Finite Element Analysis (FEA) Guidelines

AB–520

Regarding the Use of FEA to Support a Pressure Equipment Design Submission

Edition 2, Revision 0 - Issued 2023-10-26

Table of Contents

FOREWORD ..... **iii**

1.0 INTRODUCTION **1**

2.0 SCOPE **2**

2.1 Governing Code of Construction 2

2.1.1 Choice of Construction Code for New Construction 2

2.1.2 Choice of Construction Code for Repairs and Alterations 2

2.2 Other Analysis Types 3

2.3 Acceptance by Jurisdictional Authority 3

3.0 DEFINITIONS **4**

4.0 REFERENCE PUBLICATIONS **5**

5.0 GENERAL GUIDELINES **6**

5.1 FEA Operator Qualification Guidelines 6

5.1.1 Code Requirements to Be Considered 6

5.1.2 Additional Information May Be Requested 6

5.2 Selection of Analysis Software 6

5.3 FEA Report 7

5.3.1 Executive Summary 7

5.3.2 Introduction 7

5.3.3 Description and Verification of the Model 8

5.3.4 Presentation of Analysis Results 8

5.3.5 Conclusion 8

5.3.6 Certifying Statement by Qualified FEA Operator 8

5.3.7 Miscellaneous FEA Report Considerations 9

5.3.7.1 Document Control and Page Numbering 9

5.3.7.2 Reference to Figures and Tables 9

5.3.7.3 Recommended Report Media 9

5.3.7.4 Attachment of Drawings and User's Design Specification 10

5.3.7.5 Attachment of Supporting Documentation 10

6.0 FEA MODEL CONSIDERATIONS **11**

6.1 Modeling Geometry 11

6.1.1 Relation to Drawings 11

6.1.1.1 Modeling Requirements 11

6.1.1.2 Reporting Considerations 11

6.1.2 Simplification of Geometry by Lossy Approximation 11

6.1.2.1 Justification for Lossy Approximation 12

6.1.2.2 Report Considerations for Lossy Approximation 12

6.1.3 Simplification of Geometry by Lossless Reduction 13

6.1.3.1 Justification Not Required 13

6.1.3.2 Report Considerations for Lossless Reduction 13

6.2 Model Discretization (Meshing) 13
6.2.1 Selection of Elements 14
6.2.1.1 Element Type, Geometry, and Behavior Characteristics 14
6.2.2 Mesh Refinement 15
6.2.2.1 General Guidelines 15
6.2.2.2 Automated Adaptive Remeshing 15
6.2.2.3 Report Considerations 16
6.2.3 Contact Interfaces 16
6.2.3.1 Report Considerations 16
6.3 Characteristics of the Material Model 17
6.3.1 Sources of Information 17
6.3.2 Report Considerations 17
7.0 MODEL VERIFICATION AND VALIDATION 18
7.1 Verification of Inputs 18
7.1.1 Report Considerations 18
7.2 Verification of Mesh Adequacy 18
7.2.1 Boundary Conditions and Loading for Mesh Adequacy 19
7.2.2 Verification of Mesh Adequacy by Visual Inspection 19
7.2.2.1 Report Considerations 19
7.2.3 Verification of Mesh Adequacy by Nodal Stress Comparison 20
7.2.3.1 Report Considerations 21
7.2.4 Verification of Mesh Adequacy by Convergence Study 21
7.2.4.1 Report Considerations 22
7.3 Verification of Model Behavior 22
7.3.1 Verification of Response to Boundary Conditions 23
7.3.1.1 Report Considerations 23
7.3.2 Visual Inspection of Deformed Shape 23
7.3.2.1 Report Considerations 23
7.3.3 Visual Inspection of Stress Distribution 24
7.3.3.1 Report Considerations 24
7.3.4 Verification of Model Stresses by Hand Calculation 25
7.3.4.1 Report Considerations 25
7.3.5 Comparison of Model Strains to Physical Behavior 25
7.3.5.1 Report Considerations 26
8.0 RESULTS OF THE DESIGN ANALYSIS 27
8.1 Required Report Contents 27
9.0 REVISION LOG 28

FOREWORD

This document does not impose supplementary requirements beyond those outlined in established codes, standards, and regulations. Its primary objective is to offer guidance for conducting effective Finite Element Analysis (FEA) and preparing comprehensive FEA reports, aiming to enhance the accuracy and diligence.

This document was composed through extensive collaboration and consultation with industry professionals and experts in the field of FEA, reflecting a sound engineering practice.

FEA operators, designers, manufacturers, and regulatory organizations have the option to use this document as a valuable resource for preparing and evaluating FEA when it is employed to justify pressure equipment designs.

It's important to note that this document is not mandatory, as certain design scenarios may warrant additional considerations, whilst in rare scenarios some aspects of this guideline might be deemed excessive or unnecessary.

The utilization of this guideline should be carefully considered and aligned with the professional judgment of the designer and the FEA analyst.

Ultimately, the acceptance of any FEA report remains subject to approval by the relevant jurisdictional regulatory organization.

The next reaffirmation of AB-520 is scheduled for 2028.

1.0 INTRODUCTION

When pursuing registration for pressure equipment designs, it is essential to provide the required calculations and information to ensure the design is suitable for registration. Simple designs have established equations in construction codes to determine component thickness based on allowable stresses. However, for some complex designs where these equations are not applicable, the construction codes offer other ways to justify the design. These methods can include doing a proof test where a test piece is pressurized to a certain level, using calculations and concepts from other published codes, or using a computer-based approach called finite element analysis (FEA) to simulate how the component behaves under different loads, including pressure. While finite element analysis (FEA) is a potent tool with high reliability, it demands a substantial knowledge of the analysis technique, the specific software employed, the relevant construction code requirements, and a broader understanding of pressure equipment design. The accuracy of FEA results is dependent on the quality of input information, compelling the consideration of measures to prevent input errors. Prior to assuming FEA outcomes as valid justifications for pressure service, the model's performance should be thoroughly verified.

To obtain regulatory approval for the design, both the modeling process and analysis should be meticulously documented. This documentation assures the regulator that the operator possesses adequate knowledge and experience, the equipment has been correctly modeled, and the analysis conducted accurately. Ultimately, this substantiates that the component aligns with the specified service conditions within the scope of the applicable construction code.

It's essential to understand that using finite element analysis for equipment design justification is permissible only when the applicable construction code lacks appropriate equations for the specific geometries under analysis. When considering the use of finite element analysis as a method for design justification, pressure equipment designers and manufacturers are strongly advised to proactively engage with the relevant jurisdictional organizations before relying on this approach to obtain registration. This proactive step ensures alignment with regulatory requirements and prevents potential issues during the registration process.

ABSA guideline documents are periodically reviewed to ensure that they are aligned with current industry practices. We would welcome any suggestions you have to improve this document. Please provide your comments to:

Lou Petrusevski
Design Survey Manager
petrusevski@absa.ca

2.0 SCOPE

This document provides guidance and recommendations for qualification of FEA operators, selection of software, application of modeling practices, verification and validation of models, and documentation of required analyses when ASME Section VIII-2, Part 5: Design by Analysis Requirements is used to justify the design of a pressure equipment component for regulatory registration.

Reference to the use of ASME Section VIII-2, Part 5 for design justification includes its use as permitted by other adopted codes of construction, such as by:

- ASME B31.1, §104.7.2(d)
- ASME B31.3, §304.7.2(d)
- ASME Section I, Part PA[1]
- ASME Section VIII-1, U-2(g) and Mandatory Appendix 46[2]

2.1 Governing Code of Construction

Selection of the code of construction is outside of the scope of this document, and is to be in accordance with applicable legislation and regulation at the location of installation; however, the following guidance is provided with respect to equipment intended for use in Alberta that falls within the jurisdiction of the Pressure Equipment Safety Regulation. Please defer to the referred documents in determining the governing code of construction.

2.1.1 Choice of Construction Code for New Construction

For new construction, selection of a code of construction for pressure equipment is guided largely by Alberta legislation. Pressure equipment must comply with all applicable requirements of the Safety Codes Act, the Pressure Equipment Safety Regulation (PESR), the CSA B51: Boiler, Pressure Vessel, and Pressure Piping Code, and applicable codes and standards listed in section 6 of the PESR.

2.1.2 Choice of Construction Code for Repairs and Alterations

For repairs and alterations, guidance in selecting an appropriate code of construction can be found in the following ABSA documents, as applicable:

[1] ASME Section I, Part PA does not specifically refer to ASME Section VIII-2, Part 5, but rather to ASME Section VIII-2 in its entirety.

[2] ASME Section VIII-1, subparagraph U-2(g) refers to Mandatory Appendix 46 as one of three methods permitted for use when design rules are not prescribed by the code.

- AB-513: Pressure Equipment Repair and Alteration Requirements

- AB-535: Requirements for Alteration Design Registration Based on Fitness-for-Service

2.2 Other Analysis Types

This document is intended to apply to the use of finite element analysis techniques that are used to analyze for stresses induced due to the application of pressure and other loads, and to determine that the equipment design meets the intent of the applicable code of construction with respect to stress integrity.

This document does not address the use of finite element analysis for evaluation of the pressure design of a pressure equipment component by analysis methods that are outside of the scope of ASME Section VIII-2, Part 5, such as for transient thermal analyses, heat transfer problems, or the determination of thermal stresses. When such analyses are permitted by the governing code of construction or are required for regulatory registration of a design, the principles of this document may be considered, as applicable.

2.3 Acceptance by Jurisdictional Authority

Design by analysis is generally only permitted in cases where the governing code of construction does not already provide design rules suited to the component being considered, and allows for design by analysis as an alternative. When design by analysis is being considered and regulatory acceptance of the design is required, it is strongly advised that designers check with the appropriate jurisdictional authority to ensure that it is acceptable prior to committing to its use.

It should be noted that despite that ASME Section VIII-2 allows the use of Part 5: Design by Analysis Requirements to overrule its own Part 4: Design by Rules Requirements for certain classes of vessels, the Pressure Equipment Safety Regulation specifically prohibits the use of this provision in Alberta.

3.0 DEFINITIONS

FEA Operator – a suitably qualified person who undertakes and documents a finite element analysis, or who supervises other persons who undertake and document such an analysis; but in either case accepts professional responsibility for it

Finite Element Analysis (FEA) – a numerical method that is used to solve problems in engineering; the goal of the analysis is to approximate the physical behavior of a system by modeling the system using a large quantity of small, simple parts that interact with each other in a way that can be described by a large number of relatively simple mathematical equations. Loads, boundary conditions, and other rules are applied to modeled parts, and software is used to solve the series of equations in order to determine how those parts interact with each other, and thus how the overall system can be expected to physically respond to the applied loads.

Governing Code of Construction – the code of construction which governs the design of the pressure equipment component being considered, and which is listed in the Pressure Equipment Safety Regulation

Lossless Reduction – *a* reduction of the complexity of a modeled component which does not result in any loss of accuracy; a typical example is modeling a 3D axisymmetric part as a 2D section with specified axisymmetry, when the 2D representation still accurately reflects the 3-dimensional modeled part and does not represent any loss of geometric detail

Lossy Approximation – *a* modification made to a modeled component which simplifies it in order to make it easier to analyze, but which represents a loss of modeling accuracy, and for which an experienced FEA operator's judgment is required to establish that the loss of accuracy is acceptable for the purpose for which the model is being used

Model Validation – steps undertaken to show that a modeled component's behavior adequately represents the expected behavior of the physical component, thus justifying the use of the model in predicting behavior under other loading conditions; a typical example would be undertaking a simplified analysis and comparing its result to the result of hand calculations, for a region where stresses can easily be predicted

Model Verification – steps undertaken to ensure that a component has been suitably modeled, especially by double-checking modeled geometry and other characteristics; typically involves a separate verification step in which modeled dimensions, material characteristics, and similar inputted values are compared to the original drawing or reference material in a separate sitting, or by a separate individual

4.0 REFERENCE PUBLICATIONS

The following documents are referred to herein, and are listed here for reference only. These documents are listed with reference to the edition used in authoring this document, though they may be amended from time to time; in all cases, please refer to the latest applicable edition of referred documents.

AB-513: Pressure Equipment Repair and Alteration Requirements, Edition 4, Revision 0; Alberta Boilers Safety Association (ABSA), August 2023

AB-535: Requirements for Alteration Design Registration Based on Fitness for Service, Edition 2, Revision 2, Alberta Boilers Safety Association (ABSA), December 2021

ASME B31.1: Power Piping, 2022 Edition, The American Society of Mechanical Engineers (ASME), October 2022

ASME B31.3: Process Piping, 2022 Edition, The American Society of Mechanical Engineers (ASME), June 2021

ASME Boiler and Pressure Vessel Code, 2021 Edition / Section I: Rules for Construction of Power Boilers, The American Society of Mechanical Engineers (ASME), July 2021

ASME Boiler and Pressure Vessel Code, 2021 Edition / ASME Section VIII, Division 1: Rules for Construction of Pressure Vessels, The American Society of Mechanical Engineers (ASME), June 2021

ASME Boiler and Pressure Vessel Code, 2021 Edition / ASME Section VIII, Division 2: Alternative Rules, The American Society of Mechanical Engineers (ASME), June 2021

CSA B51-19: Boiler, pressure vessel, and pressure piping code, Canadian Standards Association, operating as CSA Group (CSA), March 2019

Safety Codes Act: Revised Statutes of Alberta 2000, Chapter S-1: Office Consolidation, Alberta King's Printer, Province of Alberta; consolidated with amendments through November 2022

Safety Codes Act / Pressure Equipment Safety Regulation (Alberta Regulation 49/2006): Office Consolidation, Alberta King's Printer, Province of Alberta; with amendments through Alberta Regulation 195/2015, January 2016

5.0 GENERAL GUIDELINES

5.1 FEA Operator Qualification Guidelines

An FEA operator should be a Professional Engineer meeting the requirements of the Pressure Equipment Safety Regulation, and should have the following qualifications[3]:

- Knowledge in the design of the type of pressure equipment being considered
- Formal training in the theory and application of FEA
- Four (4) years experience in the use of FEA for the design of pressure equipment
- Formal training in the use of the particular software product being used for the analysis

5.1.1 Code Requirements to Be Considered

It should be noted that the governing code of construction may establish additional requirements that need to be met for qualifications, experience, and documentation of the competency for persons performing design activities. Any such additional requirements established by the code of construction also need to be met, insofar as they are applicable.

5.1.2 Additional Information May Be Requested

As permitted by subsection 15(1)(j), 16(1)(i), or 17(1)(c) of the Pressure Equipment Safety Regulation, a design surveyor may request that the submitter provide a copy of the FEA operator's curriculum vitae (CV) or other additional information about their academic credentials and experience with respect to the use of FEA for pressure equipment design.

5.2 Selection of Analysis Software

The software chosen for finite element analysis should be suitably capable of:

- Modeling the geometric and material characteristics of the pressure equipment being analyzed
- Simulating the application of all pressure and other required loads
- Accurately and quantitatively undertaking analyses as required to meet the requirements of ASME Section VIII-2, Part 5 and of the governing code of construction

[3] In some cases, an equipment manufacturer or other organization may establish additional qualification requirements in their quality control program, integrity management system, or by contractual agreement, etc.

- Producing output adequate to meet the requirements of this document and suitable for inclusion in a report that can be printed to paper or equivalent electronic media

5.3 FEA Report

5.3.1 Executive Summary

The report should begin with an ‘Executive Summary’ which contains:

1. A short description of the component being analyzed
2. An indication of the governing code of construction, of the reason for using FEA to justify the design, and of the design conditions that are being justified
3. A brief description of the FEA model used
4. Conclusions related to the FEA results supporting the design, including:
   - A brief description as to how the model behavior was validated
   - A brief description of the level of accuracy attained in the analysis
   - A clear assertion that the analyzed component has been justified for service at the proposed design conditions, in accordance with the governing code of construction

5.3.2 Introduction

The report should contain an ‘Introduction’ section which contains:

1. A short description of the component being analyzed
2. The specific design conditions being justified
3. An indication of any scope limitations for the finite element analysis (e.g., if the FEA is being used for only a certain part of the component, or for certain design conditions, or whether it's used only to justify pressure design or secondary loading)
4. An indication of the governing code of construction, including reference to the paragraph in the governing code of construction which permits use of FEA
5. The reason for using FEA analysis (typically, that the governing code of construction does not provide appropriate design rules)
6. Drawing numbers (including revision levels) used to generate the component model[4]
7. The name, publisher, and specific version or release number of the computer software used for the analysis, including any software used for modeling, meshing, solving, and post-processing (analysis of results)

[4] The complete set of drawings used to generate the model is also required to be included as an appendix or attachment to the report; see 5.3.7.4.

5.3.3 Description and Verification of the Model

The report should include the content required by Section 6, 'Model Requirements', and Section 7, 'Model Verification and Validation', in order to describe and justify use of the numerical model used for the analysis.

5.3.4 Presentation of Analysis Results

The report should include the content required by Section 8, 'Results of the Analysis' to describe the results of each of the analyses required by ASME Section VIII-2, Part 5: Design by Analysis to justify the design.

The report should discuss any unusual findings or the exclusion of any results that were ignored, if applicable.

5.3.5 Conclusion

The report should contain a conclusion which specifically indicates that the component design was justified for the proposed service conditions in accordance with the governing code of construction.

5.3.6 Certifying Statement by Qualified FEA Operator

The report should be certified by the FEA operator who conducts or supervises the analysis and takes professional responsibility for it.

The below certification statement is to be placed on or in the report, and is to be accompanied by the FEA operator's P. Eng. stamp, signature, and date:

I, *(full name)*, the undersigned Professional Engineer, certify that:

- I have knowledge in the design of the type of pressure equipment being considered.
- I have formal training in the theory and application of finite element analysis.
- I am experienced in the use of finite element analysis for the design of pressure equipment.
- I have had formal training in the use of the finite element analysis software used for this analysis.

- I conducted or supervised the analysis described herein, and that it is accurate and suited to the intended purpose.
- I prepared, or oversaw and have thoroughly reviewed this documentation of the analysis, and accept professional responsibility for it.

I further certify that the analysis described herein meets the requirements of ASME Section VIII-2, 2021 Edition[5], Part 5: Design by Analysis, and that it adequately justifies service at the design conditions presented herein in accordance with *(governing code of construction)*.

5.3.7 Miscellaneous FEA Report Considerations

5.3.7.1 Document Control and Page Numbering

The report should have an assigned document number and revision level indicated on the first page or cover page, and on every subsequent page.

All pages of the report should be numbered with an indication of the total number of pages.

5.3.7.2 Reference to Figures and Tables

All figures and tables included in the report should be labeled in a way that they can be uniquely identified and referred to by the report text as may be required to demonstrate compliance with the governing code of construction, ASME Section VIII-2 Part 5, and this document.

5.3.7.3 Recommended Report Media

The report is recommended to be submitted electronically in PDF format when electronic submission is permitted by the jurisdiction or other organization requiring the submission.[6]

[5] The indicated code edition shall be adjusted to suit the edition of ASME Section VIII-2 used for the analysis.

[6] PDF files can most suitably be generated by printing to a PDF file directly from the software application used to create the report, rather than by scanning from a hard copy. Printing directly to an electronic file tends to optimize image quality while keeping the associated file size to a minimum.

5.3.7.4 Attachment of Drawings and User's Design Specification

A complete set of mechanical drawings should be included as a part of the report, as would be required to produce the model used for the analysis. The drawing numbers including revision levels should match those listed in the 'Introduction' as required by section 5.3.2 above. The included drawings should be legible[7], and should include all details and dimensions as required to accurately reproduce the model used for the analysis.

The drawing should indicate any corrosion allowances, machining allowances, geometric tolerances, etc., as may be required to generate a worst-case model of the component for analysis.

If required by the governing code of construction or in order to support fatigue analysis or exemption from fatigue analysis, a User's Design Specification or its equivalent should be included in the report.

5.3.7.5 Attachment of Supporting Documentation

Any additional calculations or other documentation that are required to justify or support the analysis should be attached as a part of the report.[8]

Regular code calculations and other analyses (proof test reports, etc.) that are typically included in a design registration application do not need to be included in or referred to by the report, unless they are needed to support the contents of the report.

[7] Inclusion of drawings as inset figures in a report is only suitable if all necessary details and dimensions remain legible and easily readable without magnification when the page is printed to an 11"×17" or smaller page size.

[8] It is the intent that the report will serve as a self-contained and compelling justification for the portion of the design to which it pertains, and that it can be reviewed and verified by a third party without requesting additional drawings, calculations, or other information.

6.0 FEA MODEL CONSIDERATIONS

6.1 Modeling Geometry

6.1.1 Relation to Drawings

6.1.1.1 Modeling Requirements

The model should accurately represent the geometry specified by the component drawings attached to the report but may be simplified as permitted below in sections 6.1.2, ‘Simplification of Geometry by Lossy Approximation’, and 6.1.3, ‘Simplification of Geometry by Lossless Reduction’.

The modeled geometry should account for the removal of material as may be permitted by any corrosion allowances, machining allowances, geometric tolerances, etc., that are permitted by the component drawings, in order to generate a worst-case model for analysis.

6.1.1.2 Reporting Considerations

The report should include a section describing the model and relating it to the drawings, and should include:

- An indication that the model was created from the drawings, including the drawing number and revision level
- An indication of corrosion allowances, machining allowances, geometric tolerances, etc., that have been incorporated into the model in order to generate a worst-case model for analysis; or an indication that there are no such allowances or tolerances that are applicable

6.1.2 Simplification of Geometry by Lossy Approximation

Lossy approximation[9] refers to the simplification of complex model geometry in a way that the computer model becomes only an approximate representation of the physical component, due to a loss of geometric or other detail which is judged by the FEA operator to not have a significant effect on the accuracy of the model’s behavior, with consideration given to the purpose for which it is being used.

[9] ‘Lossy’ refers to the fact that in modeling the component, there is an associated ‘loss’ of information about its real-world geometry.

An example of lossy approximation could be the removal of a small hole or other minor geometric detail from a low-stress area, when the FEA operator has experience or other means of judging that the simplification is made at a location where stresses are known to not govern the results of the analysis, and that the simplification will not have a material effect on the overall behavior of the model.

6.1.2.1 Justification for Lossy Approximation

Simplification of model geometry in a way that produces a lossy approximation of the physical component should be justified by the FEA Operator.

In some cases, the results of the analysis may need to be validated to show that the simplification did not have a material effect on the results.

6.1.2.2 Report Considerations for Lossy Approximation

Any simplifications made to the model which reduces its complexity in a way that results in any loss of accuracy in model behavior should be described in the report. The report should include:

- A description of the simplification being made to the model geometry
- Any discussion necessary to justify the simplification
- A clear statement to indicate that the simplification is made at a location where stresses are known to not govern the results of the analysis
- A clear statement to indicate that the simplification will not affect the stresses at remote locations of the model, and will not materially affect the overall behavior of the model

The means and extent of justification and of any associated validation needed for the use of a model that has been simplified by lossy approximation is left to the reasonable discretion of the FEA operator. In the case that design registration with a regulatory authority is required, such justification and validation should be acceptable to the surveyor undertaking the review.

6.1.3 Simplification of Geometry by Lossless Reduction

Lossless reduction[10] refers to simplification of complex model geometry in a way that the model is still an accurate representation of the physical component, and would not affect the results of an analysis based on the model.

An example of lossless reduction of model complexity is the use of a 2-dimensional model of a pipe wall with a central axis of symmetry, and associated 2-dimensional elements that are able to accurately model that axisymmetry, in a case where that pipe component's geometry, boundary conditions, and loadings are fully axisymmetric. In such a case, the 2-dimensional model is a much simpler mathematical representation of the component, but still provides a complete and accurate representation of its behavior.

6.1.3.1 Justification Not Required

The use of simplification of model geometry in a way that results in a lossless reduction of complexity does not need to be justified by the FEA operator, because the model is still a complete and accurate representation of the physical component.

6.1.3.2 Report Considerations for Lossless Reduction

The report should clearly describe the modeling practice that results in a lossless reduction in the numerical complexity of the model. In addition, the report should clearly indicate that, as implemented:

- The simplification does not result in a loss of accuracy of the model with respect to its representation of the physical component
- The simplification does not result in a loss of accuracy with respect to the particular types of analyses that were carried out

6.2 Model Discretization (Meshing)

Model discretization consists of reducing an arbitrary solid geometry that has been modeled to represent the component, to a set of discrete elements of simpler geometry and lower-order mathematical behavior, in order to facilitate

[10] 'Lossless' refers to the fact that in modeling the component, there is no associated 'loss' of information about the real-world geometry or behavior of the component, and there is no loss of accuracy in the results of the undertaken analysis.

analysis with relatively simple sets of equations. Analysis software takes a set of elements that are connected at their nodes, applies boundary conditions, loads, and other constraints to them, and allows them to interact with each other in a way that produces an end result for the entire set of elements that represents the expected behavior of the original solid geometry. The reduction of the arbitrary model geometry to a set of discrete elements is known as ‘model discretization’, and is also referred to as ‘meshing’, as the discretized model can be illustrated by showing the original geometry with the edges drawn as lines on its surface between elements, giving the appearance of having its surface covered with a mesh.

The proper selection and application of techniques in reducing a model to a set of discrete elements is essential to producing a meshed model that will provide behavior representative of the modeled geometry. Areas of rapidly changing geometry or rapidly changing stresses typically need smaller elements to allow local deformations and stresses to be calculated more accurately. Geometries that differ significantly from the geometry of the individual elements also tend to require smaller elements in order to provide a more-accurate representation of the expected real-world behavior.

Although there are known techniques and best practices involved in model discretization, there are slight variations in the methods available depending on the preprocessing software being used, and discretization is significantly dependent on the FEA operator’s experience. This document is relatively non-prescriptive with respect to meshing practices, focusing more on establishing guidelines that the report demonstrate that the discretized model is adequate and can produce suitable results for the geometries, boundary conditions, and loadings being analyzed. (See section 7.2 for guidelines related to verification of mesh adequacy)

6.2.1 Selection of Elements

6.2.1.1 Element Type, Geometry, and Behavior Characteristics

The report should indicate the types of elements used and their characteristics, and in the case that multiple element types or configurations are used, should give an indication of where or how each was used. The following details should be indicated in the report with respect to the selection and configuration of elements:

- Element type (2D, 3D, solid, thick or thin shell, planar/sectional, or other specialty element type, such as bolt connectors, pin connectors, springs, etc.)
- Element geometry (brick, tetrahedral, triangular, etc.)

- Element orientation, if applicable (e.g., ‘top’ or ‘bottom’ orientation for shell elements)
- Mathematical order[11] of elements
- Deformation size for iterative processing (typically referred to as ‘large’ or ‘small’ deformations)
- A brief description of meshing practices employed at local discontinuities or other significant geometric details, such as inside corners and fillets, including minimum element turn angles (as shown in CSA B51-19 Part 1, Annex J, Figure J.1)

6.2.2 Mesh Refinement

Mesh refinement consists of splitting a set of elements into a larger number of smaller elements, resulting in more accurate calculation for the refined area at the cost of a higher number of total elements and a longer associated processing time. Mesh refinement is often performed automatically by preprocessing software, or semi-automatically when an operator selects a region of a discretized model for further refinement. In either case, the meshed model needs to be carefully examined by the FEA operator to ensure that the resulting mesh is suitable.

6.2.2.1 General Guidelines

The following principles that are suggested for consideration in determining where and how to refine a mesh:

1. Mesh should be refined more extensively in areas of higher gradient, where there are rapid changes in stress or strain.
2. Mesh refinement should be considered at locations where there are rapid changes in model geometry, especially in areas of interest for analysis.
3. Mesh refinement should otherwise be tailored to the purpose of the area being meshed. In the case where a portion of a model is used only to transfer loads, a coarser mesh may be suitable, although the FEA operator needs to judge whether loads can be transferred accurately with a coarser mesh. In general, areas where stresses need to be analyzed tend to require more refinement.

6.2.2.2 Automated Adaptive Remeshing

Typical preprocessor software packages offer adaptive remeshing tools that automatically examine the model for areas where low

[11] It should be noted that the use of elements that are less than 2nd-order typically cannot be justified.

accuracy is expected, and remesh those local areas using either the ‘h’ method or the ‘p’ method in order to increase local accuracy.

The ‘h’ method of mesh refinement involves breaking nearby elements down into a larger number of elements, increasing overall accuracy by making each element responsible to model the physical behavior of a smaller portion of the geometry.

The ‘p’ method of mesh refinement involves increasing the mathematical order of existing elements by adding nodes to them. This increases the local accuracy by using a more complex mathematical equation to predict the behavior of each individual element, without changing the size or shape of the elements.

6.2.2.3 Report Considerations

The report should provide:

- The total number of elements of each type in the model
- A brief description of any methods of mesh refinement used in refining the model

6.2.3 Contact Interfaces

Arranging for proper contact between separate element types or between parts that are otherwise not connected requires considerable attention to detail, careful selection of element types and interface parameters, attention to meshing at the surface, and verification that the model is behaving correctly, that displacements of the shared nodes match between element types, and that there is no geometric interference.

6.2.3.1 Report Considerations

When multiple element types are used, or when separate parts are given contact conditions, the report should include a figure showing the surfaces that are in contact, and should include:

- A figure showing the junction between the parts or elements
- A brief description of the method used to transfer behavior between the parts or between the different element types
- An indication of the type of contact (node to surface, surface to surface, etc.)
- An indication of the coefficient of friction used (if applicable), and any other constraints or relevant contact parameters placed at the interface

- A brief description of steps undertaken to ensure that the model behavior is correctly reflected at the interface

6.3 Characteristics of the Material Model[12]

6.3.1 Sources of Information

When available, material properties should be derived from ASME Section II, Part D, or should otherwise be derived from another published and reputable source.

6.3.2 Report Considerations

All material properties (including units) that are used in the analysis are to be indicated in the report. When multiple material models are used for different portions of the model or for different types of analysis, properties of all such materials should be provided, along with an indication of where and how they are used. Typical material properties that would be expected to be provided include the subject material's:

- Modulus of elasticity
- Poisson's ratio
- Yield strength
- Ultimate tensile strength
- Density, if weight is considered in the analysis
- Stress-strain behavior model (elastic, elastic-plastic, elastic-perfectly-plastic, etc.)
- An indication that the effects of non-linear geometry were considered in the analysis
- A plot of the stress-strain curve used, showing the modulus of elasticity in the elastic region, and the stress value at the material's proportional limit

In each case, the report should indicate the bibliographical sources used for the material data.

[12] The term 'material model' as discussed here refers to the set of material properties, physical constants, and stress-strain behavior data which are used to mathematically predict the behavior of the material that a component is made from. Use of the word 'model' is not related to the modeled component geometry discussed previously.

7.0 MODEL VERIFICATION AND VALIDATION

Model verification and validation are required in order to ensure that the component has been modeled correctly and that the model will be able to adequately predict the behavior of the physical component for the types of loadings being considered. In many cases, the actual loads and boundary conditions used for the production analysis can be used to validate the model, and observations made from the production analysis can be used to validate its behavior. In other cases, analysis types such as limit-load analysis involve reliance on a fictional material model and application of fictional loadings to determine the suitability for the component for service. In such cases, the output has no intrinsic physical meaning and cannot be used to suitably validate the model's behavior, and a separate analysis may need to be undertaken in order to validate the model.[13]

7.1 Verification of Inputs

In creating the model, explicit steps are required to be taken to avoid typographical and similar types of input errors, including for the input of model geometry, material properties, and applied forces, loads, and boundary conditions.

This typically involves a formalized explicit secondary check of dimensions and properties after they have been entered and saved, ideally after the FEA operator has taken a short break, or by a separate individual.

7.1.1 Report Considerations

The report should briefly indicate what explicit steps were taken by the FEA operator to identify potential input errors.

7.2 Verification of Mesh Adequacy

Mesh adequacy should be verified by visual inspection as described in 7.2.2 below; and by either nodal stress comparison as described by 7.2.3, or by convergence study as described in 7.2.4.

[13] Limit-load analysis uses a material model which assumes perfectly-plastic yielding occurring at 1.5× the material's code-assigned allowable stress, with pressure loads increased by the same factor. The results of such an analysis are useful for justifying that the component will not be subject to plastic collapse, but the output is not physically meaningful and quantitative measurements cannot be used to draw conclusions about the model's suitability for the analysis. Similarly, a model which uses an elastic-plastic material model which *does* reflect the material's expected behavior may need to be analyzed with lesser applied stresses in order to keep the material in the elastic region of its stress-strain curve, where the resultant stresses and strains can more easily be compared to values derived from hand calculations.

7.2.1 Boundary Conditions and Loading for Mesh Adequacy

Verification of mesh adequacy by any of the below methods that involve the examination of modeled stresses requires that boundary conditions and representative loads be applied to the model in a manner similar enough to what will be used in the production analysis, that the FEA operator can expect the model behavior to be adequately representative of its behavior in the production analysis.

Loads applied to the model for the purpose of model validation need to be *suitably representative* of the loads to be used in the production analysis, but can differ in magnitude. When load adjustments are needed, as may be required to stay within the material's elastic range, the most representative results are likely to be achieved by adjusting all loads in proportion to one another.

In most cases, boundary conditions can be applied for model validation which are the same as those used for the production analysis.

7.2.2 Verification of Mesh Adequacy by Visual Inspection

Visual inspection is the primary means that an FEA Operator relies on to guide the process of mesh refinement. It typically consists of simple on-screen observation of the mesh by the operator, and relies on the operator's experience and intuition to guide further refinement.

Visual inspection is of limited use in formally justifying mesh adequacy due to its qualitative and subjective nature, and due to the fact that intuition can sometimes fail, even for experienced operators.

The inclusion of mesh images in a report provides an indication of the operator's level of experience and familiarity with proper meshing techniques, and is an easy way to show meshing practices and to subjectively show mesh suitability, and as such, is required in all reports. It also gives the operator an opportunity to provide illustrations, allowing the reader to gain a better understanding of the model. It is, however, required to be combined with one of the other specified methods of justifying mesh adequacy, as those methods are quantitative and more objective.

7.2.2.1 Report Considerations

The report should include one or more representative figures of the model, with an overlay showing meshing on the surface. An adequate number of figures are to be included as required to provide a representation of meshing practices used throughout the model, but do not need to show every detail. Some areas that

should be considered for inclusion of a figure showing meshing practices include:

- Areas where high stresses are expected
- Areas where stress concentrations are expected
- Areas where high stress gradients are expected
- Areas where mesh refinement was required
- Areas with physical discontinuities
- Other areas of interest

For the clearest portrayal of meshing practices, it is recommended that such figures be included that show the surface mesh, but without any “heat mapping” coloration to represent the stress distribution for any test load.[14] Although stress distributions can provide additional qualitative evidence of mesh adequacy, it may be recommended in such cases to show mesh quality and stress distribution in separate side-by-side figures.

7.2.3 Verification of Mesh Adequacy by Nodal Stress Comparison

Mesh adequacy can be verified by comparing the nodal and average stresses of each element in the model, and ensuring that discrepancies do not exceed an acceptable limit. This has the effect of ensuring that each such element is small enough that calculated stresses did not change rapidly across the element, and that the mesh is therefore not under-refined in areas of high stress gradient.

Although there is no direct deterministic relationship, stress gradient across a single element is related to the potential for inaccuracy in the values calculated for that element in the eventual analysis. In some cases, inaccuracies at a local area of high stress gradient can propagate to inaccuracies at other areas of the model, as associated loads can be miscalculated, or the inaccurate stresses can be transferred to other elements incorrectly.

A deviation of 5% or more between nodal and average element stresses is often evidence of bad modeling or discretization practices, or the use of a material model that is not suitable for the analysis being undertaken, and is typically not acceptable. Even in locations of high stress gradient, this level of deviation between nodal and averaged stresses is evidence of an overly coarse mesh that requires further refinement.

[14] Inclusion of colored stress mappings along with the mesh can sometimes make the mesh lines difficult for the reader to see, particularly if the report is printed.

7.2.3.1 Report Considerations

When nodal stress comparison is used to verify mesh adequacy, the report should include:

- A statement that mesh adequacy was justified by nodal stress comparison
- An indication of the maximum permissible stress deviation[15] designated by the FEA operator
- An indication of the maximum stress deviation found in the model (expressed as a percentage)
- A figure showing the local stress distribution at the area of maximum stress deviation; additional figures should be considered for other areas of interest
- A short discussion of how the identified stress deviations will affect the results of the types of analyses to be undertaken, and how those deviations will be considered in interpreting results
- In cases where the prescribed limit was not achievable at certain locations due to singularities and similar artifacts, a short discussion of those locations (including figures as necessary) and discussion of the effects of those artifacts on the types of analyses to be undertaken.

7.2.4 Verification of Mesh Adequacy by Convergence Study

Mesh adequacy can be verified by applying carefully-targeted local or in some cases global refinements to the mesh, and studying the effects of mesh refinement on convergence of the stresses predicted by the model towards an apparent limit. When additional mesh refinement is shown not to produce meaningful additional accuracy, the mesh is judged to have been adequately refined.

Mesh refinement and evaluation of convergence may be performed globally for the entire discretized model or can be performed locally at carefully-selected areas of interest. In cases where convergence study is based on local examinations, mesh adequacy for the remaining portions of the model must be shown by the FEA operator to be trivial, and of low importance for the types of analyses to be undertaken. Alternatively, mesh adequacy for such portions of the model can be proven by another method.

[15] For certainty, ‘stress deviation’ here refers specifically to the deviation between nodal and averaged (elemental) stresses for any single model element.

The general procedure involves systematically refining the mesh, applying some test load case, analyzing the model, and keeping track of peak stresses at some location of interest as they relate to either the local or global mesh density. Mesh density should be expressed quantitatively and can be expressed in terms of the total number of elements within a specific volume, or per unit volume of modeled material for the location near where stresses are being tracked. The data are plotted on a graph showing how stress predictions change with increasing mesh density, and once the FEA operator is satisfied that additional mesh refinements are not necessary to obtain the desired level of accuracy, they judge that the mesh adequacy has been verified. This is typically done by setting quantitative criteria, such as a maximum permissible change in stress level between successive iterations with some proportional increase in mesh density.

7.2.4.1 Report Considerations

When convergence study is used to verify mesh adequacy, the report should include:

- A statement that mesh adequacy was justified by convergence study
- A brief description of the procedure used, including the type and extent of mesh refinement used between successive iterations
- A figure showing the local stress distribution at the main area where stress was measured; additional figures may be considered for other areas of interest
- A graph showing predicted stress at the area of interest, as it changes with respect to the quantified mesh density
- A short discussion of how the identified stress deviations will affect the results for the types of analyses to be undertaken, and how those deviations will be considered in interpreting analysis results
- In cases where the limit prescribed by the FEA operator was not achievable at certain locations due to singularities and similar artifacts, a short discussion of those locations (including figures) and discussion of the effects of those artifacts on the types of analyses that are undertaken

7.3 Verification of Model Behavior

A representative set of loadings and boundary conditions should be applied to the model, and model behavior should be verified to be suitably representative of the component being evaluated using the techniques described in 7.3.1, 7.3.2,

and 7.3.3; and either the technique described in 7.3.4 or the technique described in 7.3.5.

In many cases, the loadings and boundary conditions used for an actual production analysis used for justification of the design can be used to verify model behavior using these techniques. In some cases, however, such as for limit load analysis, the model output does not have physical meaning, and cannot be used to verify the model's behavior. In such cases, a separate set of load conditions may need to be applied which will keep the material within its elastic range so that the analysis output will retain physical meaning and the model can be evaluated before the limit-load analysis is carried out.

7.3.1 Verification of Response to Boundary Conditions

This verification involves qualitative and/or quantitative evaluation of the model's response to applied boundary conditions. Various techniques are applied at the discretion of the operator, and can include:

- Visually examining the model for unexpected stress concentrations or deformations at locations where geometric constraints are applied to prevent rigid body motion
- Visually examining the model for unexpected displacements or unnatural changes of shape at locations where symmetry conditions have been applied
- Comparing reaction forces predicted by the model to those calculated by hand, using a summation of applied forces and moments

7.3.1.1 Report Considerations

The report should describe what the FEA Operator did to verify the response of the model to applied symmetry and boundary conditions, and include supporting figures as necessary.

7.3.2 Visual Inspection of Deformed Shape

This verification involves a qualitative evaluation of the deformed shape with what would be expected under the applied loading conditions, based on the experience of the FEA operator. It provides evidence that forces were applied in the correct direction and of approximately the right order of magnitude. Typically, deformations are imperceptibly small and need to be amplified in post-processing to facilitate visual inspection.

7.3.2.1 Report Considerations

The report should include:

- A brief description of the pressure and other loadings applied to the model to produce the deformed shape
- One or more figures to show the deformed shape of the model with reference to its undeformed shape, either side-by-side, or by superimposing the images
- An indication of what magnification factor was applied to facilitate the visualization
- A statement that the deformed shape matches the expected physical deformation of the component under similar loadings, with any brief discussion that may be required to reconcile unexpected or unintuitive results
- Comparison may be made to the deformations expected for well-understood geometries, where applicable; examples could include a reduction in the diameter at the knuckle of a semi-elliptical head, resulting in the knuckle being “pulled in”, or observed swelling of flat plates subjected to pressure, with or without rotation at the outer edge, depending on edge attachment

7.3.3 Visual Inspection of Stress Distribution

This verification involves a qualitative evaluation of the distribution of stresses in the model, based on stress distributions that are expected for similar geometries under similar loading conditions.

Examples may include concentration of stresses around openings in cylinders or near well-understood structural discontinuities, reflection of symmetry conditions when geometry and applied loads are symmetric, or agreement with known results based on previous experience with similar boundary and loading conditions.

7.3.3.1 Report Considerations

The report should include:

- A brief description of the pressure and other loadings applied to the model to produce the stress distribution
- One or more figures showing the surface stress distributions of important portions of the model, using colored “heat mapping” to show relative magnitudes of stresses
- A statement that the distribution of stress matches the expected distribution, with an indication of how the expected stress distribution was determined
- Discussion of any unexpected or unintuitive results

7.3.4 Verification of Model Stresses by Hand Calculation

This verification involves a quantitative evaluation of the stresses or reaction forces in the model, based on a comparison of model output with that of a hand calculation. In some cases, it may be required to extend a portion of the model to provide a cylindrical or other section where stresses can easily be calculated by hand.

7.3.4.1 Report Considerations

The report should include:

- A brief description of the type of calculation used to verify model output, including mention of the source of the equation or calculation procedure
- A brief description of the location in the model used where the calculable quantity was measured
- A statement directly comparing the results of the hand calculation with the model's prediction, indicating the percent error and whether it is within an acceptable range

7.3.5 Comparison of Model Strains to Physical Behavior[16,17]

This verification involves a quantitative evaluation of the stresses in the model, based on the comparison of model output with that of the measured behavior of a physical system. System behavior is most typically measured by the application of strain gauges to a physical component, and comparison of measured strains with strains predicted by the model.

Although there is typically not a physical component that matches the model that can easily be subjected to testing to obtain physical measurements, this method may be useful when there is a substantially similar component, and the model can be easily adjusted to match the available component. In such cases, modifications to the model should be simple enough that the FEA operator can justify extending the results to the model being validated.

[16] Verifying model behavior by quantitative comparison to the physical behavior of a component is only valid when a material model is used which is closely representative of the behavior of the physical material, such as an accurate elastic-plastic material model, or when all relevant portions of the component stay in the elastic stress range.

[17] This method of model validation is understood to be impractical in many cases and is included here mainly for historical reasons.

7.3.5.1 Report Considerations

The report should include:

- A brief description of the physical test undertaken to obtain physical measurements, including details of the component that was tested and any details in which it differs from the model; details of pressure and other loadings applied to the component; and placement of the strain gauges or other measuring devices
- A brief description of any changes made to the model to match the physical tested component, and brief discussion as required to justify that the result is applicable to the unaltered model that is to be used in the production analysis
- A statement directly comparing the measured strains with those predicted by the model, indicating the percent error and whether it is within an acceptable range

8.0 RESULTS OF THE DESIGN ANALYSIS

This section applies with respect to the design analysis, which is undertaken in order to justify the design for the proposed design conditions with respect to the governing code of construction, which typically refers to ASME Section VIII-2, Part 5: Design by Analysis.

Section VIII-2 Part 5 requires each of several failure modes to be addressed, and for each failure mode permits several types of analysis. The proper application of the various analyses is outside of the scope of this document. Considerations for the report with respect to those analyses are instead established in general terms, with the understanding that the FEA operator is following the analysis requirements as established by Section VIII-2 Part 5, and supplementing them with applicable best practices.

The below considerations for documentation of the analyses aim at requiring the FEA operator to provide an indication of the analysis types chosen, relevant details of those analyses, and clear statements indicating acceptability of the results.

8.1 Required Report Contents

For each analysis required to be undertaken, the report should provide a section documenting it, and should include:

- The failure mode addressed (e.g., plastic collapse, local failure, buckling, etc.)
- The type of analysis used, with reference to the supporting paragraph in Section VIII-2 (e.g., “Limit-Load Analysis per ASME VIII-2 para. 5.2.3”)
- The magnitudes of loads (pressures and other loads) used for all required load cases, and an indication of where each is applied to the model
- A figure clearly showing the application of loads and boundary conditions for at least one representative load case
- A discussion of how the analysis was undertaken, with details adequate that another qualified FEA Operator can be reasonably confident that the analysis was undertaken correctly
- A clear statement that the code acceptance criteria was met for the analysis, or otherwise clear portrayal of the results

9.0 REVISION LOG

Edition #	Revision #	Date	Description
First Edition issued 2009-11-23			
2	0	2023-10-26	Complete revision