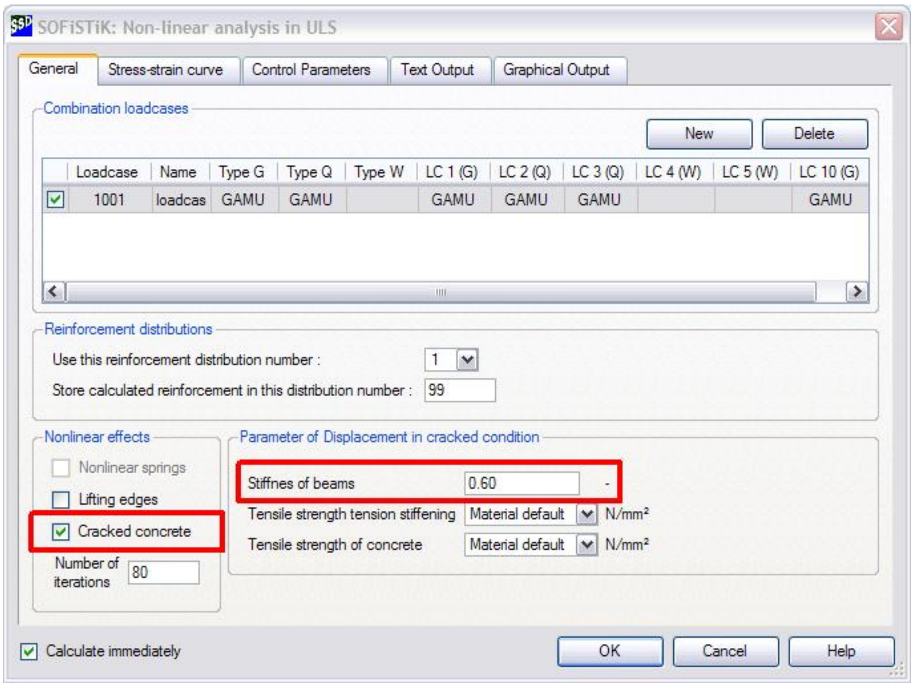


Figure 64: Dialog Non-linear analysis in ULS – General tab

This calculation is useful to check the loading capacity of members where insufficient reinforcement is provided. For this calculation, a work law will be used. The different pre-defined work laws can be selected in the register “Stress-strain-curves”. The ultimate limit state with safety factors is pre-selected.

6.9 Non-linear Analysis in SLS

This task may be used for the calculation of long term deflections in concrete structures. The nonlinear material will be used and therefore you will get very accurate results. The influence of creep and shrinkage will also be taken into account.

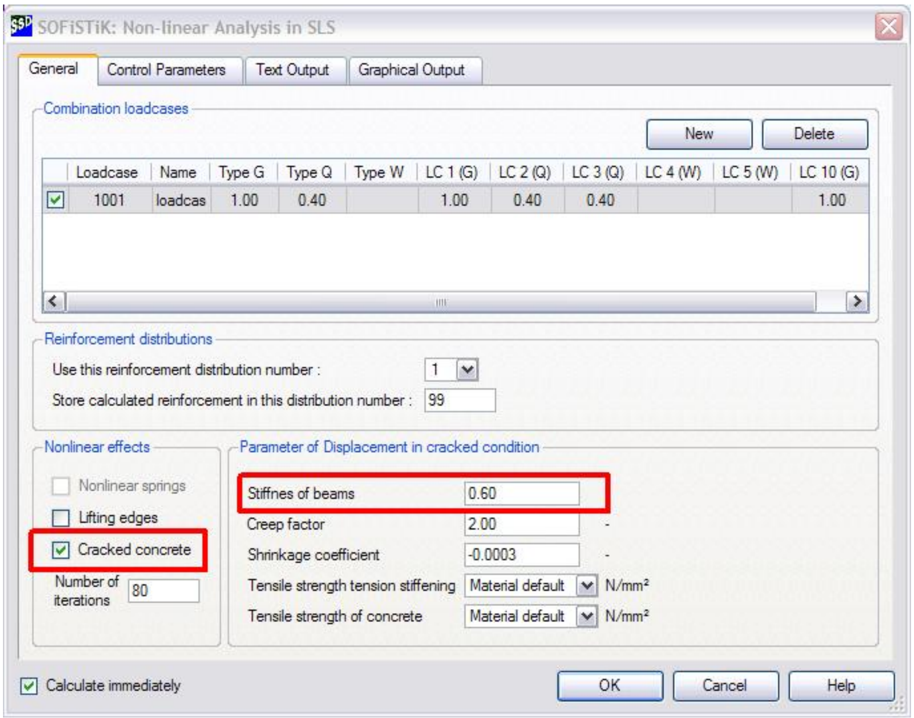


Figure 65: Dialog Non-linear Analysis in SLS – General tab

6.10 Construction Stage Manager

With this task it is very easy to define a construction schedule for your system. Simply create the construction stages, the groups per construction stage and the load cases.

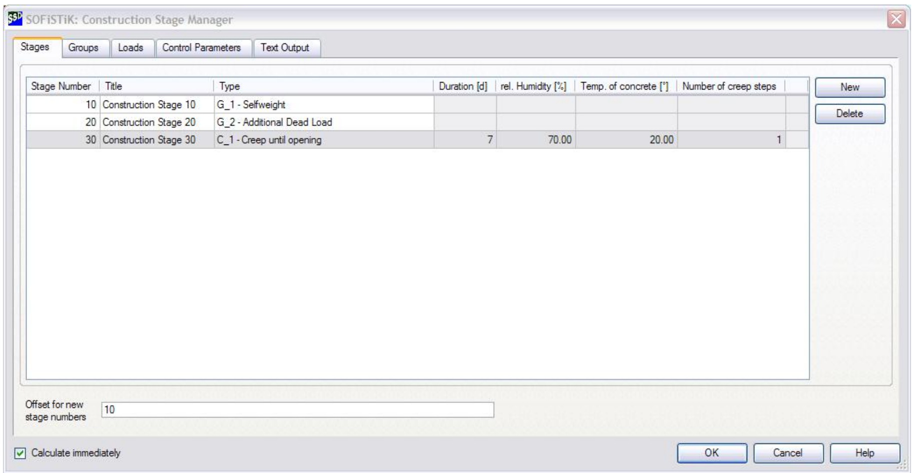


Figure 66: Construction Stage Manager

6.11 Eigenvalues

Eigenvalues are simply calculated by the named task. Just add the number of sought Eigenvalues and the first storage loadcase number and start the calculation. The results will be saved in storage loadcases starting with the first number from the input dialog.

The storage loadcase number has to be different from any other loadcase number. In situation where a loadcase 2001 already exists, the new storage loadcase 2001 from the Eigenvalue calculation will overwrite the other results.

It is only possible to convert one loadcase into additional masses.

Additional masses from only one loadcase is possible.

Using a primary loadcase, only the stresses will be used for the Eigenvalue calculation.

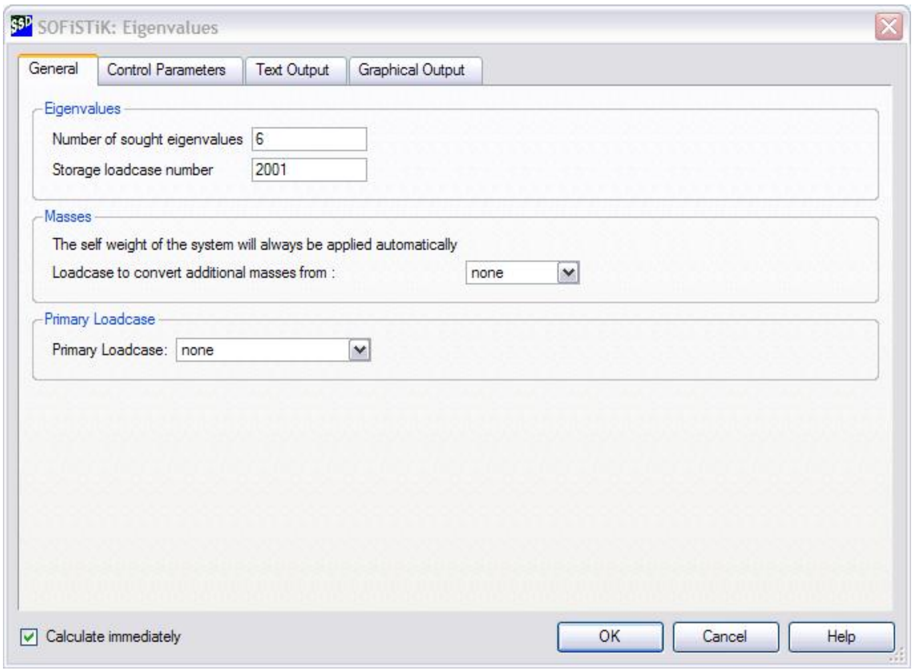


Figure 67: Dialog Eigenvalues

6.12 Buckling Eigenvalues

Use this task for stability checks. Just add the number of sought Eigenvalues and the first storage loadcase number and start the calculation. The results will be saved in storage loadcases starting with the first number from the input dialog.

The storage loadcase number has to be different from any other loadcase number. In situation where a loadcase 3001 already exists, the new storage loadcase 3001 from the Eigenvalue calculation will overwrite the other results.

It is only possible to convert one loadcase into additional masses.

Additional masses from only one loadcase is possible.

Using a primary loadcase, only the stresses will be used for the Eigenvalue calculation.

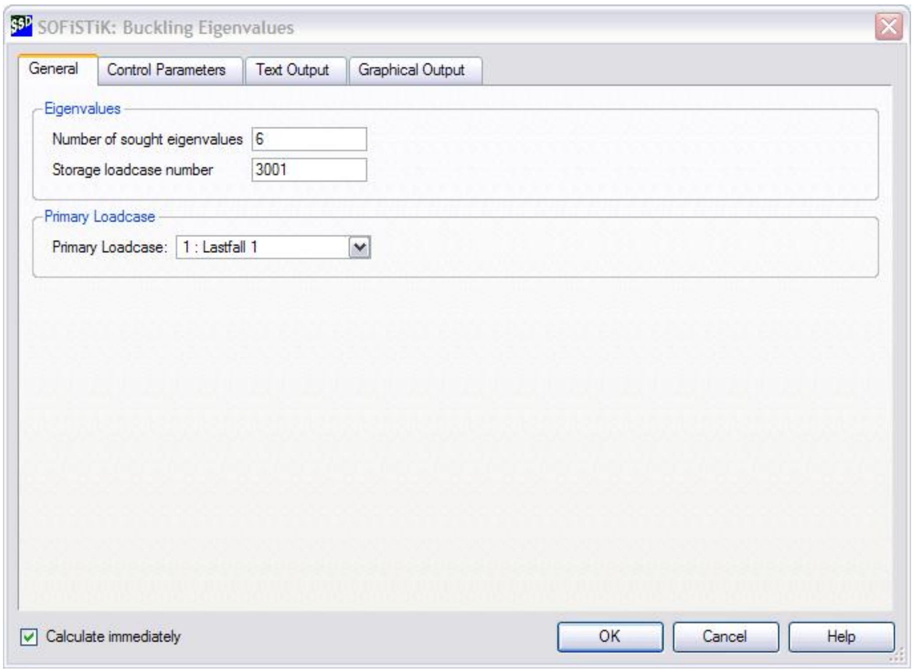


Figure 68: Task „Buckling Eigenvalues“

6.13 Design RC Section

This task may be used for simple design checks, without analysis the whole model. Just input the design forces and start the calculation.

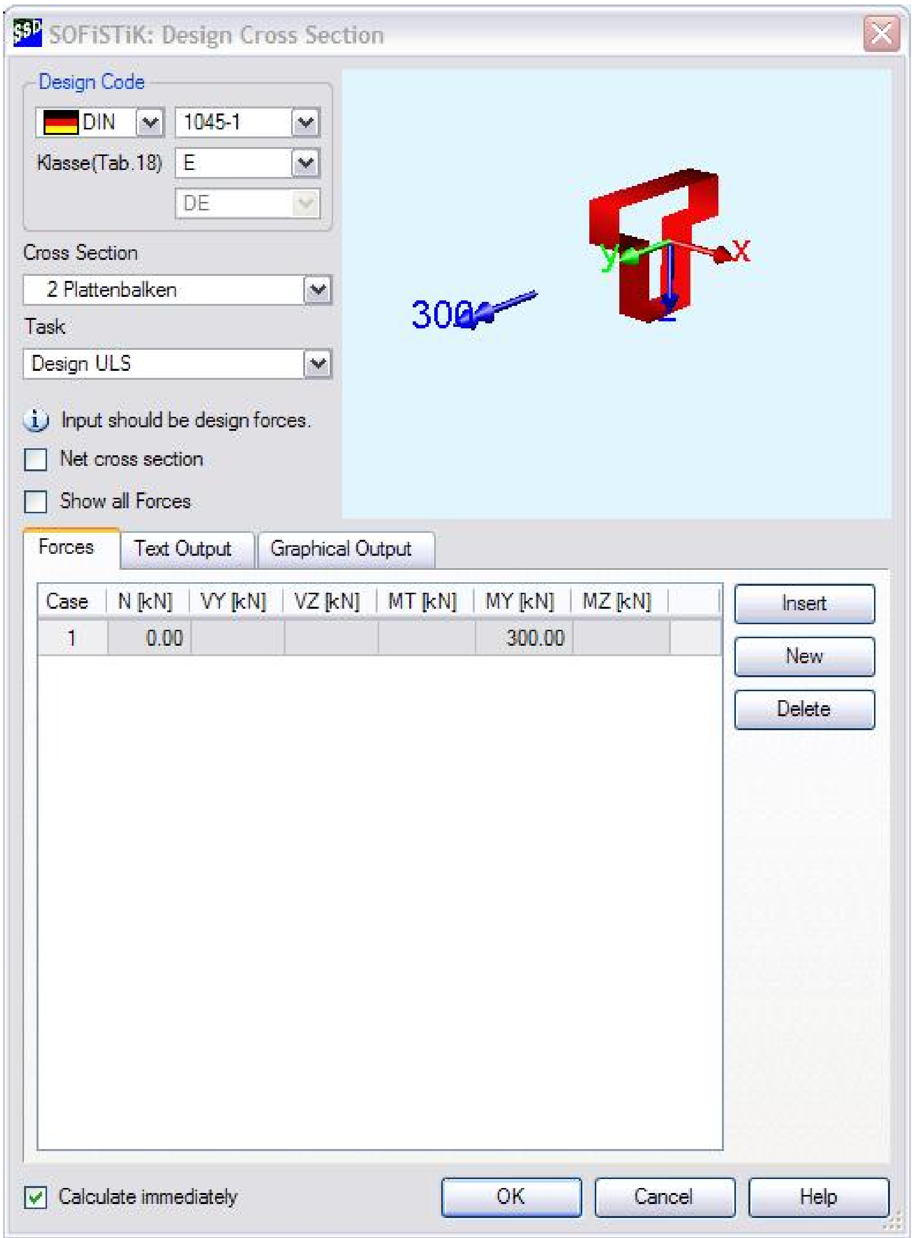


Figure 69: Task Design RC Section

6.14 Summary of Masses

A very useful task is the “Summary of Masses”, which gives you a quick overview of all system relevant masses.

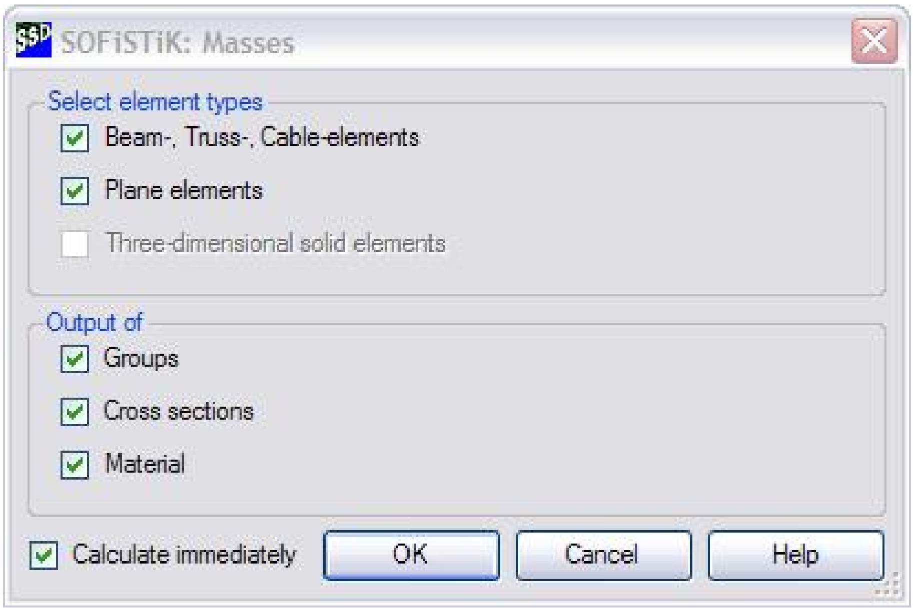


Figure 70: Task

All Masses are related only to the structural system.

The mass of reinforcement is calculated without lap length and additional structural reinforcement

7 Support

In case you have further question please contact our Hotline Services via Email support@sofistik.de or via phone under +49-(0)700-76347845 (12.4 ct/Minute). We are on your service daily from 9-12 am and 2-5 pm on Fridays 2-4 pm.
For every Support request we need to know your Customer number and all used program versions. In most cases it is much faster to send us an Email, containing all relvant data including a small example, which will show your problem.

Inside the SSD menu Help you will find a command "SOFiSTiK Hotline…" which will guide you through 4 dialog boxes to create a new support request with all necessary data. You may send this request directly out of SSD oder copy and paste text and attachment to your Email browser and send it to support@sofiistik.de .
The 4 Steps to create an support Request are shown below.

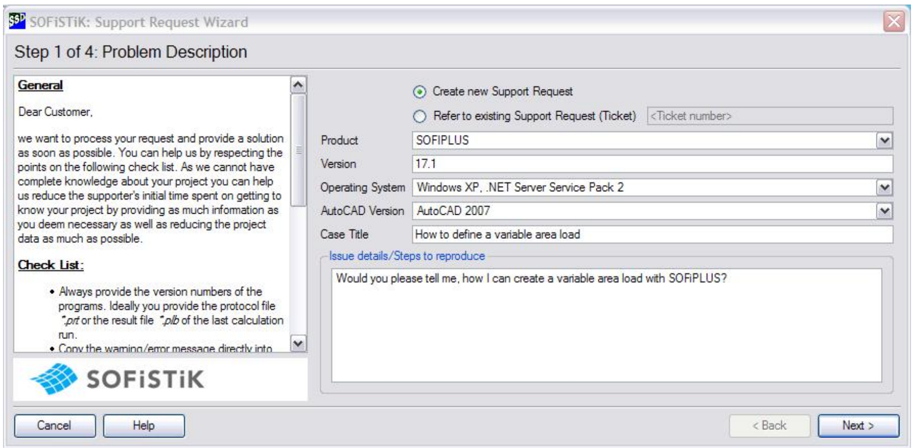


Figure 71: Step 1 of 4 Problem Descritption

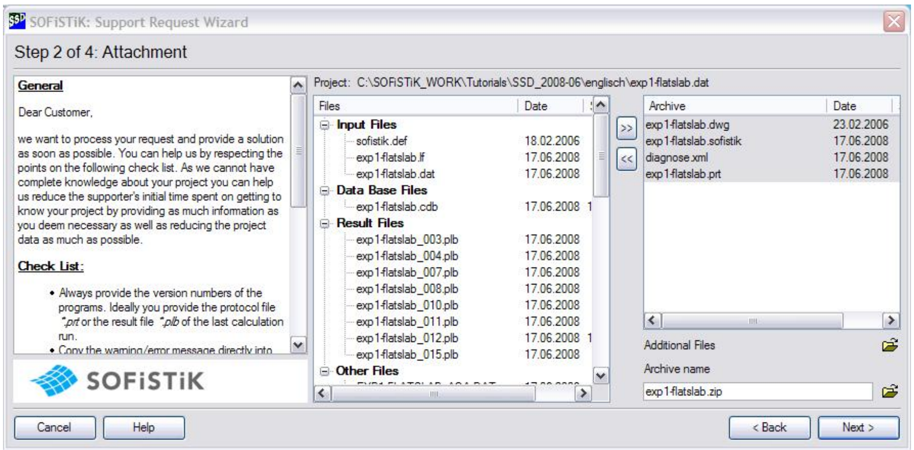


Figure 72: Step 2 of 4 Attachments

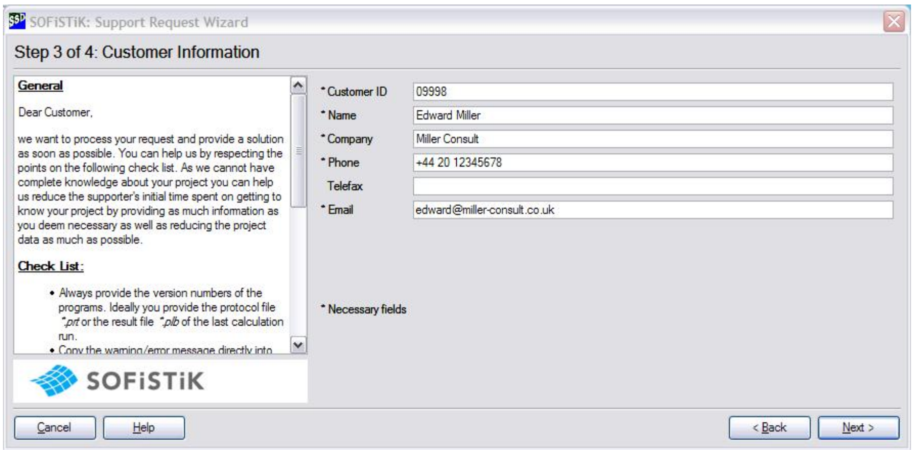


Figure 73: Step 3 of 4 Customer Information

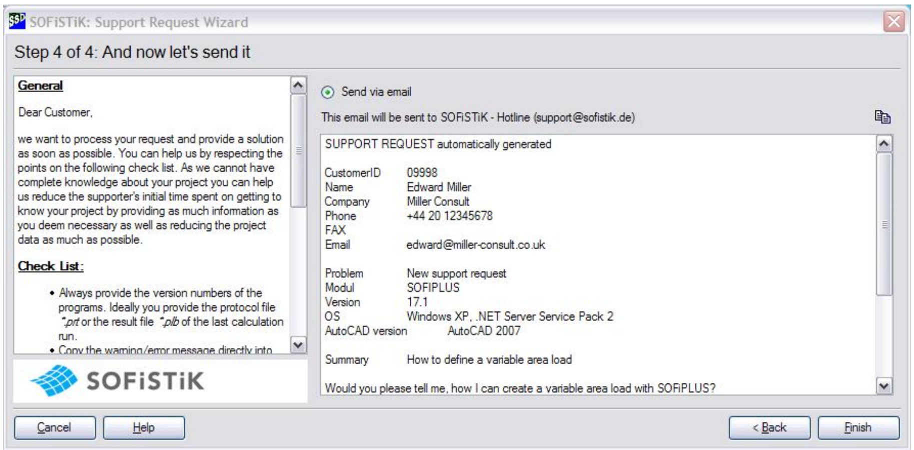


Figure 74: Step 4 of 4 Send mail to support

In almost every case, it is much faster to send your request via mail, because all relevant data including reduced data files are attached. This will fasten up your

We also have a SOFiSTiK Forum you may use. Going to our website www.sofistik.com you will find a link to this Forum. The Forum is for communication and information for all our customers. We also post news, tips and tricks.

Additional Tutorials are stored on our ftp-server http://ftp.sofistik.de/.

For alle customers with maintenance contract, we offer a special web based SOFiSTiK Online Portal. There you may use the FAQ Data Base and you may directly post your hotline Request in our System, which is the most comfortable and also the fastest way to contact our Hotline. For additional Information to our SOFiSTiK Online please contact us or your local sales partner.

8 Literature

[1] Betonkalender 2006, Teil II, Kapitel 7.1.6 Durchstanzen bei Flachdecken mit Randüberständen, S. 181-187

[2] *Beutel R.; Hegger J., Lingemann J.*; Stahlbetonflachdecke nach DIN 1045-1, Beton- und Stahlbetonbau 99 (2004), Heft 9, S. 754 – 760

[3] *Hünersen G.; Fritsche E.;* Stahlbau in Beispielen Berechnungspraxis nach DIN 18800 Teil 1 bis Teil 3, 4. Auflage Werner Verlag 1998

Further Information you will find in our SOFiSTiK manuals.