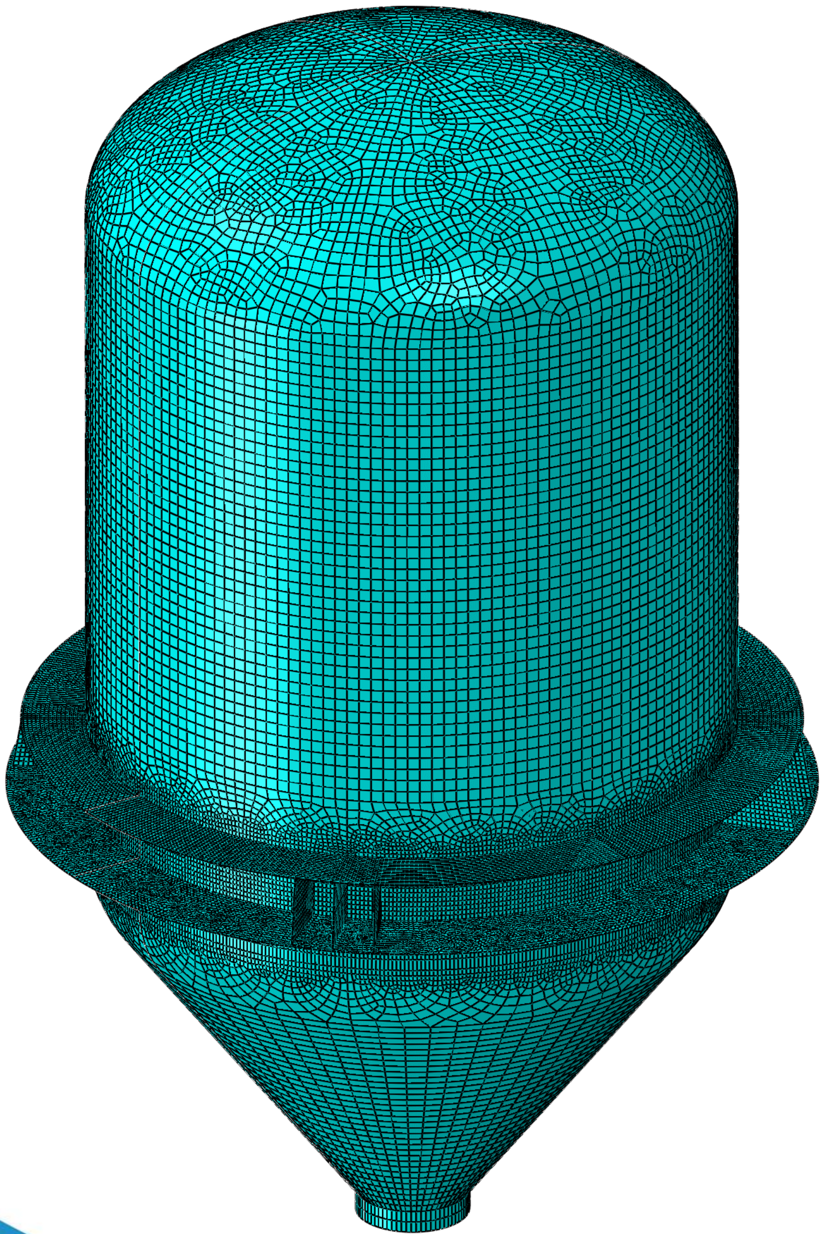
What to expect when you're expecting FEA – A guide to good practice



FE Consultants
31 Folkestone Street, Bowen Hills, QLD Australia, 4006
+61 7 3252 7400 I www.feconsult.com

1. Background

Finite Element Analysis (FEA) has transformed design procedures for engineers. Allowing more complex geometry, loading and restraint cases to be analysed. FEA is a powerful tool for engineers to simulate problems that would otherwise be too cumbersome for hand calculations. It has become more commonplace in industry over the last 5 years and while it has produced some incredible results, it has also produced some abysmal misuse.

This article has been written to help project engineers when commissioning analysis work, and to help junior engineers appreciate what should be reported as part of any FEA work.

2. Purpose

Too many requests for analysis begin with "We need you to FEA this". The problem with this approach is that it's not focussed on a solution to the mechanical problem. Firstly, FEA should only be used when solving problems that fit one of the following profiles:

1. No accurate solution exists via hand calculation;
2. The problem is non-linear (material, boundary conditions, geometry);
3. The problem is very large and would impose significant processing time using conventional techniques (excel, Matlab, MathCAD)

The outcome of finite element modelling is quantitative values for stress, strain, displacement, frequency and other measurable physical quantities. This can be thought of as simulation, because until engineering judgement has been applied to those results, they are otherwise interesting but useless.

The next step, which should link back to the original objective of the analysis, is assessing the results to determine acceptability. One of the best definitions for this assessment procedure is given by ASME Section VIII Division 2 – Part 5. This Standard has defined the requirements for design by analysis into demonstrating four protections that lead an engineer to determine if a result is acceptable. They are:

1. Plastic Collapse (does the component have sufficient strength to resist applied loads without uncontrolled plastic deformation)
2. Local Failure (a failure criterion concerned with elastic-plastic modelling using Mises stress formulation and triaxial strain)
3. Cyclic loading (resistance to fatigue from cyclic loading and ratcheting due to incremental plastic strain)
4. Buckling (numerical instability of a component, not related to strength)

While some problems only need to compare the relative behaviour of two different designs e.g. stiffness of two springs, in all instances, the focus should be on having the engineer draw conclusions from the results in the greater context of the problem.

3. Reference Information

Reference material should be reported to ensure that the geometry, materials and loadings applied in the analysis are correct. Most importantly, by providing this information it gives the chance for review by others. These items include drawings, process conditions, cycle data, item datasheets and any other supporting information for the analysis. If this information is not supplied, incorrect assumptions are often made, so in our experience, the more information the better. It leads to a better result, less obsolete communication, and less chance for variations due to unexpected requirements.

4. Materials

Material properties are a key input to the analysis and should be sourced from current and validated sources such as literature and standards. Internet based references can be poorly defined or erroneous. When applying a particular standard, the definitions within that standard should be applied e.g. ASME II Part D material properties for ASME Section VIII designs.

For designs at various temperatures, the properties should be tabulated to demonstrate the change with respect to temperature and how they have been applied in the model. This extends to young's modulus, thermal expansion, yield and tensile stress.

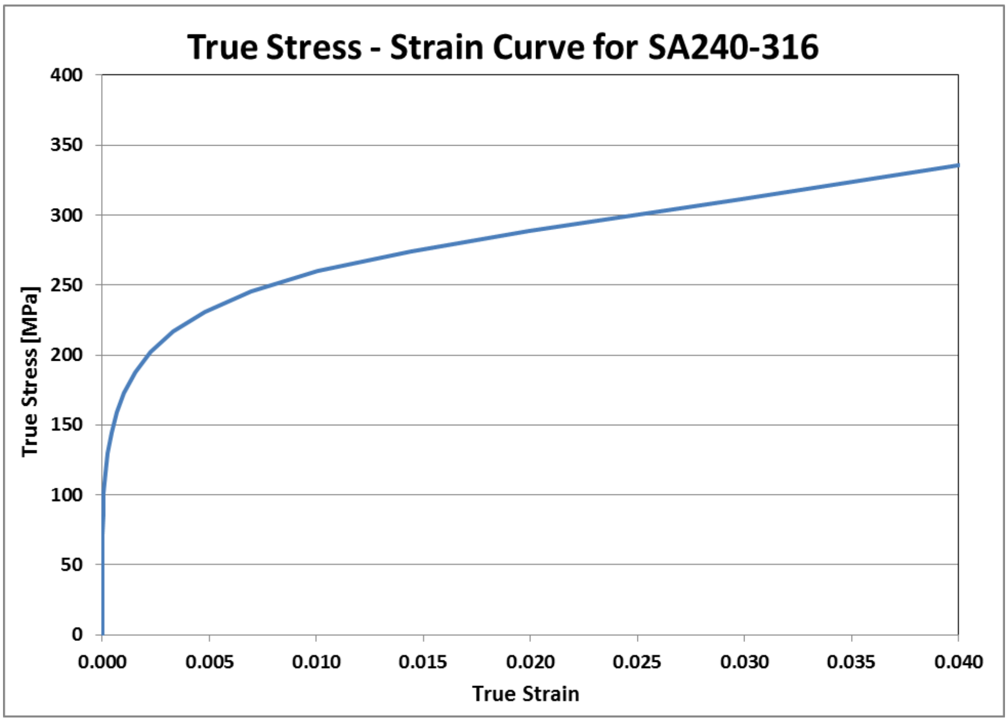


Figure 1: True Stress –Strain (Plastic) Curve for SA240-316 Developed from ASME II Part D

5. Modelling

There are many modelling inputs and assumptions that can be used in analysis of various components. Some of the fundamentals are shown below:

- Geometry simplifications
    - Solid > shell > beam
    - Symmetry (axi, planar and cyclic)
    - Shell-solid couplings
    - Sub-models
- Mesh details (element type, geometric order, formulation)
- Mesh density
- Material properties (as discussed above)
    - Young's Modulus
    - Poisson's Ratio
    - Density
    - Basic allowable stress
    - Yield strength
    - Tensile strength

  - Elongation
  - Coefficient of thermal expansion
  - Specific heat
  - Thermal conductivity
- Modelling simplifications
  - Lumped mass
  - Couplings
  - Rigid elements
  - Zero weight elements
  - Connector elements
  - Ties (rigidly connected elements)
  - Rigid parts
- Interactions
  - Convection coefficients and sink temperature
  - Radiation sources
  - Contact areas
  - Contact properties (normal and transverse)
  - Damping properties

6. Analysis Type

Demonstrating the acceptability of a design will often require numerous analysis types. The type of analysis completed should be clear as well as any key inputs to that solution. Some of the most common analysis types are shown in Table 1.

Table 1: Typical analysis types and related failure mechanisms

Type of Analysis	Related Failure Mechanisms			
	Plastic Collapse	Local Failure	Cyclic Loading	Buckling Stability
Linear perturbation (Linear Only)	Yes	Yes	Yes	No
General Static Solution (contact, non-linearity)	Yes	Yes	Yes	Yes
Eigenvalue (Buckling)	No	No	No	Yes
Natural Frequency (Harmonic)	No	No	Yes	No
Heat Transfer (Thermal Only)	Yes	Yes	Yes	No
Coupled Thermal-Stress (Structural Thermal)	Yes	Yes	Yes	No

7. Loads and Boundary Conditions

Depending on the analysis type selected above and the purpose of the analysis, will determine the relevant loads to be applied. Various loadings are shown below:

- Gravity (self-weight);
- Acceleration;

- Pressure;
- Hydrostatic pressure;
- Wind Load (uniform or circumferential distribution profile);
- Seismic Load (base acceleration or spectral response analysis);
- Initial Temperature, Convection, Radiation;

The documentation should show how the model was restrained:

- Displacement restraints;
- Rotation restraints;
- Applied regions;

8. Solution Validation

While FEA can complete sophisticated analyses, it still conforms to the principle that it's only as good as the inputs. Validation of the solution is critical to ensuring a correct result. All FE results should be compared to a known solution. This can take many forms, checking the solution can use any of the following:

1. Overall reaction loads
2. Membrane hoop stress in remote regions
3. Membrane longitudinal stress in remote regions
4. General review to an analytical standard or reference
    a. Nozzle loads – WRC
    b. Hold down design – Published reference e.g. Moss or Bednar
    c. Stress concentration – Solid mechanics texts
    d. Fracture – Literature, analytical solutions from standards
    e. Frequency – Solid mechanics literature and texts

9. Convergence

After validating that the loads, boundary conditions, material properties and analysis inputs are correct, the next step is to ensure the accuracy of the finite element solution. Discretisation of the solution using finite elements results in a loss of accuracy from the 'ideal' solution. The engineer's role is to ensure that the discretisation error is sufficiently small by completing a convergence check. This takes place by completing solutions at various levels of mesh refinement and reviewing convergence of a quantity such as stress as the solution becomes more refined. While all analyses should have a convergence study, some basic rules of thumb are applied as a common sense check for convergence:

- Minimum number of solid elements through the thickness of sections in bending. Depends on the order of the elements, quadratic elements obviously will not require as many as linear elements.
- Aspect ratio for elements can adversely affect results and basic recommendations are not to exceed 0.33 and 3. Mesh density and element order also come into play.
- Number of elements local to small details such as radii, holes, junctions and weld toes. Many small sharp features are discontinuities in the solution and as such the theoretical peak stress will approach infinity as the mesh is refined.
- Review of the stress gradient across a single element and the magnitude of the change.

All of the above techniques are rules of thumb which give some evidence that convergence should have been achieved, they are however, not a replacement for a convergence study. Additional information on calculation methods and good practice is provided in AS1210-2010 Appendix I2. It should also be noted that subject to the purpose of the analysis, convergence of a particular variable may not be relevant to the outcome of the analysis. For example, sometimes elements are modelled only to utilise their elastic stiffness, in those cases discretion is obviously required.

10. Additional Considerations

The number of items are provided below as a reference of additional considerations that should be reviewed as part of good practice FE analysis:

- Corrosion, forming and under-tolerance allowances;
- Suitability of boundary conditions (rigid or compliant);
- Number of through wall elements in solid element models;
- Element aspect ratio and stress gradation across elements;
- Buckling stability and initial imperfections;
- Temperature considerations (modulus and yield);
- Environmental load cases;
- Assembly and disassembly load cases;
- Transportation and lifting load cases;
- Far field stress checks;
- Global reaction checks;
- Serviceability limits (deflection/vibration);
- Consideration of all failure modes;
- Application of repudiable assessment criteria or design code;

11. Example Documentation

Provided in the following Appendix is an example of FE analysis of a particular component. The accompanying documentation aims to fulfil the requirements of a best practice FE report for the purpose of which the component was analysed.

The component being analysed is the example problem from Part 5.2 of ASME PTB-3-2010 (Division 2 Example Problem Manual).

Index of Appendices

Appendix A – Example Finite Element Analysis Summary

Appendix A.1. Model Details

The finite element method was applied to analyse the plastic collapse criteria of ASME VIII Division 2 Part 5 for the vessel top head and shell region. If intending to completely satisfy code requirements, one would also need to satisfy protection against local failure and cyclic loading.

The purpose of this analysis is to demonstrate the plastic collapse protection requirements of the detail when subject to design loads. The only design load considered in this case is internal pressure. Nozzle loads, external pressure and occasional loads are not required to be reviewed in this instance. Elastic assessment criteria will be used.

A commercial, verified finite element analysis (FEA) code was used to undertake the analysis. The following details summarise the approach and key parameters applied for this analysis.

Software Package:	Abaqus 6.14
Model File Names:	Vessel_Nozzle_Elastic.cae
Units Considered:	Length: mm; Mass: T; Force: N; Stress / Pressure: MPa; Temperature: K.
Material Model:	Elastic - Ref. Table A.1 for Material Properties
Section Assignments	Refer Table A.2 and Figure A-1 for Section Assignments
Large Displacement Theory:	No
Constraints Applied:	Nil
Boundary Conditions Applied:	Refer to Figure A-2
Interactions Applied:	Nil
Element Type & Quantity	• 4,689 Elements, Type CAX8R: An 8-node biquadratic axisymmetric quadrilateral, reduced integration.
Load Cases:	See Table A.3 for load case combinations.
Analysis Performance Criteria:	• Plastic collapse assessment per ASME Section VIII Div. 2 Section 5.2.2

Table A.1: Material Models

Material Name	SA-105	SA-516-70N
Density	7.9×10^{-9} T/mm^3	7.9×10^{-9} T/mm^3
Young's Modulus	199,583 N/mm^2	200,962 N/mm^2
Poisson's Ratio	0.3	0.3
Coefficient of Thermal Expansion	N/A	N/A
Thermal Conductivity	N/A	N/A
Specific Heat Capacity	N/A	N/A
Plasticity	N/A	N/A

Table A.2: Section Assignments

Section Name	Material Model Name	Section Type
SA-105	SA-105	Solid, Homogeneous
SA-516-70	SA-516-70	Solid, Homogeneous

Table A.3: Applied Loads and Load Case Combinations

Load Type	Internal Pressure
Internal Pressure	2.8958 MPa
Nozzle Pressure Thrust	6.773 MPa (applied to gasket face)

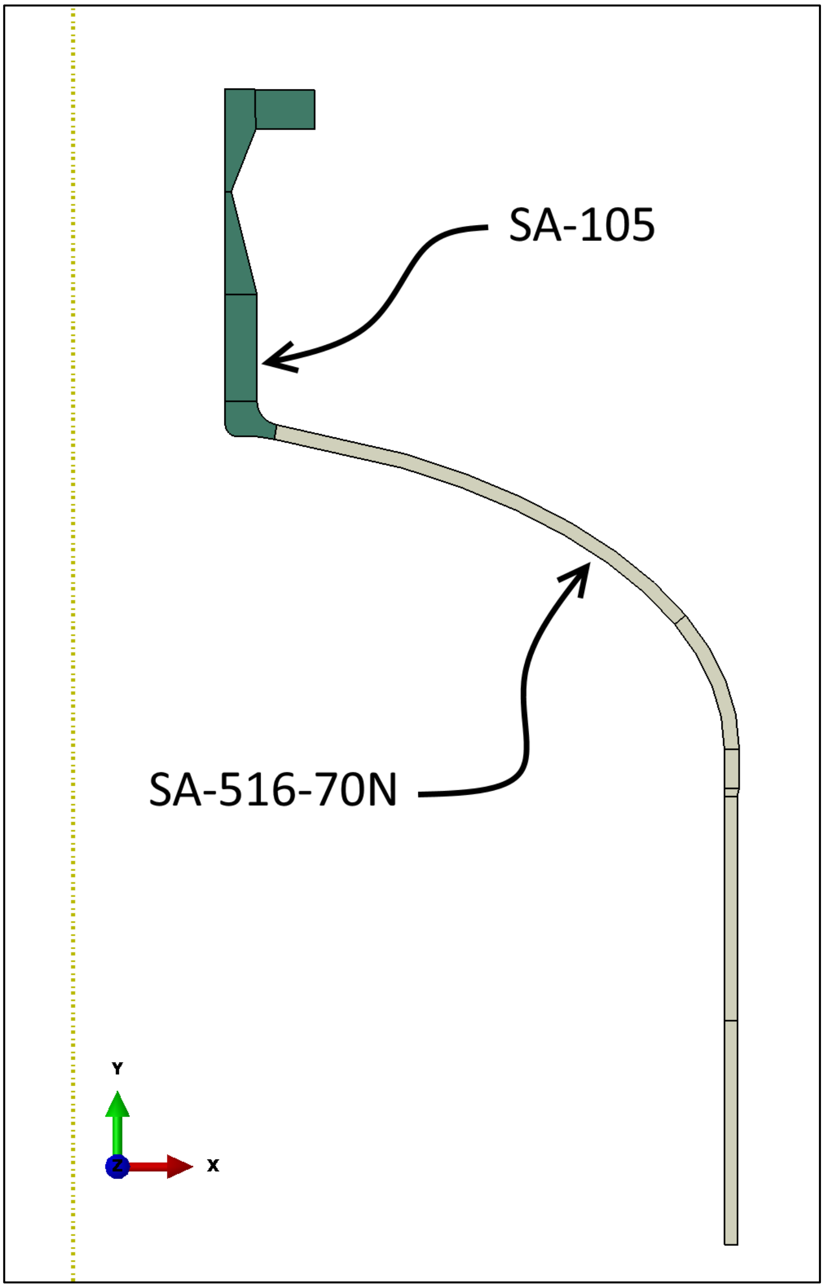


Figure A-1: Material Sections

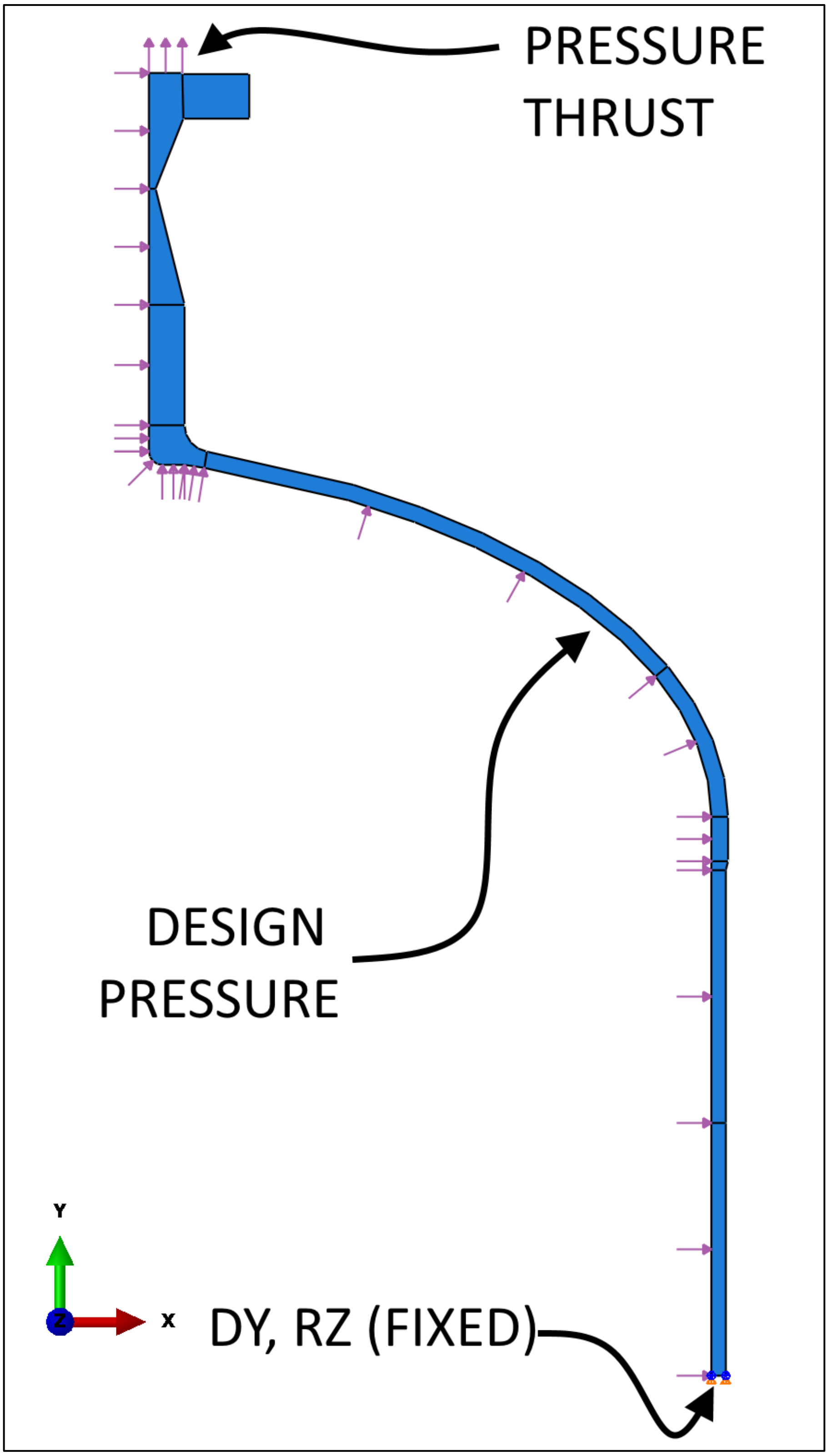


Figure A-2: Applied Loads and Boundary Conditions

Appendix A.2. Mesh Details

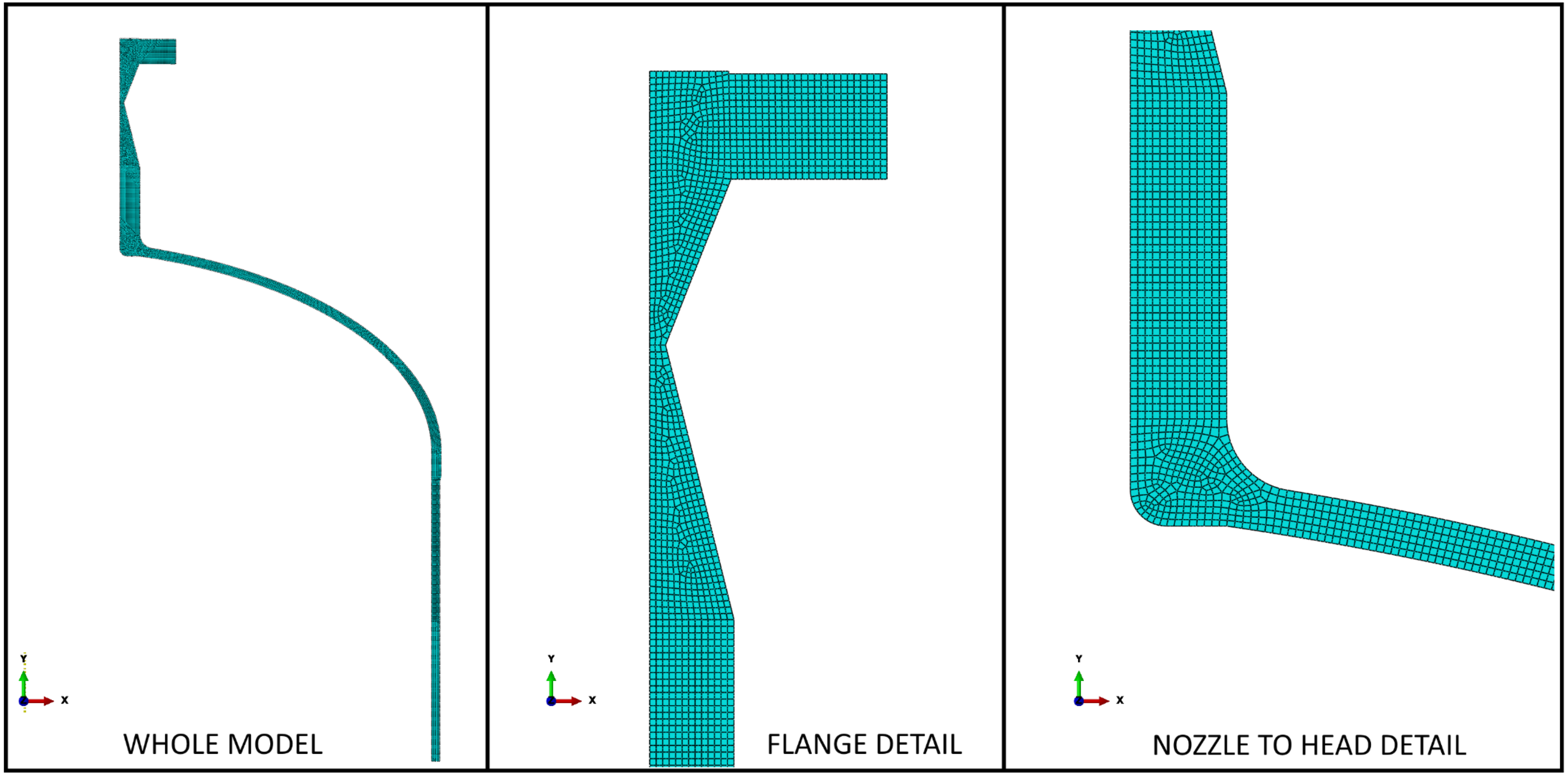


Figure A-3: Mesh Refinement Details

Appendix A.3. General Stress Profile

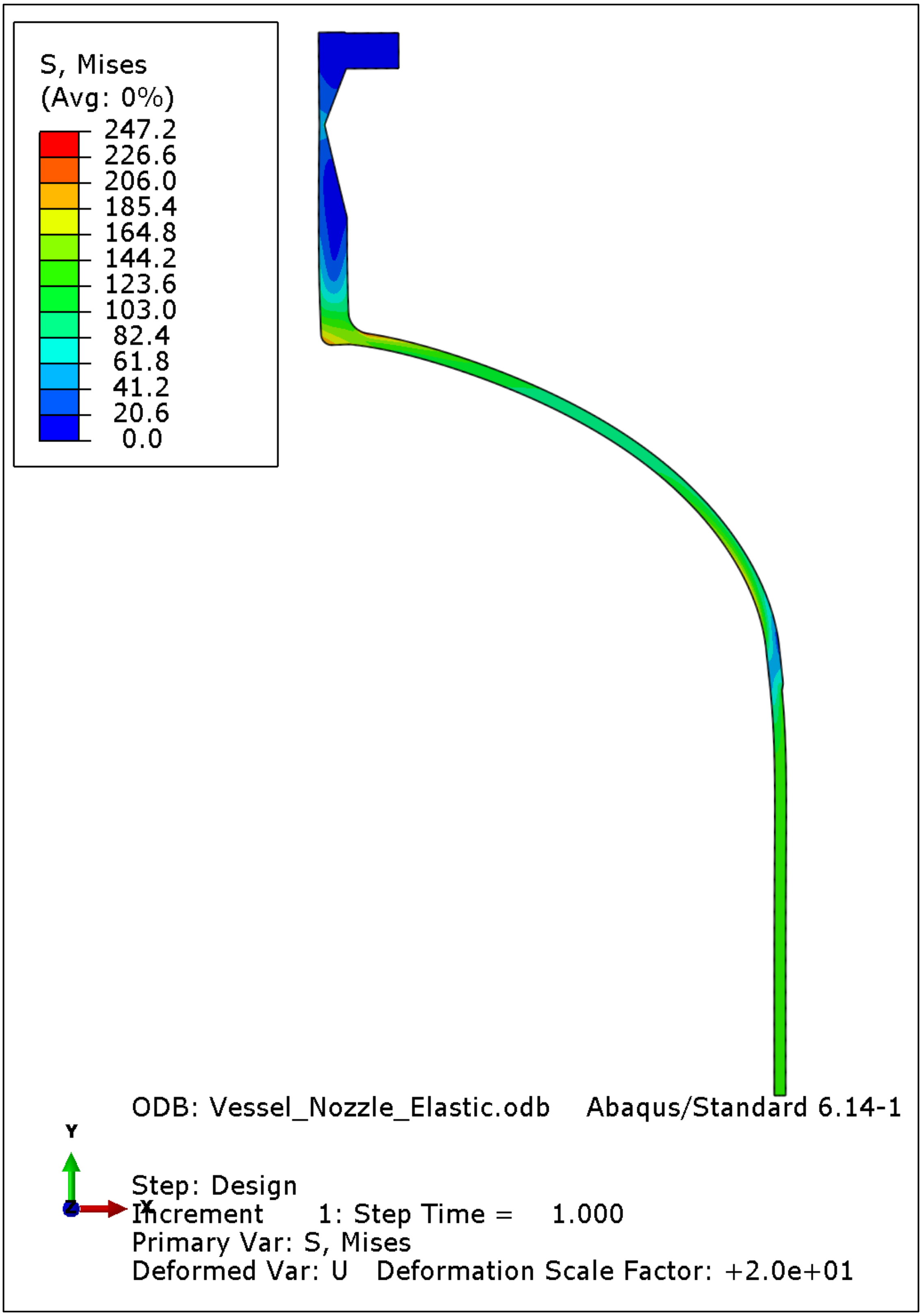


Figure A-4: Mises Stress Profile under Internal Pressure (Results not averaged)

Appendix A.4. Summary of FEA Results

A concise summary of the FEA results should be provided. If the FEA has been completed to review compliance against a standard, both the criteria limit and calculated result should be specified. This provides an indication of how close the design is to the specified limits, which may have a bearing on the conservatism built into any assumptions that have gone into the analysis.

An example of an Elastic assessment of plastic collapse, per AS1210 criteria, is shown below. In this procedure, visual review of the results does not qualify as a proper assessment. Stresses are linearised using a Stress Classification Line (SCL) and broken down into components of membrane, bending and secondary stress. SCLs are applied at strategic locations and the classification of the results require further understanding of the location and the type of load applied.

The allowable stress limits are taken from Table B1(B) of AS1210 for the design temperature.

Classification	Allowable Stress (MPa)
f_l	177
f_l+f_b	177
$f_l+f_b+f_g$	354

Stress Classification Line (SCL)	Stress Classification Assessed	Calculated Stress	Ratio to Allowable
1	f_l	46.0	26%
1	$f_l+f_b+f_g$	167.7	47%
2	f_l	46.4	26%
2	$f_l+f_b+f_g$	165.0	47%
3	f_l	56.2	32%
3	$f_l+f_b+f_g$	193.3	55%
4	f_l	56.4	32%
4	$f_l+f_b+f_g$	190.7	54%
5	f_l	132.8	75%
5	$f_l+f_b+f_g$	160.4	45%
6	f_l	131.1	74%
6	$f_l+f_b+f_g$	158.1	45%

All stress results are considered acceptable per AS1210-2010 and protection against plastic collapse is satisfied.

FEC FE Consultants

31 Folkestone Street, Bowen Hills, QLD Australia, 4006
+61 7 3252 7400 I www.feconsult.com