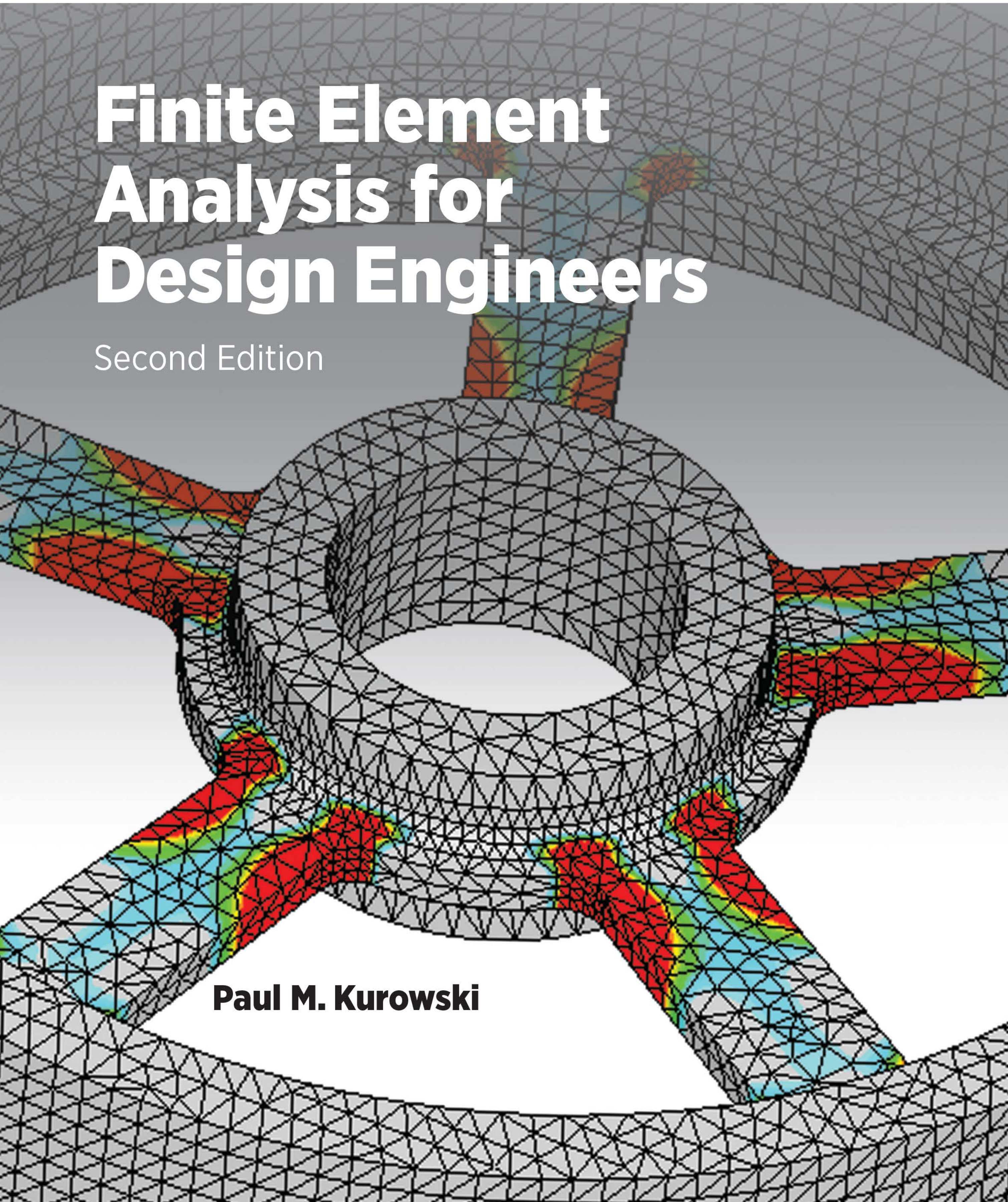
SAE INTERNATIONAL

Finite Element Analysis for Design Engineers

Second Edition



Paul M. Kurowski

Finite Element Analysis for Design Engineers

Second Edition

Other SAE books of interest:

Dynamic Analysis and Control System Design of Automatic Transmissions
Shushan Bai, Joel M. Maguire, and Huei Peng
(Product Code: R-413)

Principles of Vibration Analysis with Applications in Automotive Engineering
Ronald L. Huston and C. Q. Liu
(Product Code: R-395)

Fundamentals of Automobile Body Structure Design
Donald E. Malen
(Product Code: R-394)

Design and the Reliability Factor
John Day
(Product Code: PT-174)

CAE Design and Failure Analysis of Automotive Composites
Srikanth Pilla and Charles Lu
(Product Code: PT-166)

Finite Element Analysis for Design Engineers

Second Edition

Paul M. Kurowski

SAE INTERNATIONAL

Warrendale, Pennsylvania, USA

 eISBN: 978-0-7680-8369-9

INTERNATIONAL®

400 Commonwealth Drive
Warrendale, PA 15096 USA
E-mail: CustomerService@sae.org
Phone: +1 877.606.7323 (inside USA and Canada)
+1 724.776.4970 (outside USA)
Fax: +1 724.776.0790

SAE Order Number R-449
http://dx.doi.org/10.4271/r-449

Library of Congress Control Number: 2016945394

ISBN-Print 978-0-7680-8231-9 **ISBN-PDF** 978-0-7680-8369-9
ISBN-epub 978-0-7680-8371-2 **ISBN-prc** 978-0-7680-8370-5

To purchase bulk quantities, please contact SAE Customer Service:

Email: CustomerService@sae.org
Phone: 1+877-606-7323 *(inside USA and Canada)*
1+724-776-4970 *(outside USA)*
Fax: 1+724-776-0790

Visit the SAE International Bookstore at books.sae.org

Acknowledgements

This book is dedicated to my wife Elzbieta Kurowska for her encouragement and support.

Paul M. Kurowski

Contents

Preface

During 60+ years of its development the Finite Element Analysis evolved from an exotic analysis method accessible only to specialized analysts into a mainstream engineering tool. Phenomenal progress in computer hardware and operating systems combined with same progress in Computer Aided Design made the Finite Element Analysis available to design engineers to use as a design tool during product design process.

Many books have been written about the Finite Element Analysis. At one end of the spectrum we find books going very deep into theory and at the other end of the spectrum, software manuals explaining how to use certain FEA program. There is little FEA literature taking "middle ground" approach and specifically addressing the needs of design engineers who use the FEA as an everyday design tool. This book attempts to fill this void by focusing on understanding of FEA fundamentals which are explained by simple, intuitive examples understandable by any mechanical engineer. "Finite Element Analysis for Design Engineers" takes practical approach, characteristic to the attitudes of design engineers, and offers the readers an opportunity to try out all discussed topics by solving downloadable exercises using their own FEA program.

Finite Element Analysis for Design Engineers is a very broad field of knowledge. It is difficult to write a book in a "linear" fashion; repetition in discussing concepts, techniques and methods can't be avoided. For this reason, some topics are discussed more than once taking advantage of a growing body of knowledge as reader progresses through the book.

Chapter 1
Introduction

1.1 What Is Finite Element Analysis?

Finite element analysis, commonly referred to as FEA, is a numerical method used for the analysis of structural and thermal problems encountered by mechanical engineers during design process. There are other applications of FEA, but in this book, we will discuss structural and thermal problems only.

It is appropriate to start our discussion with the definition of what design analysis in general is and how does it relate to FEA. Design analysis is a process of investigating certain properties of parts or assemblies. Design analysis can be conducted on real objects or on models that represent certain aspects of the real object. If models are used instead of real objects, the analysis can be conducted earlier in the design process before the final product or even prototypes are built. Those models can be physical models (scaled-down models, mockups, photo-elastic models, etc.) or mathematical models where certain behavior of part or assembly is described by a mathematical apparatus. Design analysis conducted with the use of mathematical models can be further broken down based on what methods are used to obtain solution. Simple mathematical models can be solved analytically. More complex models require the use of numerical methods. FEA is one of those numerical methods used to solve complex mathematical models. The FEA has numerous uses in science and engineering, but as we have already mentioned, the focus of this book is on structural and thermal analysis. We will alternate between two terms and two acronyms that became synonymous in the engineering practice: the FEA and the finite-element method (FEM).

The FEA is a powerful but demanding tool for engineering analysis. The expertise expected from FEA users depends on the extent and complexity of conducted analysis and requires familiarity with mechanics of materials, kinematics and dynamics, vibration, heat transfer, engineering design, and other topics found in every undergraduate mechanical engineering curriculum. For this reason, many introductory FEA books offer

the readers a quick review of those engineering fundamentals. Rather than duplicating the efforts of other authors, chapter 15 refers the readers to some of those books.

1.2 What Is the Place of Finite Element Analysis Among Other Tools of Computer-Aided Engineering?

FEA is one of many tools of computer-aided engineering (CAE) used in mechanical design process. Other CAE tools include fluid flow analysis commonly called computation fluid dynamics (CFD) and mechanism analysis. These three major CAE tools: FEA, CFD, and motion analysis are integrated with computer-aided design (CAD), which is a hub for all CAE applications. Geometry and material properties can be exchanged between CAD and add-ins as well as directly between different add-ins (Figure 1.1).

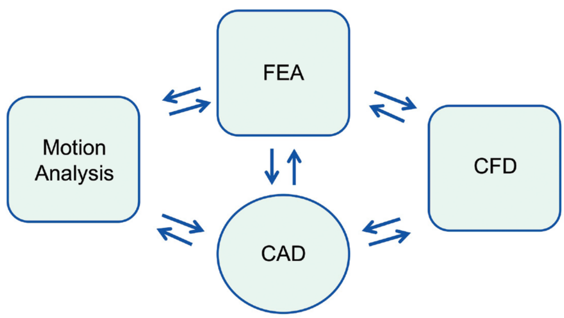


Figure 1.1 CAE applications such as FEA, CFD, and motion analysis are add-ins to CAD. They can exchange data with CAD and in between themselves.

FEA, CFD, and mechanism analysis have been developed independently and are based on different numerical techniques; the integration shown in Figure 1.1 is a relatively new development. However, even if CAE tools are stand-alone programs and not add-ins to CAD, they can still be interfaced with CAD.

The main difference between FEA and motion analysis is the field of application. FEA is used for analysis of structures subjected to loads and motion analysis is used for analysis of motion of mechanisms. Telling apart structure from mechanisms may bring about some confusion. These differences will be clarified in the next chapter.

1.3 Fields of Application of FEA and Mechanism Analysis; Differences Between Structures and Mechanisms

A mechanism is not firmly supported and can move without having to deform; components of a mechanism can move as rigid bodies. On the contrary, any motion of a structure must involve deformation because a structure is, by definition, firmly supported. Motion of a structure may take the form of a one-time deformation when a static load is applied, or the structure may be oscillating about the position of equilibrium when a time-varying load is present. In short, a mechanism may move without having to deform its components, while any motion of a structure must be accompanied by deformation.

If an object cannot move without experiencing deformation, then it can be classified as a structure. Examples of the mechanism and the structure are shown in Figure 1.2.

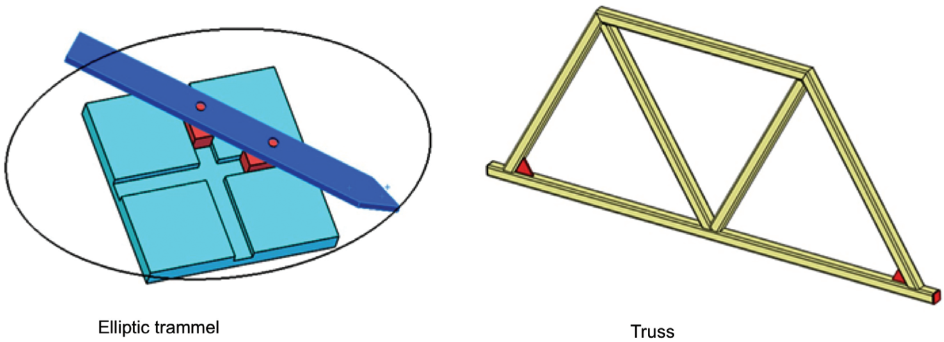


Figure 1.2 An elliptic trammel is a mechanism; its components may move without experiencing any deformation. Truss, which is a weldment made with tubes, is firmly supported. The only movement it may experience is one time deformation under static load or oscillation about the position of equilibrium under dynamic load.

An elliptic trammel is a mechanism; it is designed to trace an ellipse when it moves. This motion can be studied without considering deformation of its components. A truss that is made with welded tubes is a structure. It is designed to stand still and can move only as it deforms from its position of equilibrium. Any motion of a structure is always accompanied by deformation.

Depending on the objective of analysis, an object or its components may be treated as either a mechanism or a structure. A helicopter rotor is a mechanism; it spins relative to the hull. A rotor may be treated as a rigid body or an assembly of rigid bodies or an assembly of elastic bodies. An individual blade may be treated as a rigid body or as a structure if its vibration characteristics need to be analyzed (Figure 1.3).

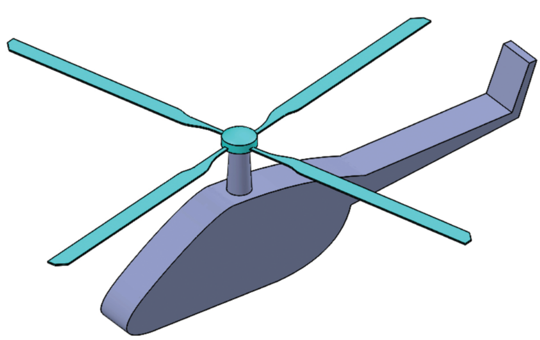


Figure 1.3 A helicopter rotor can be considered as a mechanism composed of rigid bodies, a mechanism composed of elastic bodies, or as a structure.

A rigid body cannot deform under load; a rigid body it is a mathematical abstract being convenient in mechanism analysis but rarely used in structural analysis.

1.4 Fields of Application of FEA and CFD

In this book, we will discuss the FEA as a tool of structural analysis and as a tool of heat transfer analysis in solid bodies. Chapter 12 dedicated to heat transfer analysis will clearly differentiate between FEA and computational fluid dynamics (CFD). We will point out that heat transfer in solid bodies is analyzed with FEA, while heat transfer in fluids requires CFD.

1.5 What Is "FEA for Design Engineers"?

What exactly distinguishes "FEA for design engineers" from FEA performed by a specialized analyst? To set tone for the rest of this book, we will highlight the most essential characteristics of FEA for design engineers as opposed to FEA performed by analysts:

- *FEA Is Just Another Design Tool:* For Design Engineers, the FEA is one of many design tools and is used along CAD, spreadsheets, catalogs, data bases, hand calculations, text books, etc.
- *FEA Is Based on CAD Models:* Design is nowadays always created using CAD, and, therefore, the CAD model is the starting point for FEA.
- *FEA is Concurrent With the Design Process:* Because FEA is a design tool, it should be used concurrently with the design process. It should keep up or, better, drive the design process. Analysis iterations must be performed fast and because results are used to make design decisions, the results must be reliable even though not enough input data may be available for analysis conducted early in the design process.
- *Limitations of "FEA for Design Engineers":* As we can see, the FEA used in the design environment should meet quite high requirements. It must be executed fast and accurately even though it is in the hands of design engineers and not FEA specialists. An obvious question is would it be better to have a dedicated specialist perform FEA and let design engineers do what they do best: designing new products? The answer depends on the size of organization, type of products, company organization, and culture, and many other tangible and nontangible factors. A general consensus is that design engineers should handle relatively simple types of analysis in support of design process. More complex types of analyses, too complex and too time consuming to be executed concurrently with design process, are usually either better handled by a dedicated analyst or contracted out to specialized consultants.
- *Objectives of "FEA for Design Engineers":* The ultimate objective of using the FEA as a design tool is to change the design process from iterative cycles of "design, prototype, and test" into a streamlined process where prototypes are used only for final design verification. With the use of FEA, design iterations are moved from

physical space of prototyping and testing into virtual space of computer-based simulations (Figure 1.4). The FEA is not, of course, the only tool for computerized simulation used in the design process. As we have mentioned before, there are others like CFD and motion analysis, jointly called the tools of CAE.

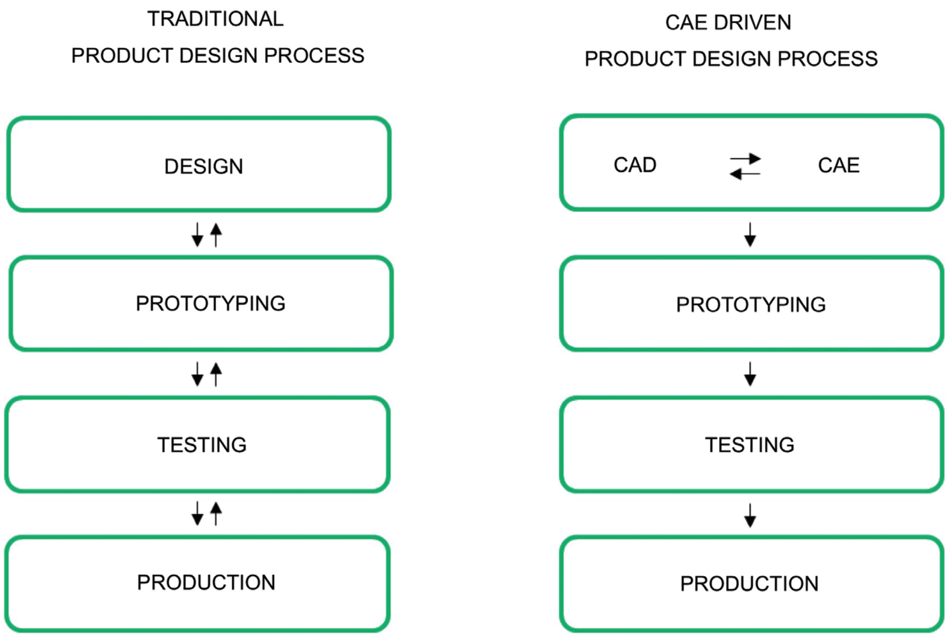


Figure 1.4 Traditional product development needs prototypes to support design process. CAE-driven product development process uses numerical models, rather than physical prototypes, to support the product design process.

1.6 Importance of Hands-On Exercises

Based on many years of authors' teaching experience, reading this book (or any FEA-related book for that matter) is not enough for the knowledge to "sink in." To assure an effective transfer of knowledge, it is necessary to complete hands-on exercises. Therefore, the topics discussed in this book are accompanied by simple yet informative examples presented at the end of related chapters. The exercises are not specific to any particular software and can be solved using any commercial FEA program. Completing exercises is essential in the learning process facilitated in this book. For the readers' convenience, geometry for all exercises can be downloaded in Parasolid and SOLIDWORKS 2016 formats from http://designgenerator.com/index.php/downloads.

Chapter 2

From CAD Model to Results of Finite Element Analysis

2.1 Formulation of the Mathematical Model

The starting point of an FEA project is a CAD model, which is the basis for creating the mathematical model. To underscore the importance of mathematical model in the analysis process, it is important to describe what a mathematical model is, where it fits in the design analysis process and how different is it from the CAD model and from the finite-element (FE) model.

Suppose we need to find displacements and stresses of an idler pulley under a belt load. A CAD model defines a volume, which is our solution domain (Figure 2.1). The volume has material properties assigned to it and certain conditions are defined on all external faces that define domain boundaries. The conditions defined on external faces of a model are called boundary conditions.

The boundary conditions can be defined in terms of displacements and loads. Displacement boundary conditions are called essential boundary conditions, and load boundary conditions are called natural boundary conditions.

In our example, displacement boundary conditions are defined on the inner cylindrical face as zero displacement to represent support provided by a bearing; a bearing itself is not modeled. Load boundary conditions are applied to a section of the outer cylindrical face as a pressure and represent the belt load. The above displacement and load boundary conditions are defined explicitly. All the remaining surfaces have implicit load boundary conditions calling for zero tractions, which means zero stress in the direction

normal to the surface. This reflects the fact that normal stresses must not exist on an unloaded surface.

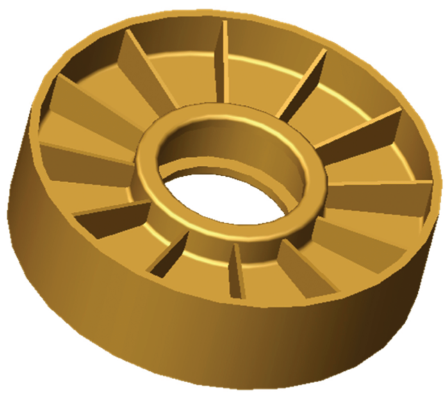


Figure 2.1 A model of an idler pulley presented as a volume (solid geometry). Representing the model as a volume, and not as a surfaces as in the next illustration, affords inclusion of many important modeling details like small rounds.

It is important to notice that the geometry illustrated in Figure 2.1 is not the only possible representation of the analyzed pulley. Figure 2.2 presents another possibility where pulley geometry is represented by surfaces. Notice that this approach necessitates the removal of small rounds.

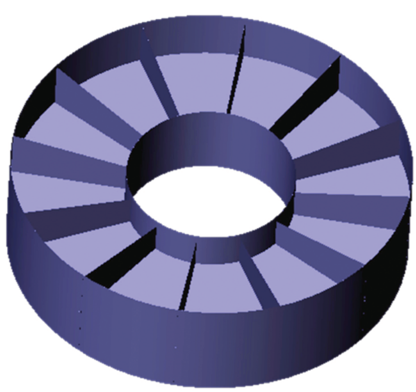


Figure 2.2 An alternative geometry representation of the idler pulley; the geometry is represented with surfaces. Rounds cannot be modeled when surfaces are used to represent the pulley geometry.

In many cases, more than one geometry representation is possible for a given problem. Pulley geometry lends itself to representing it with either volumes or surfaces. Modeling a beam offers more choices as the beam can be represented by volumes, surfaces, or curves, as shown in Figures 2.3–2.5.

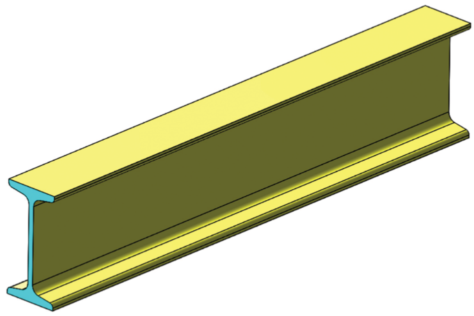


Figure 2.3 The beam represented by a volume; this affords inclusion of all details of the cross-sectional geometry.

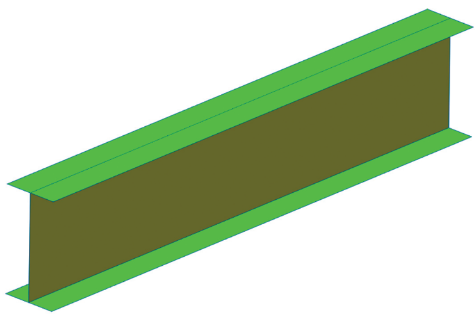


Figure 2.4 The beam represented by surfaces cannot include cross-sectional details like rounds, fillets, etc. Surface geometry misses one dimension: thickness, which cannot be derived from the geometry and has to be defined as a number.

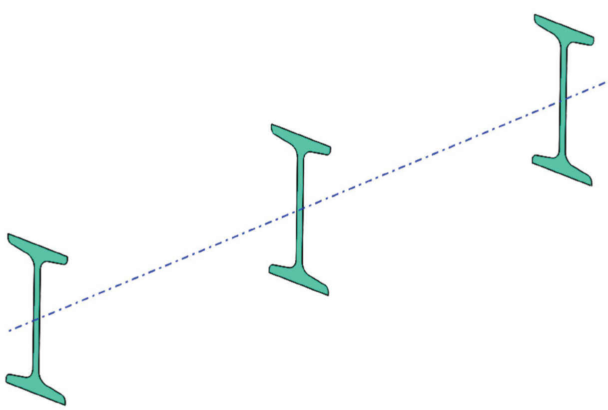


Figure 2.5 The beam represented by a curve that is shown here as a dotted line. In this highly idealized model, beam properties are assigned to lines. The model contains no details and can represent only global beam behavior. Representing the geometry by curves misses two dimensions. The beam cross section and second moments of inertia cannot be derived from geometry and must be defined as numbers.

The I beam shown in Figure 2.3 is represented by a 3D geometry. Figures 2.4 and 2.5 illustrate progressive idealization of the 3D geometry, surface geometry misses one dimension, and curve geometry misses two dimensions.

Notice that the geometry alone does not fully define the mathematical model. Boundary conditions (loads, restraints) and material properties are also building blocks of mathematical model and may be defined in many different ways to differentiate between the static and dynamic analyses and linear and nonlinear material analyses, etc.

If the same problem can be described by more than one mathematical model, can we say which model is the best one? This depends on the objective of our analysis. The best mathematical model is the one that adequately represents those aspects of real design that are of interest to us, we call them the data of interest, and does that at the lowest cost. Consequently, a mathematical model must be constructed keeping in mind the objective of analysis. Creating a mathematical model that properly represents the data of interest is the most important step in the modeling process. Yet, its importance is often overlooked in practice or worse, the distinctions among the CAD model, mathematical model, and finite element model are not recognized.

2.2 Selecting Numerical Method to Solve the Mathematical Model

Having formulated a mathematical model of a structure to be solved for displacements and stresses (or for temperatures and heat flux), we have formulated the boundary value problem. This boundary value problem can now be solved with different numerical methods; the finite-element method (FEM) is one of them.

2.2.1 Selected Numerical Methods in Computer Aided Engineering

- *Finite Difference and Finite Volume Methods:* Finite difference and finite volume methods are based on the differential formulation of a boundary value problem. This results in a densely populated, often ill-conditioned matrix leading to numerical difficulties. The solution domain is divided into cells. These methods are often used in flow analysis problems.
- *Boundary Element Method:* The boundary element method is based on the integral equation formulation of a boundary value problem. This also results in a densely populated, nonsymmetric matrix. Boundary element methods are efficient for “compact” 3D shapes but are difficult to implement for more “spread out” geometries. Only the domain boundary, but not inside, is divided into segments.
- *FEM:* The FEM is based on the variational formulation of a boundary value problem. In the FEM, the unknown functions are approximated by functions generated from polynomials. These functions are effective for the reasons of numerical efficiency. The entire solution domain (model geometry) must be discretized (meshed) into simply shaped subdomains called elements.

2.2.2 Reasons for the Dominance of Finite Element Method

When numerical analyses were first introduced in engineering practice in the 1960s, many analysis methods were in use, but over time, the FEM became the dominant numerical method because of its generality and numerical efficiency. While other methods retain advantages in certain niche applications, they are difficult or impossible to apply to other types of analyses. At the same time, the FEM can be applied to just about any type of analysis. This generality and numerical efficiency is a major consideration for programmers when they decide which method to use in commercial analysis program. The development of modern analysis software consisting of several million lines of code is a huge investment, which can be recouped only by creating a versatile and efficient product. The FEM delivers that versatility and efficiency and for this reason has come to dominate the market of commercial analysis software.

Different numerical methods used for solving engineering design analysis problems are schematically presented in Figure 2.6.

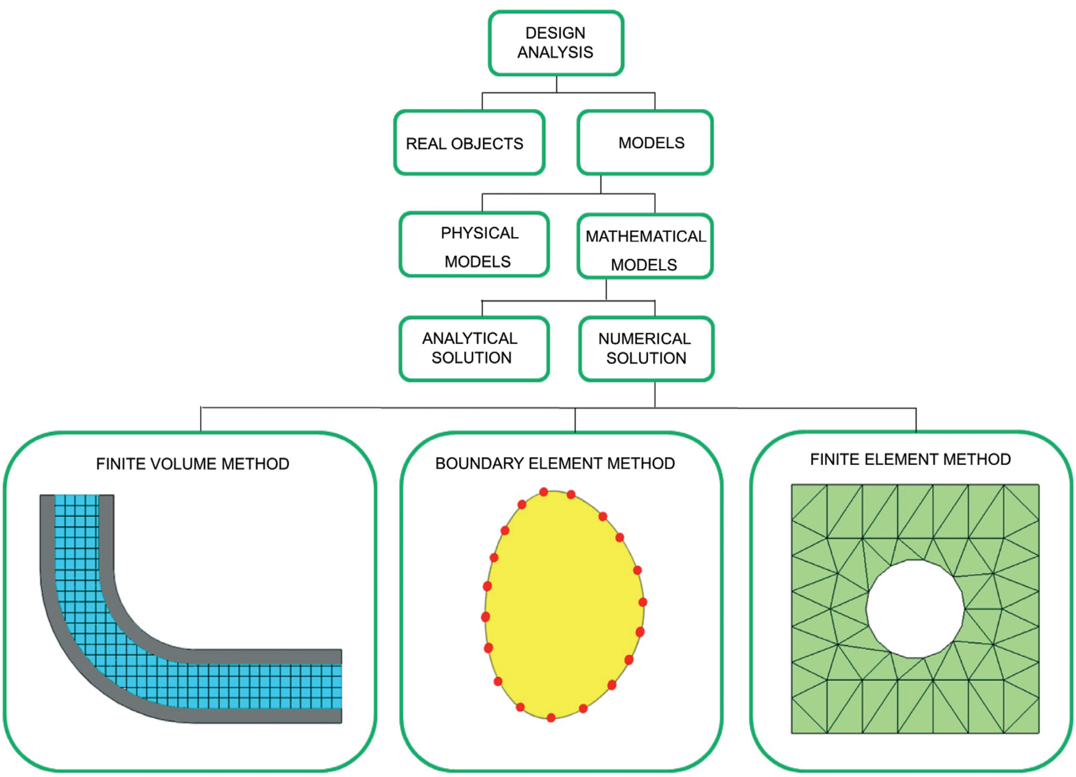


Figure 2.6 Different numerical methods used for solving engineering analysis problems.

2.3 The Finite Element Model

2.3.1 Meshing

Having decided to use the FEM to solve our mathematical model, we now have to follow the path of FEM. First, we need to split solution domain into simply shaped subdomains called finite elements. This is discretization process commonly called meshing and elements are called "finite" because of their finite, rather than infinitesimally small, size.

Why exactly is meshing required? Risking some oversimplification, we may picture the FEM as a method of representing variables like displacements by polynomial functions that produce the displacement field compatible with applied boundary conditions and at the same time minimize the total potential energy of the model. Obviously, to describe the entire model "in one piece," without splitting it into elements, those polynomial functions would have to be very complex. This is where the need for meshing becomes obvious. Meshing splits up the solution domain as in Figure 2.7 into simply shaped elements or subdomains as shown in Figure 2.8, allowing reasonably simple polynomials to be used to approximate displacement or temperature field in each element.

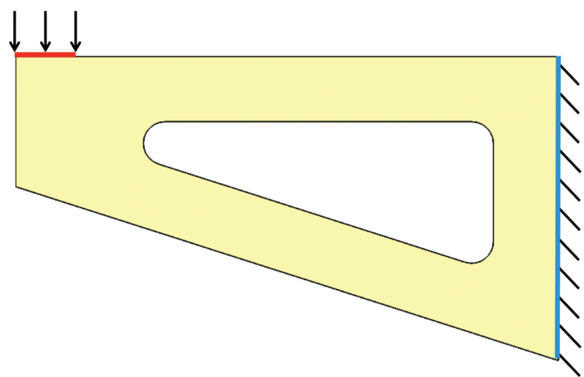


Figure 2.7 A mathematical model of a bracket is defined by a 2D domain with applied boundary conditions (loads and restraints constitute boundary conditions), by material properties and problem formulation (static analysis, buckling analysis, etc.) In this illustration, a uniformly distributed load (natural boundary conditions) is applied over the length of the red line. Fixed restraints (essential boundary conditions) are applied over the length of the blue lines.

Meshed geometry is a result of discretization, but discretization is not limited only to geometry. Everything in the originally continuous mathematical model is discretized; this includes mass, loads, and restraints. As shown in Figure 2.8, originally continuous boundary conditions (loads and restraints) are now represented by loads and restraints applied to element nodes.

As we will discuss it later, the behavior of each element is fully characterized by displacements of its nodes. Therefore, the continuous mathematical model with an

infinite number of degrees of freedom after discretization is approximated by a discrete finite element model with a finite number of degrees of freedom.

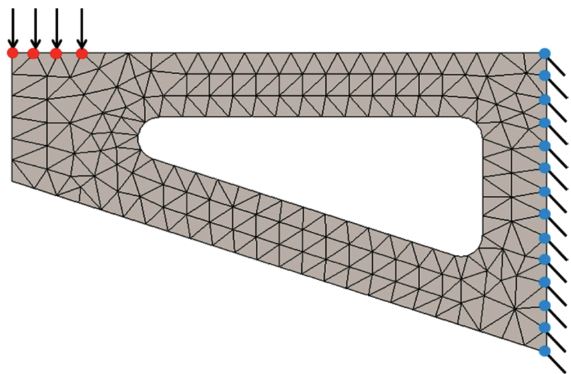


Figure 2.8 FE model of the support bracket; the 2D solution domain (model geometry) and boundary conditions (restraints and loads) have been discretized. The red dots indicate nodes where an equivalent nodal load is applied and the blue dots indicate nodes where restraints are applied. This illustration shows model meshed with second-order elements; second order elements have corner nodes and mid-side nodes as discussed in chapter 3. Loads and restraints are applied to corner nodes and mid-side nodes but for clarity of this illustration, the color dots are shown only in corner nodes, mid-side nodes can't be seen in this illustration.

2.3.2 Formulation of Finite-Element Equations

Out of the infinite number of sets of nodal displacements that are allowed by restraints, only one set of nodal displacements minimizes the total potential energy of the model. This state of the minimum total potential energy corresponds to the state of equilibrium. Therefore, by finding the set of nodal displacements that minimizes the total potential energy of the model, we can find a state of equilibrium of this model under the applied loads and restraints. The application of the principle of the minimum total potential energy leads to the formulation of the fundamental FEA equations:

$$[K] * [d] = [F] \tag{2.1}$$

where $[K]$ is the known stiffness matrix, $[F]$ is the known vector of nodal loads, and $[d]$ is the unknown vector of nodal displacements.

The stiffness matrix $[K]$ is a function of model geometry, material properties, and displacement boundary conditions commonly called restraints.

FEM equations (2.1) take the form of linear algebraic equations and can be solved with various numerical solvers. Nodal displacements (or temperatures) are calculated in this step. If desired, and most often this is indeed desired, we may proceed with calculations of strains and stresses (or temperature gradients and heat flux).

2.3.3 Errors in FEA Results

When using finite element analysis (FEA) results to make design decision, we need to remember that results have been produced in a process that consists of several steps (Figure 2.9) and each step adds errors to the FEA results.

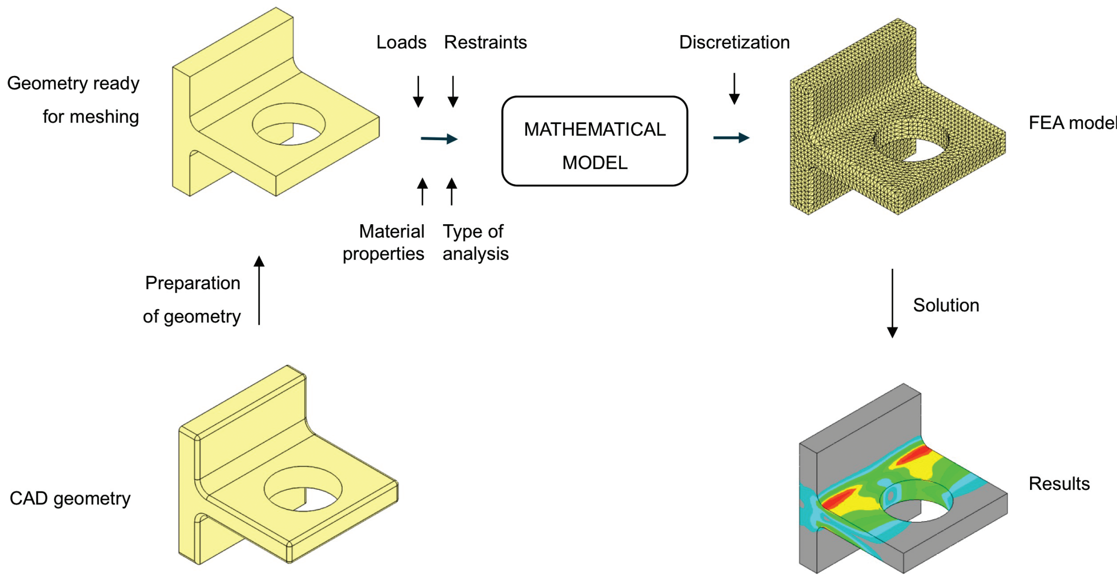


Figure 2.9 Steps in an FEA project: preparation of geometry, definition of loads, restraints, material properties, type of analysis, discretization, solution, and analysis of results.

- *Step 1: Creating Mathematical Model:* We start with a CAD model that represents the real life geometry. Let us assume that CAD geometry is "clean"; there are no CAD modeling errors. We need to simplify that geometry before meshing. Manufacturing specific CAD geometry almost always requires simplifications that may take form of idealizations (e.g., using surfaces or curves in place of volumes) and defeaturing (suppressing or deleting geometry details that we deem unnecessary for analysis). For example, Figure 2.8 shows that rounds have been removed from the outside edges. A simplified geometry, called FEA geometry, is the first building block of a mathematical model. Load, restraints (displacement boundary conditions and load boundary conditions), and material properties are applied to that geometry and the type of analysis is defined. In this way, a mathematical model is created. Step 1 introduces modeling errors.
- *Step 2: Discretization:* The mathematical model is discretized into finite elements. Remember the discretization is done both on geometry and boundary conditions. There is nothing continuous left in the FEA model. Finite element equations (2.1) are created in this step. Step 2 introduces discretization error.

- *Step 3: Solution:* FEA equations are solved in this step; this introduces solution error, also called numerical error or round-off error.
- *Step 4: Analysis of Results:* Results produced in step 3 are now used to make a design decision. If results are misunderstood, then the error of interpretation of results is introduced in this. Modeling errors, discretization errors, and solution errors are unavoidable; the error of interpretation of results is, of course, fully preventable.

We will comprehensively discuss all those errors in further chapters. Here, we just mention that while all errors affect results, only discretization errors are specific to the FEA. Therefore, only discretization errors can be controlled using FEA techniques.

2.4 Verification and Validation of FEA Results

We will now define verification and validation as they apply to design analysis with FEA:

- Verification checks if the mathematical model, as submitted to be solved with FEA, has been correctly discretized and solved.
- Validation determines if an FEA model correctly represents the reality from the perspective of the intended use of the model. It checks if results correctly describe the real-life behavior of the analyzed object.

Verification and validation in the FEA process are pictured in Figure 2.10.

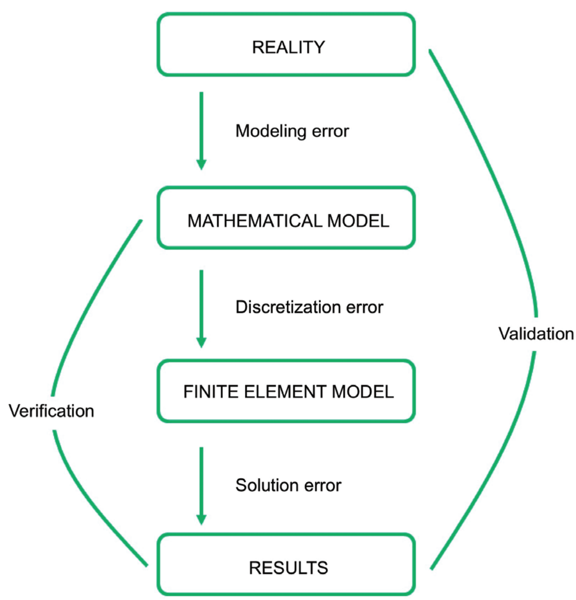


Figure 2.10 Verification and Validation of the FEA results. Verification is concerned with numerical solution of mathematical model. Validation is concerned with how well do the results describe the reality. The above flowchart also shows what errors are introduced at each step of the FEA project.

A model with meshing errors would not pass a verification test. For example, having been discretized into too large of elements, the mathematical model would be solved incorrectly. The verification fails if discretization and/or solution errors invalidate results. Convergence analysis will usually reveal problems causing verification failure; verification problems can be treated by changes to the mesh such as mesh refinement or using higher order elements.

A model with incorrect load definitions would pass the verification test because the verification concerns only itself with the correctness of solution of the mathematical model, not if the mathematical model itself is correct.

Establishing the correctness of a mathematical model along with the correctness of its solution is the process of validation that should follow verification. Validation will fail because of conceptual errors in the definition of the mathematical model. These conceptual errors may escape the modeler's attention, because there is no well-defined structured process to reveal conceptual errors. Our only protection is understanding of the analyzed problem.

Chapter 3
Fundamental Concepts of Finite Element Analysis

3.1 Formulation of a Finite Element

3.1.1 Closer Look at Finite Element

We are looking for displacements in a thin plate. The plate can be modeled as two dimensional (2D) plane stress problem using triangular 2D plane stress elements. The displacement field inside each element and along its edges is described by certain polynomial functions called displacement interpolation functions. The order of the displacement interpolation function used in the element defines the order of element. If the element uses linear (first order) displacement interpolation functions, it is called the first-order element. If element uses second-order displacement interpolation functions, it is called the second-order element, etc.

Arguments of displacement interpolation functions are nodal displacements and once nodal displacements are found, displacements anywhere in the element can be calculated based on nodal displacements. Under a load, the element will deform and assume a new shape, each node will move from its original location to a new one. In the case of a 2D plane stress element, we only need to know x and y displacement components of all nodes to describe element transformation from the old to the new shape. This is because in a 2D element, nodal displacements are fully defined by only two in-plane displacement components. The ability to perform the given translation or rotation is called a degree of freedom (DOF). The three-noded element in Figure 3.1 (top) has 2 DOF/ per

node and the total of 6 DOFs. If second-order polynomials are used for displacement interpolation functions, then element edges do not have to remain straight; they may "bend" assuming curvilinear shape. To describe this, we need to add midside nodes as shown in Figure 3.1 (bottom). Midside nodes differentiate the first-order solid element from the second-order element. This applies to two dimensional as well as to three dimensional (3D) elements as shown in Figures 3.1 and 3.2.

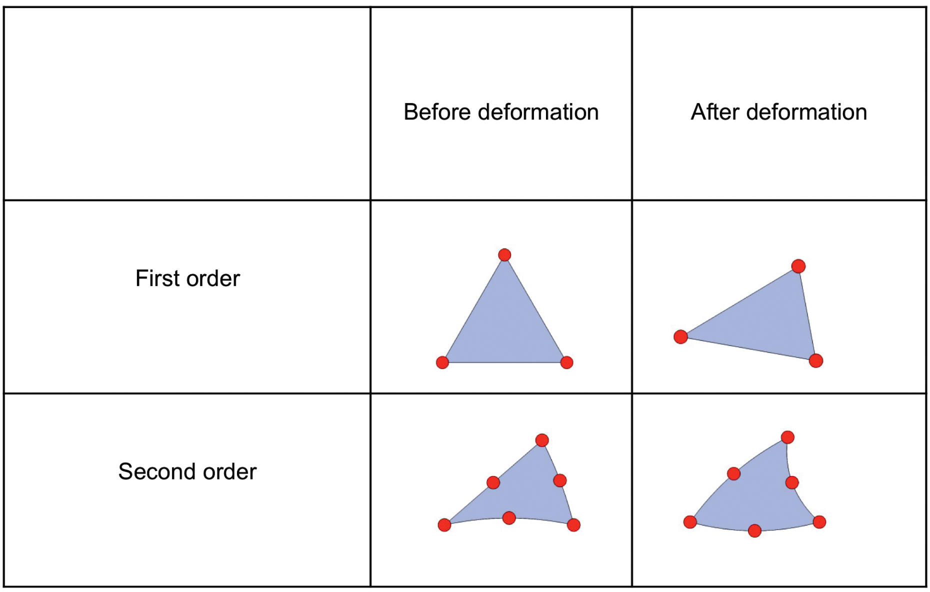


Figure 3.1 A three-noded, first-order 2D element has 2 DOF/ per node: *x* and *y* translations. The total number of DOF is 6, meaning that six independent variables fully describe its transformation from the undeformed to deformed shape. A six-noded second-order 2D element has the same 2 DOF/ per node, but the total number of DOF for the element is 12. Upon deformation, the edges of the second-order element may assume a curved (second-order) shape. Edges of the second-order element may be curved even before deformation if the element is mapped onto a curvilinear geometry. Rotational DOFs are not required to describe deformation of these elements.

Strains are calculated by differentiating displacements. Therefore, if displacements are described by the first-order polynomial functions (linear functions), then strains and stresses are constant within the element. If displacements are described by the second-order polynomial functions, then strains and stresses are distributed linearly within the element (Figure 3.3).

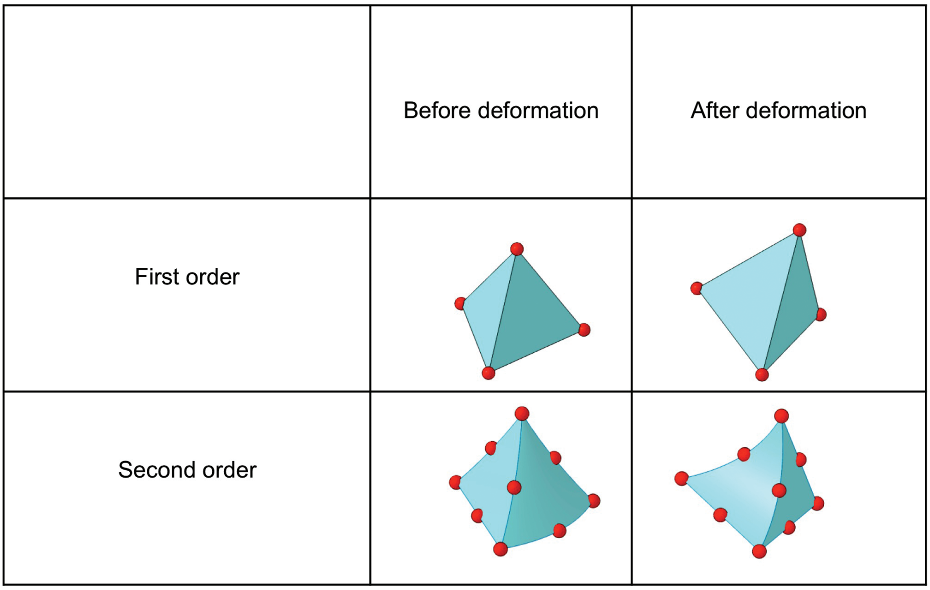


Figure 3.2 A four-noded, first-order 3D solid tetrahedral element (top) has 3 DOF/ per node: *x*, *y*, and *z* translations. In this element, linear displacement interpolation functions model the linear displacement field along the edges, faces, and inside volume of the element. The element has the total of 12 DOFs. A ten-noded second-order 3D solid tetrahedral element (bottom) models the second-order displacement field and requires midside nodes. The element has the total of 30 DOF. The edges and faces of the second-order element may be curved even before deformation if the element is mapped onto a curvilinear geometry. Rotational DOFs are not required to describe element deformation.

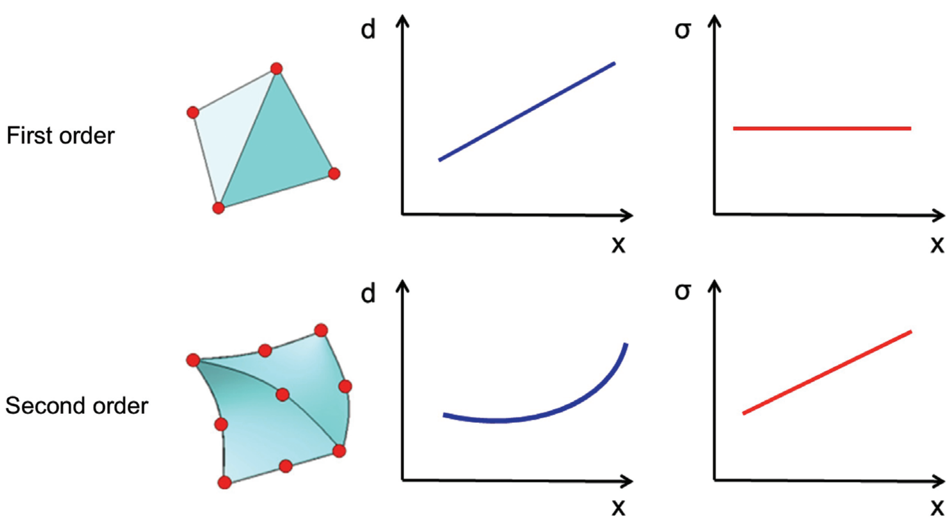


Figure 3.3 The first-order element models linear displacements and constant stress; the second-order element models second-order displacements and linear stress. This is shown in graphs where the ordinate schematically denotes the location in the element.

3.1.2 Requirements to be Satisfied by Displacement Interpolation Functions

The reason why polynomial functions are used as displacement interpolation functions is their versatility. Programmers have considerable freedom selecting particular form of polynomials to build displacement interpolation functions. For example, incomplete polynomials or different-order polynomials along different edges and faces may be used. However, certain requirements must be always met to assure that the displacement field inside the element as well as in the entire mesh is continuous:

- *Internal Compatibility:* Displacement interpolation functions must be continuous over the entire element.
- *Interelement Compatibility:* Displacement along the common edge and the common face between two elements must be described by exactly the same function, so "cracks" or "overlaps" do not form between elements (displacement field is continuous).
- *Rigid Body Motion:* If the element is displaced as a rigid body (without deformation), the element must show zero strain; this is called patch test.
- *Constant Strain:* Displacement interpolation functions must be able to model constant strain cases.

3.1.3 Artificial Restraints

We will illustrate the impact of the choice of element displacement interpolation functions on FEA results using an example of a thin plate with a circular hole. It is represented by a mathematical model (Figure 3.4) as a 2D plane stress problem. Our goal is to find displacements and stresses; we use von Mises stress to present results because the plate material is ductile. For meshing, we use first-order triangular elements capable of modeling linear displacements; the mesh is shown in Figure 3.5. Elements are too large to produce accurate results; we use large elements to make it easy to see the effects that displacement interpolation functions have on the results.

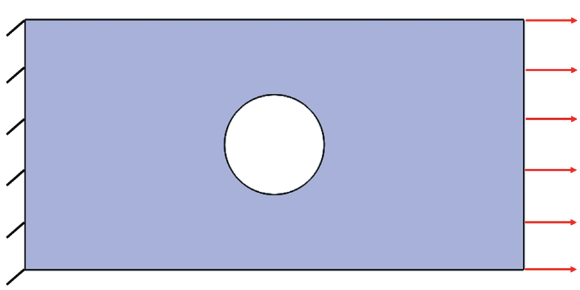


Figure 3.4 A thin, hollow plate is restrained along the left vertical edge and loaded with a tensile load in the horizontal direction applied to the right vertical edge.

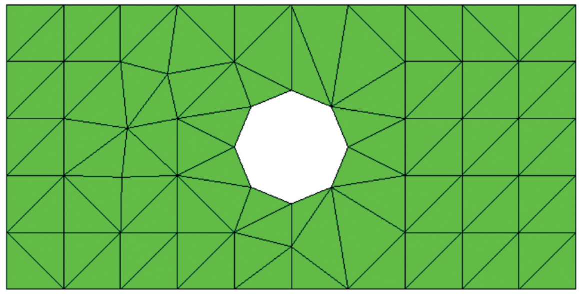


Figure 3.5 A thin, hollow plate meshed with first-order triangular elements and loads and restraints are shown in Figure 3.4. The hole can be represented only approximately because the circle must be replaced by piecewise straight element edges. The mesh is too coarse for any "real" analysis, we use it here only for illustration of the effect of the first-order displacement interpolation functions on displacement and stress patterns.

Having obtained a solution, we observe linearly distributed displacements (Figure 3.6) and constant stresses in each element (Figure 3.7).

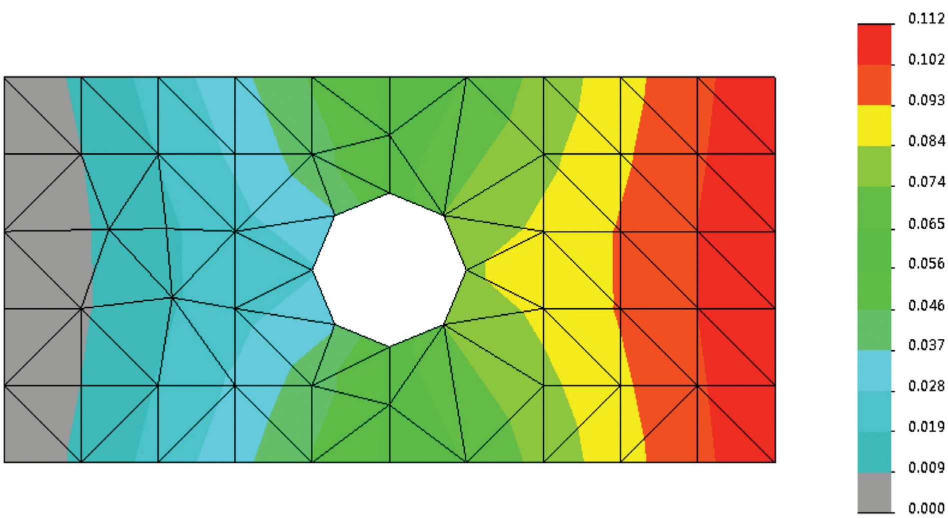


Figure 3.6 Resultant displacement results in the hollow plate. The maximum displacement is 0.112 mm. Notice the linear displacement field in each element as evidenced by straight lines demarcating colors.

The stress result shown in Figure 3.7 may come as a surprise. We never see that "rough" results in FEA reports, but this is because smaller elements are usually used and results are most often "massaged" in preparation for display using the stress averaging technique. The stress averaging technique masks the fact that in the first-order element, stresses are constant in each element and that stress distribution is discontinuous across element boundaries.

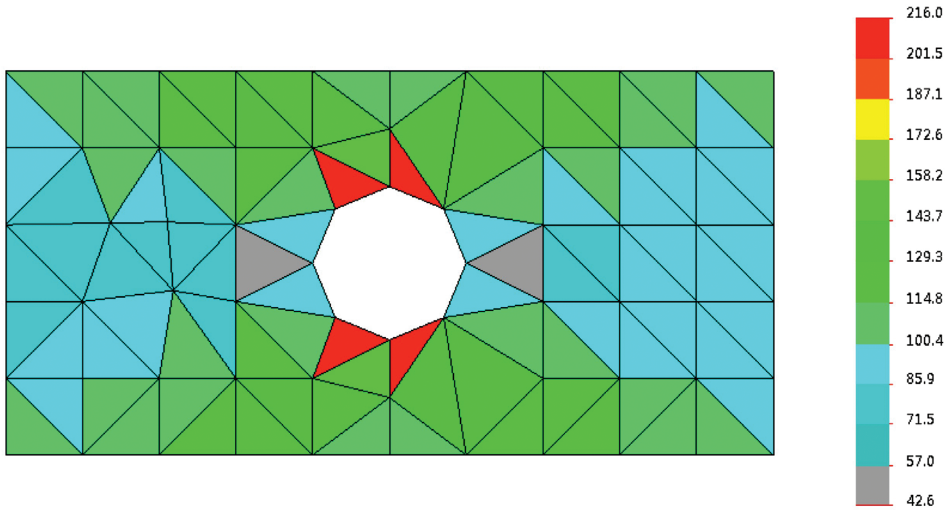


Figure 3.7 Von Mises stress results in a hollow plate; the results show constant stress in each element. The maximum von Mises stress is 217 MPa.

Displacement interpolation functions impose artificial restraints on the FE model because the model has to comply not only with the applied boundary conditions as the mathematical model does but also with the displacement patterns imposed by element definition as seen in Figure 3.6. As a result, the discretized FE model is always stiffer than the corresponding mathematical model.

3.2 The Choice of Discretization

Theoretically, there is an infinite number of ways in which a mathematical model can be turned into FE model by meshing. A particular mesh is a realization of certain choice of discretization. Three major factors define the choice of discretization:

- *Element Size:* The element size is defined by its characteristic dimension. Also important is the relative size of the element in relation to the size of the discretized features. Typically, element size is understood to be the diameter of the smallest circle that can be circumscribed on that element. The element size is commonly denoted by h (Figure 3.8).
- *Element Order:* Element order is defined by the order of displacement interpolation functions describing the displacement field inside the element, along the element edges and element faces.
- *Element Mapping:* Element shape functions are defined for a standard element shape before the element is mapped to assume the actual shape in the finite-element mesh. As a result of mapping, the element shape distorts from that ideal shape (Figure 3.9). Elements outside of the allowed range of shape distortion are called degenerated elements.

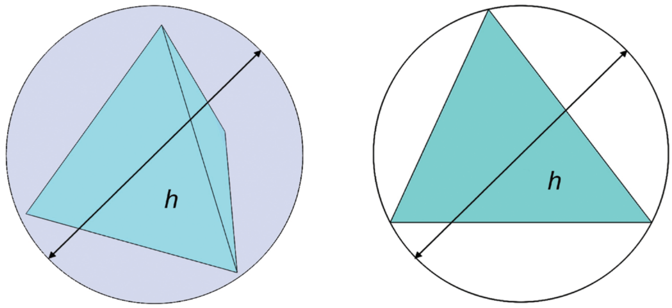


Figure 3.8 The characteristic element size for a tetrahedral element is the diameter of the circumscribed sphere (left). This is easier to illustrate with a 2D analogy of a circle circumscribed on a triangle.

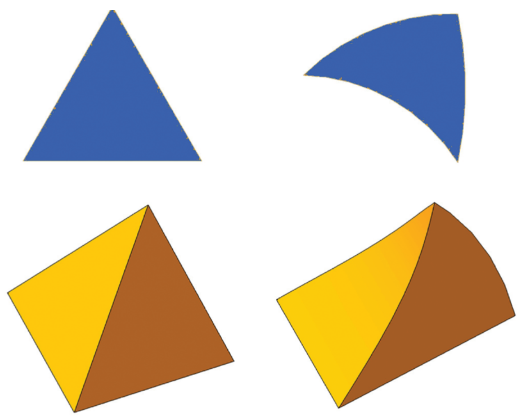


Figure 3.9 Mapping transforms the original element shape (left) into an element conforming to curvilinear geometry in the mesh (right) as illustrated with 2D triangular (top) and 3D tetrahedral elements (bottom).

3.3 Types of Finite Elements

Commercial FEA programs use many different types of elements. The choice of particular element type suitable for the given analysis problem is of utmost importance. There are many ways to classify finite elements; we will present only the most commonly used element classifications.

3.3.1 Element Dimensionality

Classification according to the way how element represents the displacement field in three dimensions distinguishes among 3D elements (solid, shell, membrane, and beam), 2D elements, and 1D elements.

3.3.1.1 3D Solid Element (Figure 3.10): Solid element fully represents all three dimensions. The displacement field in a solid element is 3D and each displacement component is approximated by polynomials of the same degree. Nodes of solid elements

have 3 DOF, which are translations. Three translations are all what are needed to define element transformation from the undeformed to deformed shape.

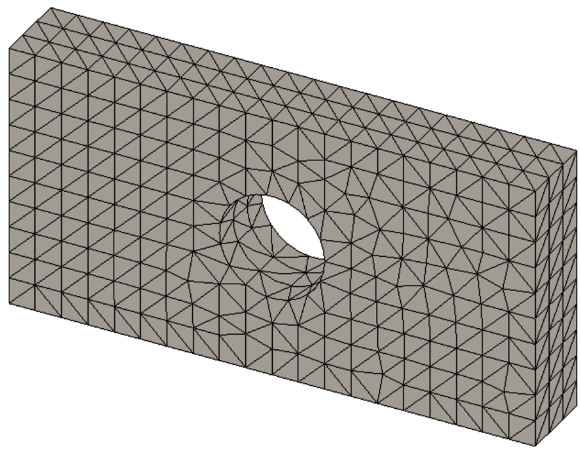


Figure 3.10 3D solid elements.

3.3.1.2 3D Shell Element and 3D Membrane Element (Figure 3.11): The shell element has one dimension collapsed. That collapsed dimension is the thickness of the shell, which is considered small relative to the other dimensions. Assumptions have to be made about stress distribution along that missing dimension. Stresses normal to the shell cross-section are assumed to have linear distribution; consequently, shell element can model bending. Transverse shear stresses are assumed to be constant in thin shell element formulation and parabolic in thick shell element formulation. The shell element models the displacement field with two variables; displacements across the thickness are not modeled.

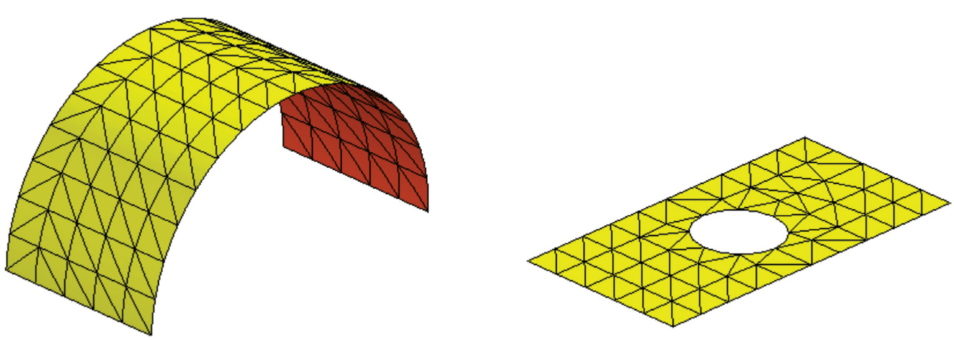


Figure 3.11 The arc in this illustration (left) may be meshed either with shell or with membrane elements; the shell element and the membrane element have the same appearance; flat shell elements (right) look exactly the same as 2D plate elements. Note that the elements shown in these illustrations would be too large for analysis.

3.3.1.3 3D Membrane Element: The membrane element looks the same as a shell element, but stresses normal to the membrane cross-section are assumed to be constant; transverse shear stresses are not modeled. As the name implies, the membrane element can only model membrane stresses but not bending stresses. Shell and membrane elements have 6 degrees of freedom (DOF)/ per node: three translations and three rotations. The difference in how stresses normal to the cross section are modeled in the shell and membrane elements is shown in Figure 3.12.

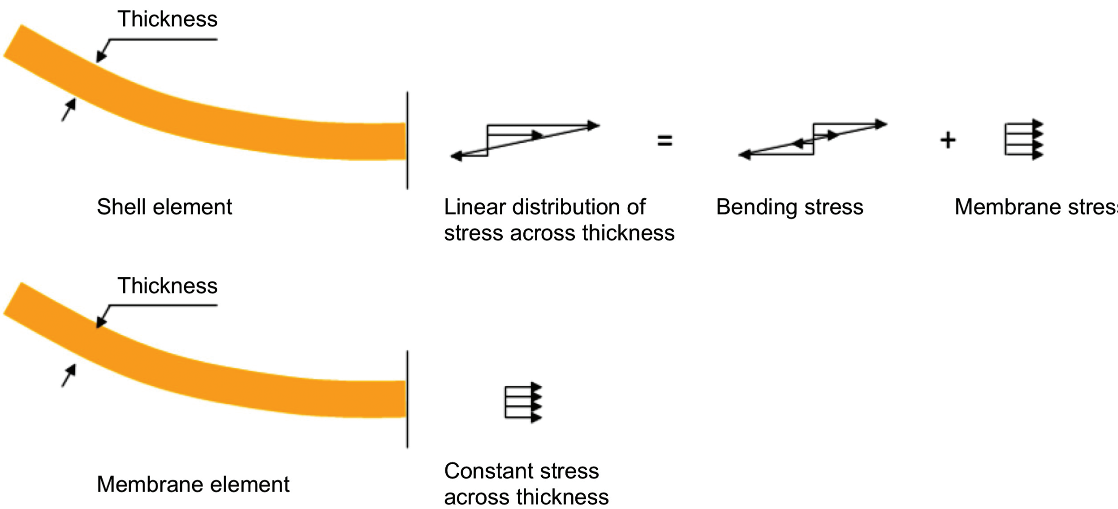


Figure 3.12 The shell element (top) models the linear distribution of stress normal to cross section. These stresses can be separated into bending stress component, where the magnitude of stress on the opposite sites is the same but the direction is different, and membrane stress component, where the stress is constant across element thickness. The shell element has both bending stiffness and membrane stiffness. Membrane element (bottom) can model only constant stress across the thickness. The membrane element has only membrane stiffness. Both illustrations show element thickness even though thickness is not represented by a physical dimension.

3.3.1.4 3D Beam Element: The beam element has two dimensions collapsed. It is assumed that the width and height of cross section are small in comparison to the length. The displacement field is 3D; we assume a known linear variation of stress in two directions perpendicular to the cross section normal to the beam. The beam element models 3D displacement field with one variable. Beam element can be seen as a line with cross-sectional properties (cross-sectional area and second moments of inertia) added. Beam elements have 6 DOF/ per node: three translations and three rotations, the same as shell elements.

The visual representation of beam elements differs between different FEA programs. Some examples of how beam element mesh may be presented visually are shown in Figure 3.13.

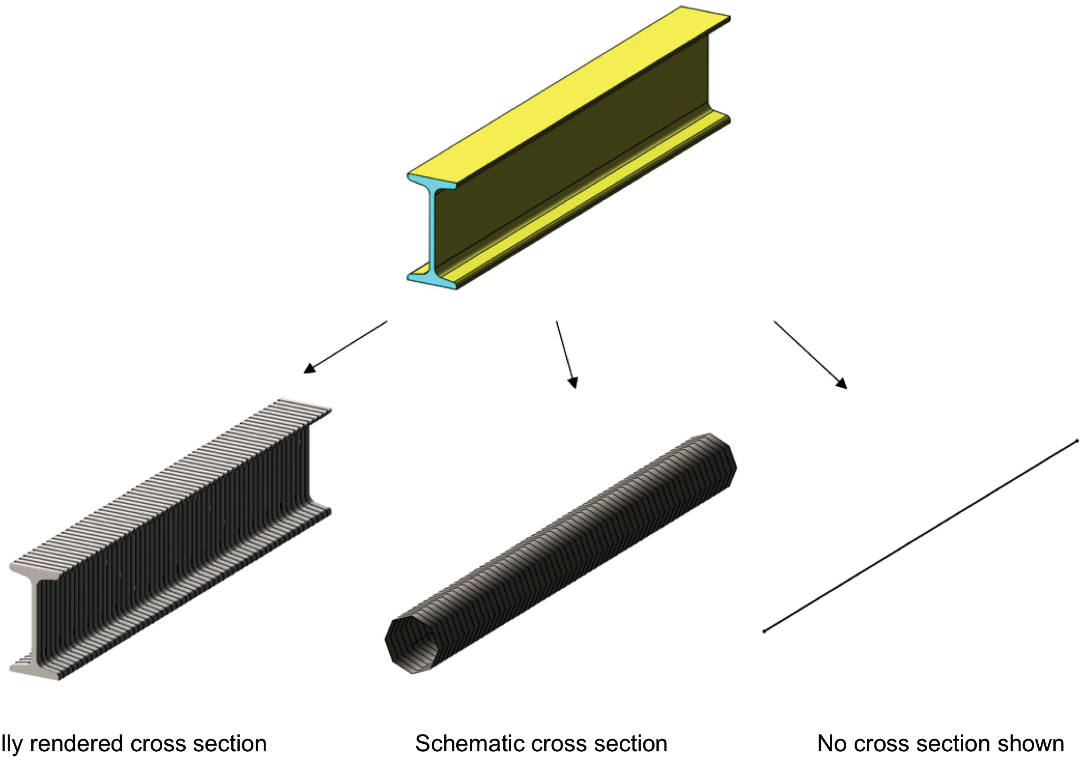


Figure 3.13 An appearance of the beam element may differ in how beam cross section is represented. The cross section may be fully rendered; it may be shown schematically (here shown as a circle) or not shown at all.

3.3.1.5 2D Elements: There are cases where a structure's response to load can be fully described in two dimensions; this is when 2D elements are used. 2D elements fall into three categories: 1) plane stress, 2) plane strain, and 3) axisymmetric elements.

Plane stress elements are intended for the analysis of thin planar structures loaded in plane, where the out-of-plane stress is assumed to be equal zero. Plane strain elements are intended for the analysis of thick prismatic structures loaded in plane, where the out-of-plane strain is assumed to be equal zero. Axisymmetric elements are intended for the analysis of axisymmetric structures under axisymmetric load. In all of these cases, the structure deformation can be fully described using elements with only 2 DOF/ per node. For plane stress and plane strain, these are two components of in-plane translation. For axisymmetric elements, these are radial and axial displacements. The 2D plane stress

model, 2D plane strain model, and axisymmetric model are shown in Figures 3.14–3.16, respectively.

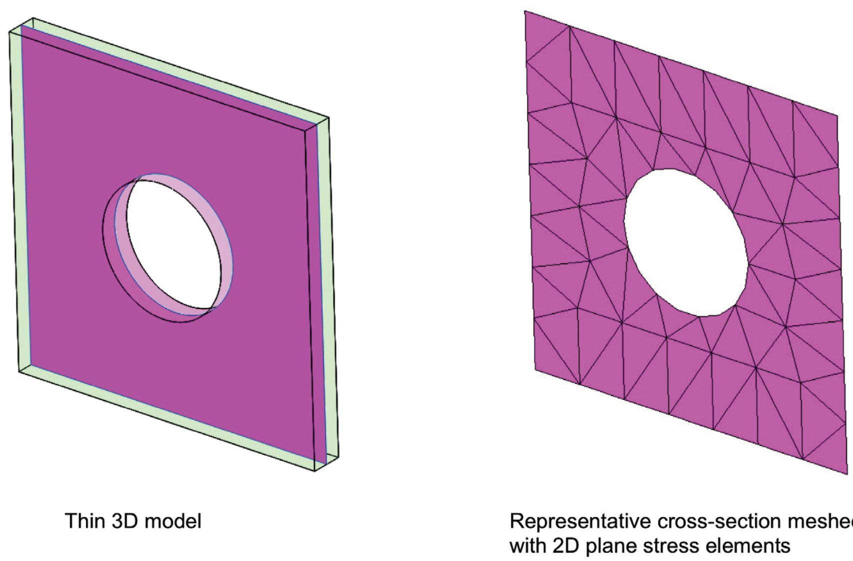


Figure 3.14 2D plane stress elements. Large elements are shown for clarity of this illustration. They would be too large for analysis.

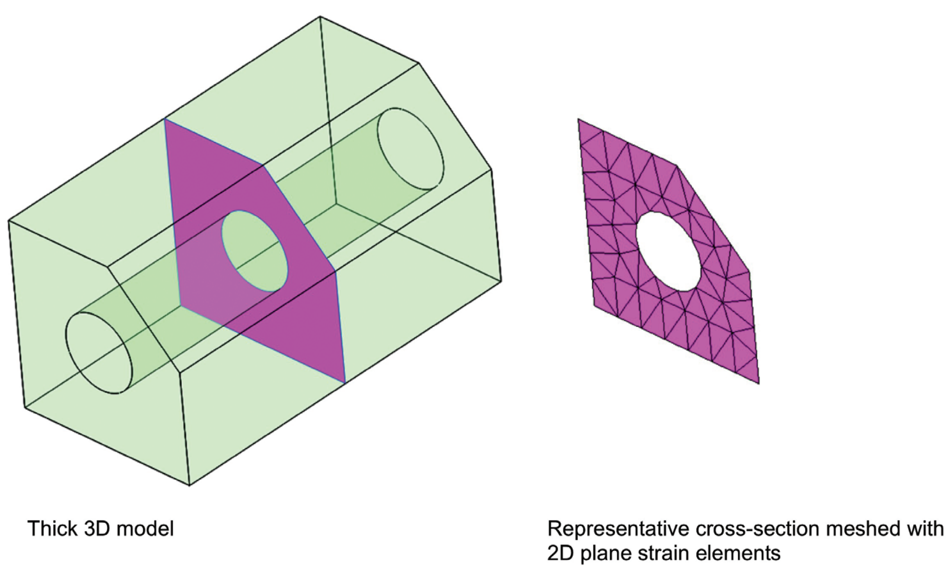


Figure 3.15 2D plane strain elements. Large elements are show for clarity of this illustration. They would be too large for analysis.

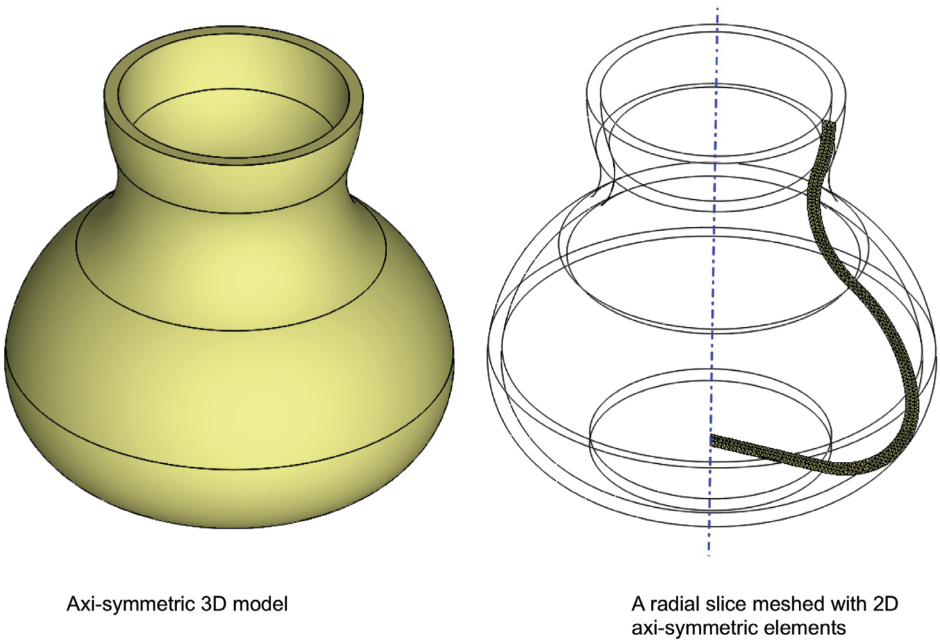


Figure 3.16 2D axisymmetric elements.

3.3.1.6 1D Elements: 1D elements have no practical applications, but because of their simplicity, they are used extensively in introductory FEA courses. 1D elements have only 1 DOF/ per node; all elements in the 1D element mesh must be aligned along a straight line. An example of the 1D element mesh that can be found in every introductory FEA textbook is shown in Figure 3.17.

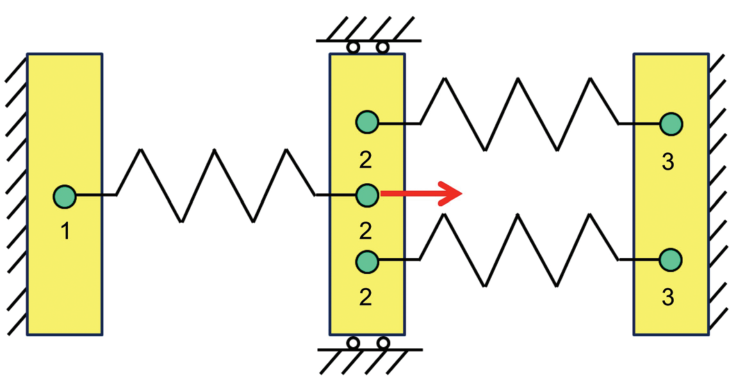


Figure 3.17 Mesh of three 1D elements shown as springs; nodes 2 and 3 are "spread out" vertically only for clarity of this illustration.

3.3.2 Element Shape

Theoretically, an element of any shape could be designed. However, for practical reasons, only simple element shapes are used because only those can mesh any geometry. Therefore, 2D elements come as triangles and quadrilaterals, 3D elements come as tetrahedral (tetras), and pentahedral (prisms) and hexahedral (brick) elements. Tetrahedral, pentahedral and hexahedral elements are illustrated in Figure 3.18.

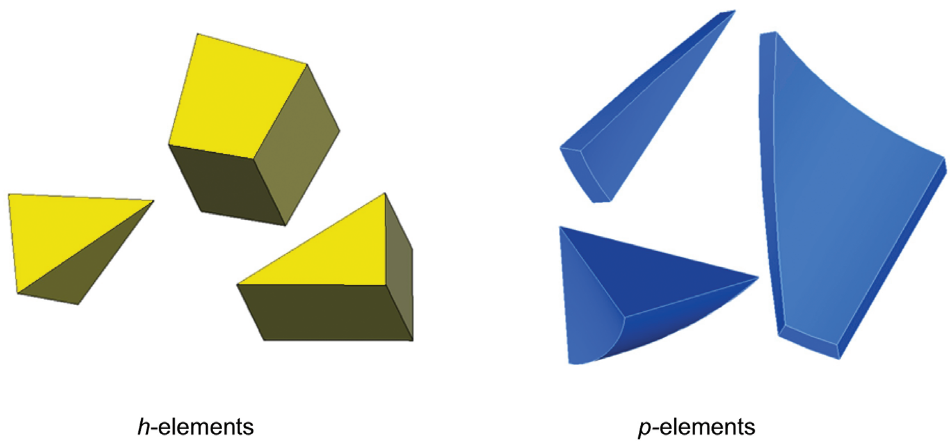


Figure 3.18 Shapes of *h* elements and *p* elements are limited to simple primitives: tetrahedron, pentahedron and hexahedron. *p* elements are allowed much higher distortion from the ideal shape when mapped onto model geometry.

Automeshers work most reliably with tetrahedral elements for volume meshing and triangular elements for surface meshing, and for this reason, tetrahedral and triangular elements are most often used element shapes.

3.3.3 Element Order and Element Type

Element order is defined by the order of displacement interpolation functions used by the element. The first-order element uses the first-order displacement interpolation functions; the second-order element uses second-order displacement functions, etc. Element type denotes if the element order is fixed or if it can be changed without the need for remeshing. Elements for which the order is fixed are used by *h* version of the FEM and are called *h* elements. Elements for which the order can be changed automatically are used by *p* version of the FEM and are called *p* elements. In most of the commercial implementations of h version of the FEM, elements are set to either the first or second order. In *p* version of the FEM, elements can be automatically upgraded to higher orders anywhere between the fifth and the tenth order depending on the particular software implementation. *h* and *p* elements can be usually told apart by appearance as shown in Figures 3.19 and 3.20. Name *h* comes from characteristic element size, name *p* comes from polynomial displacement interpolation function.

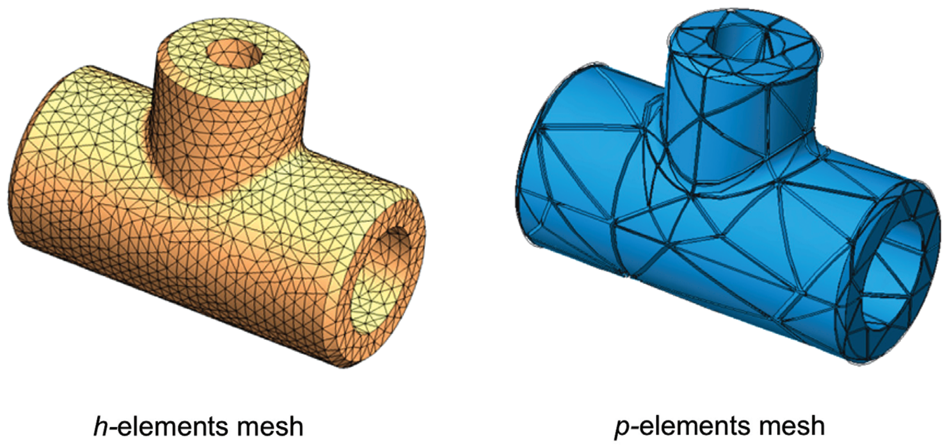


Figure 3.19 The same model meshed with solid tetrahedral *h* elements (left) and solid tetrahedral *p* elements (right). Element shrinkage has been applied to the *p* element mesh.

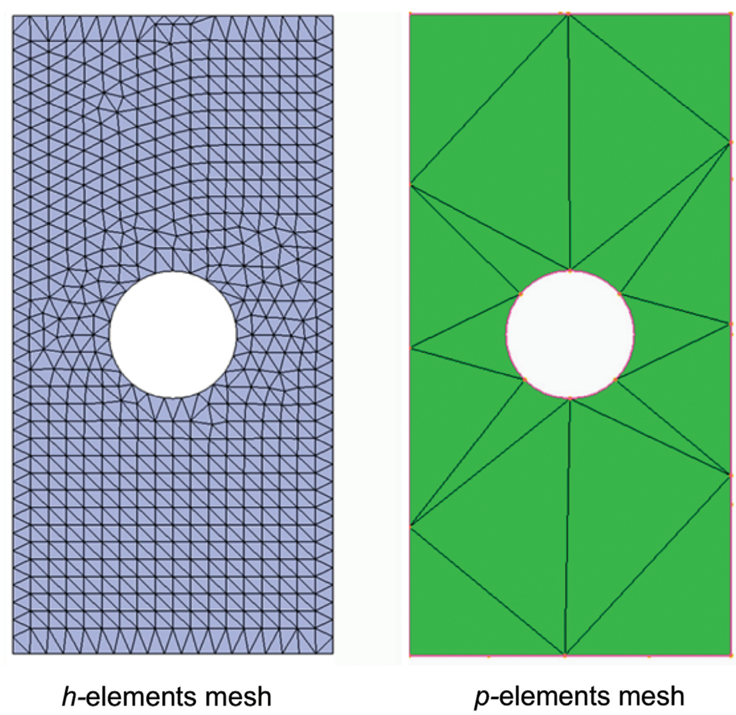


Figure 3.20 The same model meshed with triangular shell *h* elements (left) and triangular shell *p* elements (right).

The *h*-method uses elements with low shape distortion modeling the first- or second-order displacement fields. Those two factors combined (elements must not be too distorted and the displacement field in the element is of low order) require that a large number of small elements must be used to represent the expected displacement and stress patterns. Even though meshing of the prepared geometry is most often done automatically, it is the user's responsibility to determine whether the mesh is good enough to deliver the sought results. The *p*-method uses elements of more complex shapes and more complex displacement fields.

Differences between the *h* and *p* elements are summarized in Table 3.1.

Table 3.1 Differences between *h* and *p* elements

	h elements	*p* elements
Element shape	Tetrahedron, pentahedron, hexahedron	Tetrahedron, pentahedron, hexahedron
Mapping	Only low deviation from the ideal shape is allowed	Higher deviation from the ideal shape is allowed, but this may introduce errors on highly curved edges and surfaces
Displacement/temperature field	Described by lower order polynomials (first or second); the polynomial order does not change during the solution	Described by higher order polynomials; the polynomial order is adjusted automatically to meet the accuracy requirements
Convergence process	Mesh refinement	Element order upgrade
Implementation date	Since day one of the FEA	Since the early 1990s

3.3.4 Summary of Commonly Used Elements

The majority of commercial FEA programs use *h* elements; notable exceptions are Creo/Simulate solely based on *p* elements and StressCheck that can use both use *h* and *p* elements.

The summary of commonly used *h* elements and element orders is given in Figure 3.21. Notice that the element order does not apply to beam elements; transverse displacements along the beam element are always of the third order.

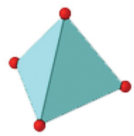
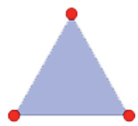
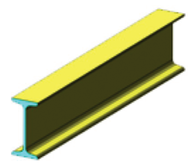
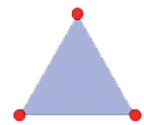
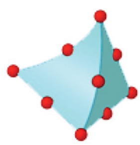
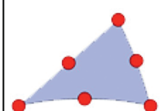
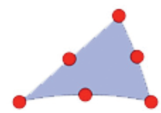
Element order	3D elements			2D elements
	Solid elements	Shell elements	Beam elements	Plate elements
First order Linear displacement Constant stress				
Second order Parabolic displacement Linear stress				

Figure 3.21 Classification of commonly used *h* elements according to dimensionality and order.

3.3.5 Element Modeling Capabilities

Elements can be also classified based on their capabilities to support certain types of analysis. In this way, we distinguish between linear and nonlinear material elements, small and large strain elements, thin and thick shell elements, etc. There are also elements developed for specific purposes. This group includes elements modeling concentrated mass, spring elements, rigid connection elements (rigid links), gap elements, and many more. The graphic representation of those special elements depends on the FEA program. Figure 3.22 shows spring, mass, and link elements as they are pictured in SOLIDWORKS simulation.

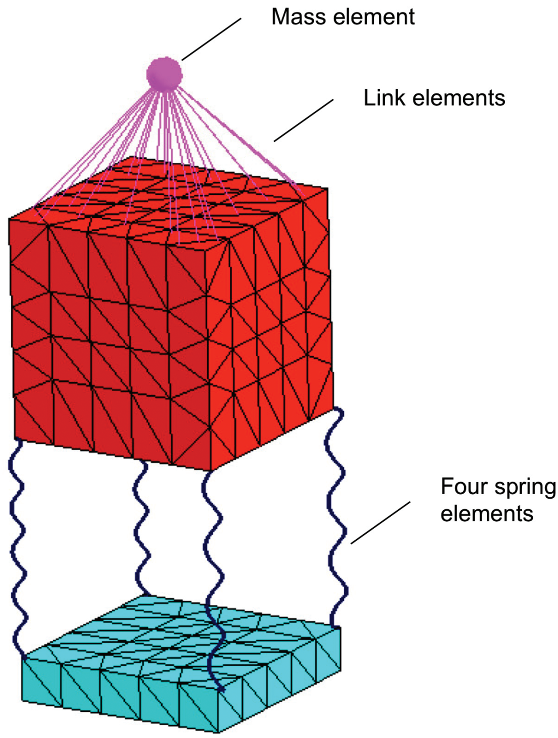


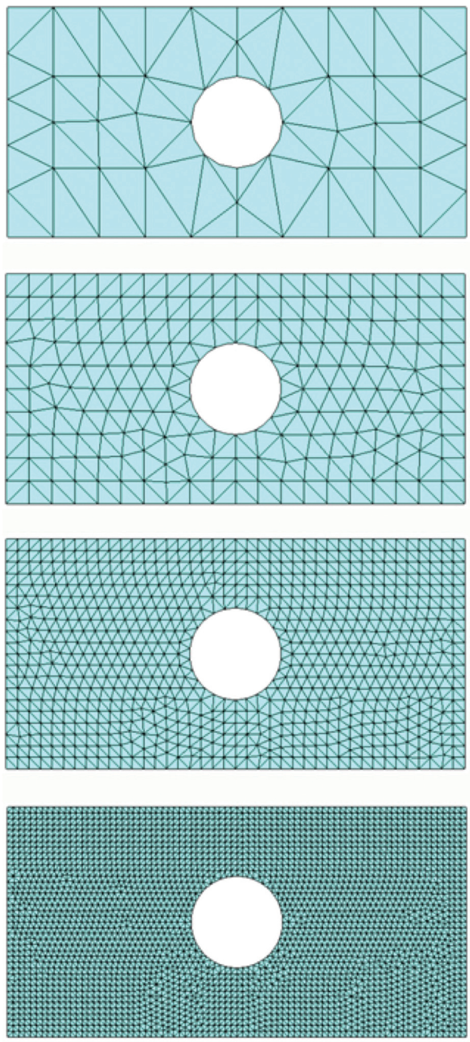
Figure 3.22 Four spring elements connect the blue base and the red cube. The point mass is located above the top face of cube; it is attached to the top face with rigid link elements. The base (blue) and the cube (red) are meshed with solid tetrahedral elements.

Chapter 4
Controlling Discretization Errors

Modeling errors, discretization errors, and solution errors, summarized in Figure 2.10, all affect FEA results. However, only discretization errors are specific to the FEA and only discretization errors can be controlled using FEA tools. For this reason, we will discuss them first and the discussion of other errors will follow.

The objective of discretization error control is to find out how is the data of interest such as displacement or stress, dependent on the choice of discretization (mesh). Note that the objective is not to minimize the discretization error but to obtain a solution where the data of interest do not significantly depend on the choice of discretization. The solution cannot be considered as reliable unless we have a reliable estimate of the discretization error. The analysis of a discretization error is done in a convergence process.

An example of different choices of discretization is shown in Figure 4.1 where the same model is meshed with four meshes, each one with a different element size.



Element size 20mm

Element size 10mm

Element size 5mm

Element size 2.5mm

Figure 4.1 The same model meshed with shell elements of different sizes; this illustrates a global mesh refinement.

4.1 Presenting Stress Results

Before we discuss convergence process and its variations, we need to look at different ways of presenting stress results.

As we already know, nodal displacements are computed first. Strains and then stresses are calculated based on displacement results. Stresses are first calculated inside the element at certain locations called Gauss points. Next, stress results are extrapolated

to the elements' nodes. If one node belongs to more than one element (which is always almost always the case), then the stress results from all the elements sharing a given node are averaged and one stress value, called a node value, is reported for each node. This stress value is called a nodal stress or averaged stress.

An alternate procedure to present stress results is by obtaining stresses at Gauss points, then averaging them in between themselves. This means that one stress value is calculated for the entire element. This stress value is called an element stress or non-averaged stress because stresses are not averaged between neighboring elements. Gauss points are locations used in numerical generations of the element stiffness matrix; the location of Gauss points in an element is schematically shown in Figure 4.2.

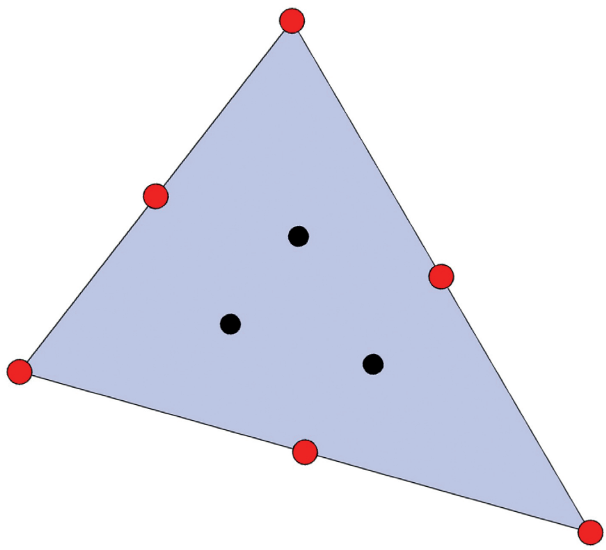


Figure 4.2 Schematic locations of the Gauss point in the second-order 2D element. The red dots denote element nodes, and the black dots denote Gauss points.

Nodal stresses are used more often because they offer smoothed out, continuous stress results. However, examination of element stresses provides an important feedback on the quality of the results. If element stresses in two adjacent elements differ too much, it indicates that the element size at this location is too large to properly model the stress gradient.

Nodal and element stresses in mesh 1 from Figure 4.1 are shown in Figure 4.3. The model is restrained along the left vertical edge and subjected to a uniform tension applied to the right vertical edge. Large elements are used for clarity of this illustration.

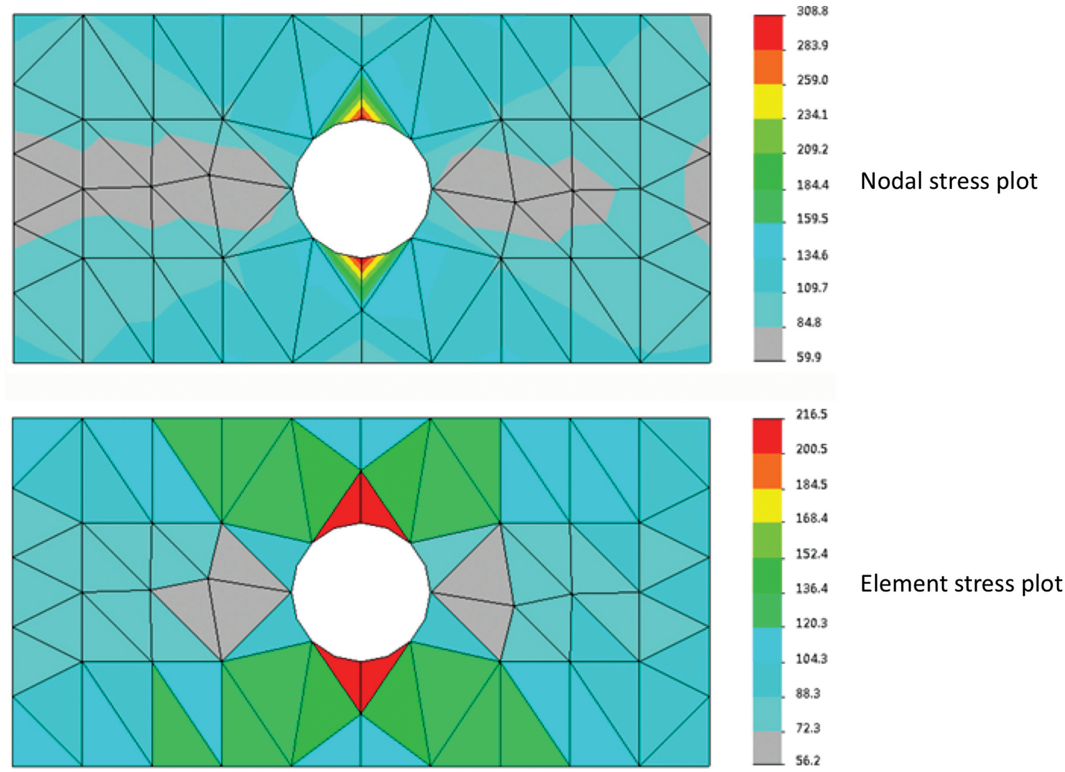


Figure 4.3 Nodal stress plot (top) and element stress (bottom) produced by a coarse mesh of second-order elements. The model represents a hollow plate in tension. Large elements are used for clarity of illustration; these elements are too large to produce any meaningful results.

4.2 Types of Convergence Process

Discretization errors can be found in the process of making systematic changes to the choice of discretization and studying the impact of those changes on the data of interest. The process is called the convergence process; there are many variations in the convergence process.

4.2.1 *h* Convergence by Global Mesh Refinement

On way to make systematic changes to the choice of discretization is to modify the element size through mesh refinement. As *h* denotes the characteristic element size as shown in Figure 3.8, the convergence process through mesh refinement is called *h* convergence process. In this process, the size of elements is gradually reduced. We will illustrate it with the already familiar example of the hollow plate. The plate is supported along the left vertical edge and loaded with the tensile load uniformly distributed along the right vertical edge. Figure 4.1 shows the model meshed with four different meshes. With every mesh refinement, the element size is halved, meaning that each triangular element is replaced with four smaller triangular elements.

Step 1 in the *h* convergence process uses the element size of 20 mm. Step 4, which is the last step, uses the element size of 2.5 mm. The displacement results are summarized in Figure 4.4 and von Mises stress results are summarized in Figure 4.5. Shell elements were used to produce these results, but as the model is flat, loaded in plane, and thin, the 2D plane stress element could have been used as well.

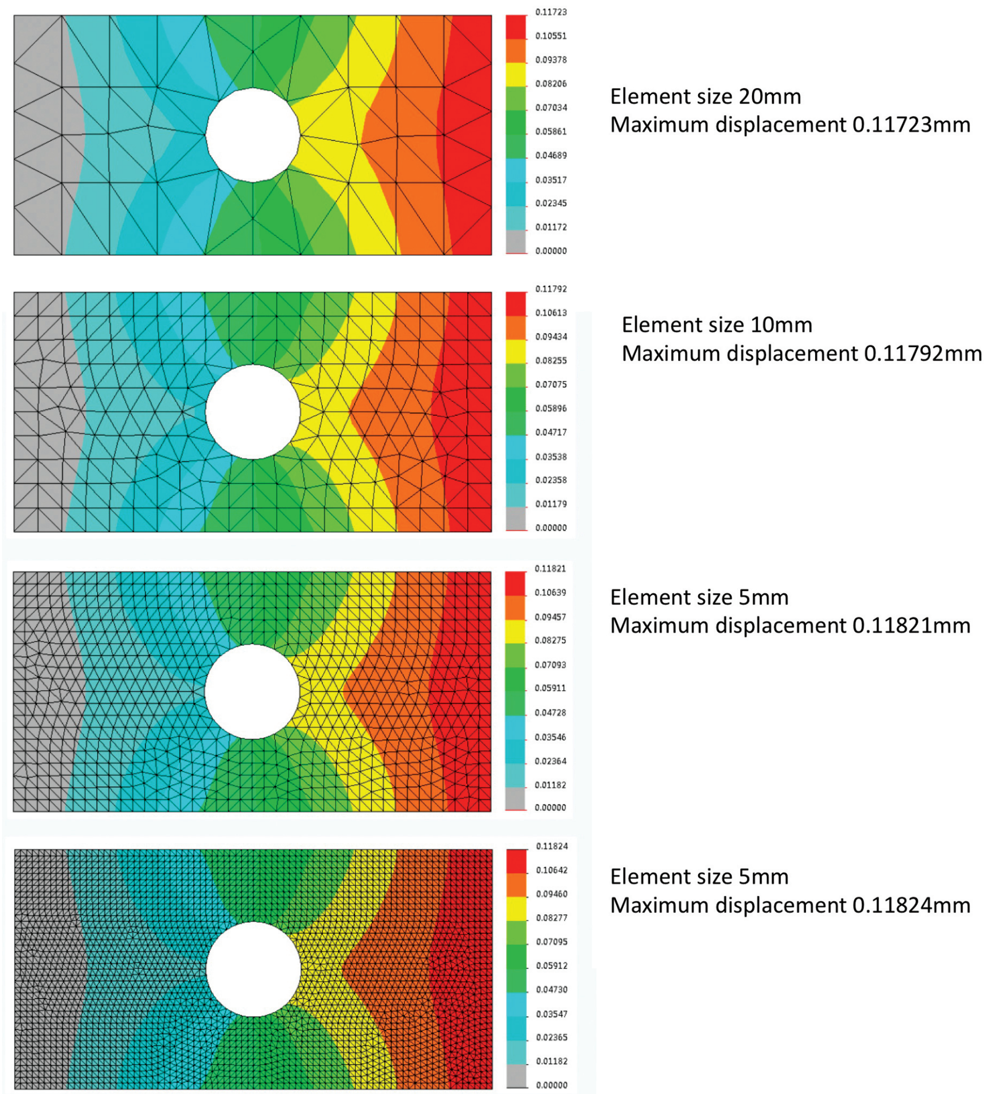


Figure 4.4 Displacement results produced by four different meshes representing the same problem; this illustrates the *h* convergence process by global mesh refinement.

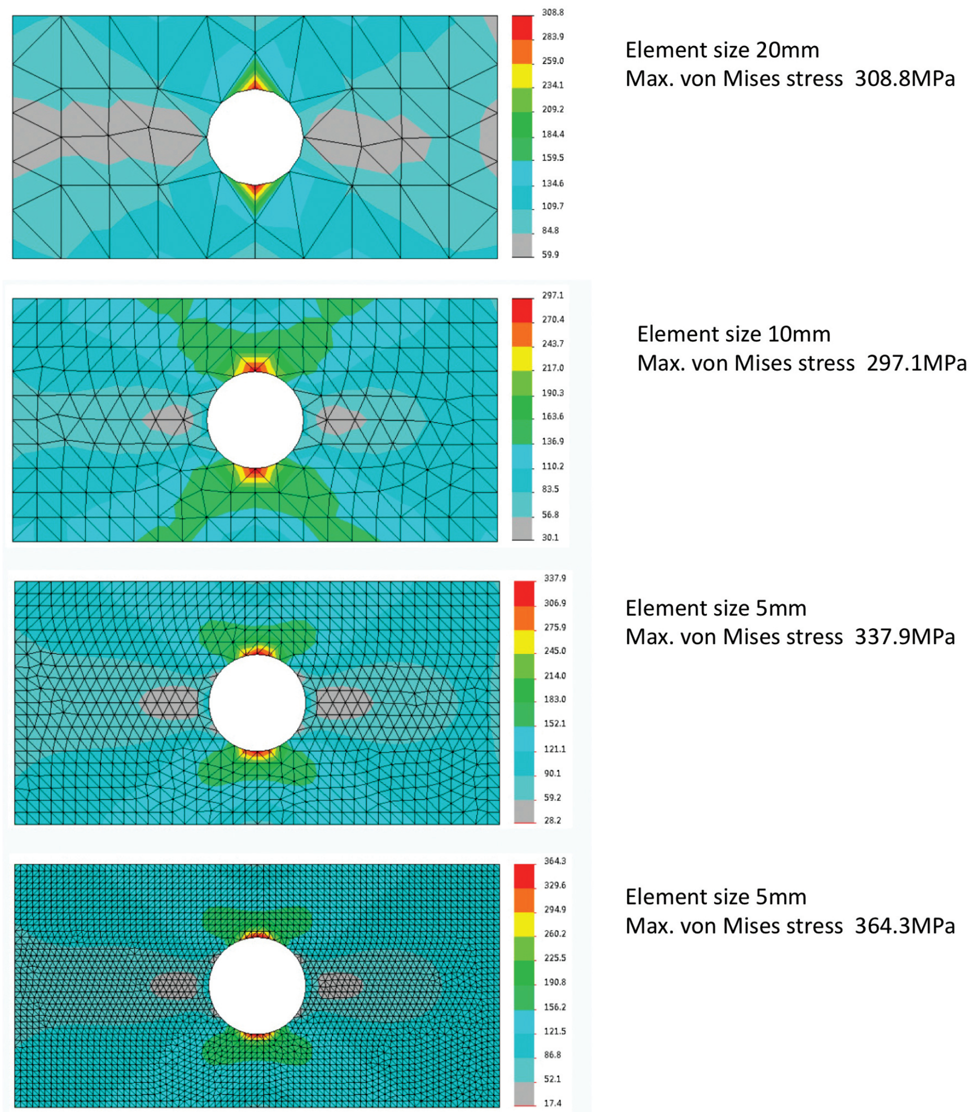


Figure 4.5 Von Mises stress results produced by four different meshes representing the same problem. this illustrates the *h* convergence process by global mesh refinement.

Having performed four iterations in the *h* convergence process, we can summarize the results in Table 4.1.

Table 4.1 Summary of the displacement and stress results produced in four steps of the *h* convergence process

Mesh #	Element size mm	1/*h*	# of nodes	# of elements	#DOF	Max horizontal displacement mm	Displacement convergence error	Max von Mises stress MPa	Stress convergence error
1	20	0.05	194	78	1098	0.11723	unknown	308.8	unknown
2	10	0.1	822	374	4806	0.11792	0.59%	297.1	3.94%
3	5	0.2	3142	1498	18606	0.11821	0.25%	337.9	12.07%
4	2.5	0.4	12284	5596	73218	0.11824	0.03%	364.3	7.25%

Notice that the total number of degrees of freedom is not exactly equal to the number of nodes times the number of degrees of freedom per node (here 6 DOF/ per node) because some degrees of freedom are eliminated by the support and, therefore, not counted. The data of interest (here the maximum displacement and the maximum von Mises stress) may be plotted against the number of degrees of freedom or some other related measure. The graph in Figure 4.6 shows the maximum displacement in the direction of load as a function of the inverse of the element size. The graph in Figure 4.7 shows the maximum von Mises stress as a function of the inverse of the element size.

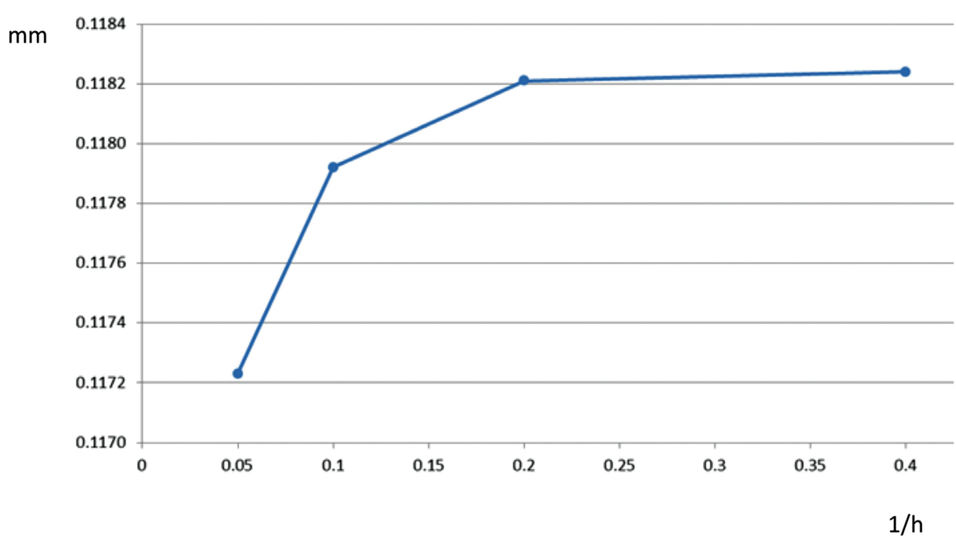


Figure 4.6 Global mesh refinement and convergence of maximum displacement. The maximum displacement is graphed as a function of 1/*h*, where *h* is the element size.

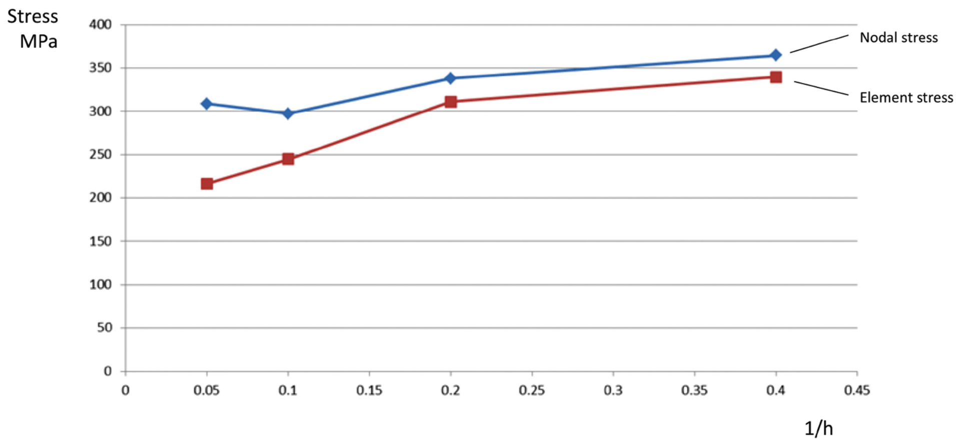


Figure 4.7 Global mesh refinement and convergence of maximum von Mises stress. The maximum stress is graphed as a function of 1/*h*, where *h* is the element size.

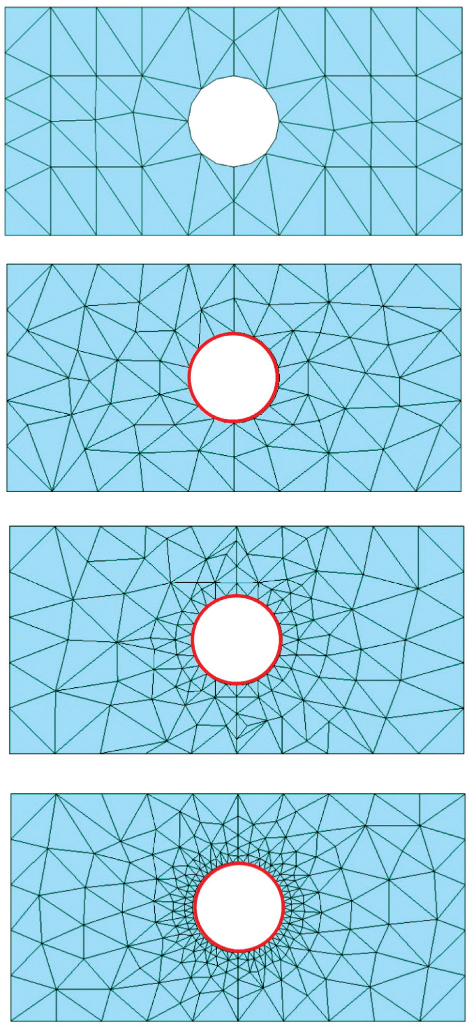
In a discretized model, displacements have to follow displacement pattern designed in the elements used for meshing. This adds some (very low) stiffness, which we call artificial stiffness. Every mesh refinement reduces that artificial stiffness and this explains why the maximum displacement increases, as shown in Figure 4.6. Strain is found as a derivative of displacement and stress is calculated once strain has been found; therefore, stress also increases with mesh refinement. Note the "dip" in the nodal stresses graph shown in Figure 4.7 caused by stress averaging, which is a part of nodal stress calculations.

Having completed the four steps of the *h* convergence process by global mesh refinement, we notice that displacements converge faster than stresses. To explain this, we need to differentiate between the global and local FEM results. The maximum displacement is a global result; stiffness of the entire model contributes to the maximum displacement results. The global results converge fast with mesh refinement. The maximum stress (here von Mises stress) is a local result and is modeled only by a few elements; for a faster convergence of the maximum stress, we would need a more aggressive refinement in the area of stress concentration.

4.2.2 *h* Convergence Process by Local Mesh Refinement

An alternative to refining mesh everywhere is a local mesh refinement that produces meshes shown in Figure 4.8. Local mesh refinement adds fewer degrees of freedom to the model compared with the global mesh refinement and consecutive models solve faster. The *h* convergence process by local mesh refinement is easier to execute compared to using global mesh refinement as long as we know where to apply the mesh refinement; the *h* convergence process by local mesh refinement requires a prerequisite

knowledge of stress pattern in the analyzed model. There is a risk of not finding stress concentrations if local mesh refinement is applied in an incorrect location.



Element size 20mm
No mesh bias

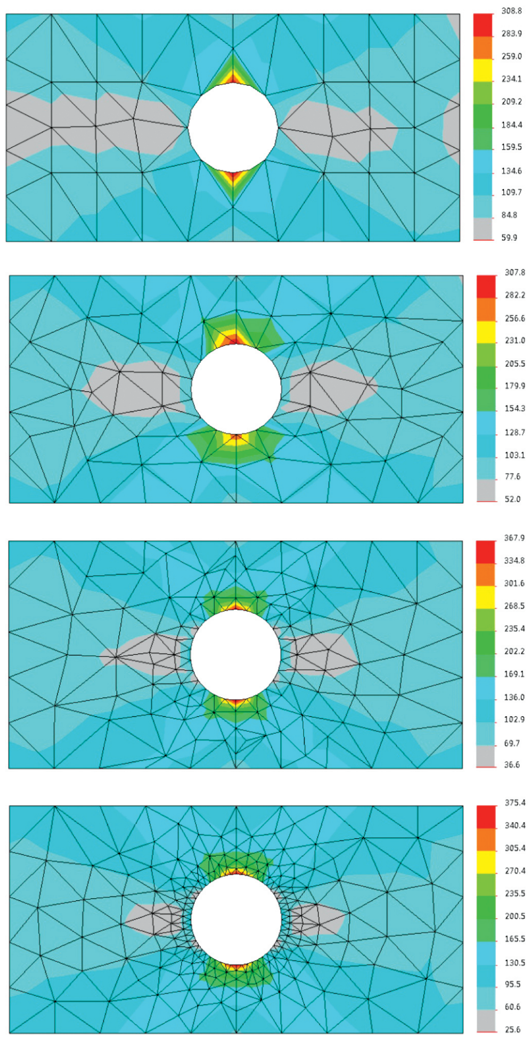
Element size 20mm
Mesh bias 10mm along the red edge

Element size 20mm
Mesh bias 5mm along the red edge

Element size 20mm
Mesh bias 2.5mm along the red edge

Figure 4.8 The same model meshed with shell elements of different sizes along the controlled entity (edge of the hole). This illustrates a local mesh refinement. Mesh bias expressed in mm characterizes the element size on the entity where mesh bias has been applied.

The von Mises stress results produced by four meshes used in the *h* convergence process by local mesh refinement are shown in Figure 4.9; the convergence graph of the maximum von Mises stress (nodal) is shown in Figure 4.10.



Element size 20mm
No mesh bias
Max. von Mises stress 308.8MPa

Element size 20mm
Mesh bias 10mm
Max. von Mises stress 307.8MPa

Element size 20mm
Mesh bias 5mm
Max. von Mises stress 367.9MPa

Element size 20mm
Mesh bias 2.5mm
Max. von Mises stress 375.4MPa

Figure 4.9 Von Mises stress results produced by four different meshes representing the same problem. This illustrates the *h* convergence process by local mesh refinement.

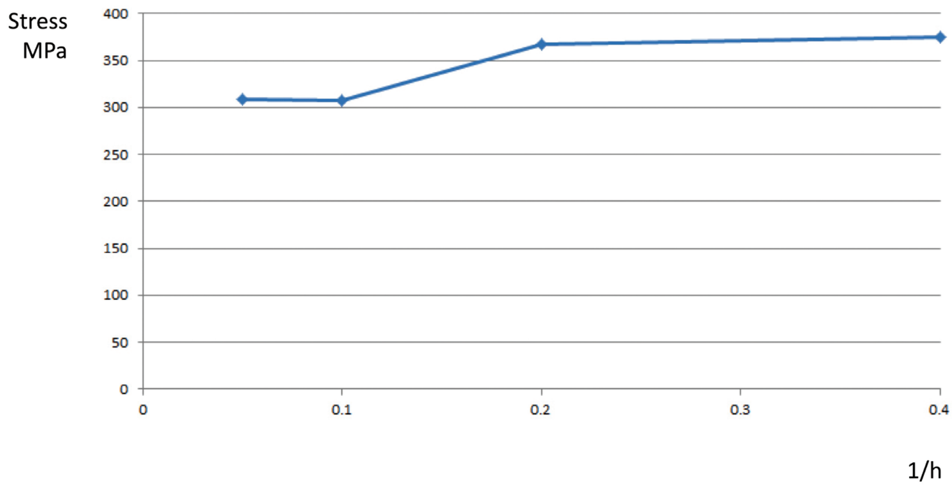


Figure 4.10 Convergence of maximum nodal von Mises stress during the *h* convergence process by local mesh refinement. The maximum stress is a function of 1/*h*, where *h* is the element size. "Dip" is stress in the second iteration is caused by stress averaging.

4.2.3 Adaptive *h* Convergence Process

The *h* convergence process by global mesh refinement is time consuming; the number of degrees of freedom increases with every mesh refinement and the model grows is size very fast. The *h* convergence process by local mesh refinement is faster because local mesh refinement has less effect on the number of degrees of freedom in the analyzed model, but it requires the user's decision on where to refine the mesh. This disadvantage of local *h* convergence process is eliminated if the adaptive *h* convergence process is used.

In the adaptive *h* convergence process, the mesh is refined automatically during an iterative solution. The initial mesh is defined by user and a solution is obtained; let us call it iteration #1. Based on errors found in this first solution, the mesh is automatically refined in the locations characterized by high errors and a new solution is obtained; this is iteration #2. Mesh refinements continue in the subsequent iterations until the user specified accuracy has been satisfied or the number of the allowed iterations has been reached. The user does not have a direct control over how the final mesh will look like.

The name "adaptive" derives from the fact that mesh refinement is adapted the stress errors found. The measure of error is related to the difference between nodal stress and element stresses.

Consecutive mesh refinements during the *h* adaptive convergence process are illustrated in Figure 4.11. A hollow plate is subjected to a uniform tension causing stress concentrations on the cylindrical surface of the hole and the mesh is automatically refined there. There are important differences between this problem and that shown in Figures 4.5 and 4.9. First, the model is subjected to a tension on both end faces and stresses are perfectly

symmetric. Second, because of a larger thickness, the problem requires 3D solid elements rather than 3D shell elements or 2D plate elements. Lower stresses are reported because the thickness has increased while the load has remained the same.

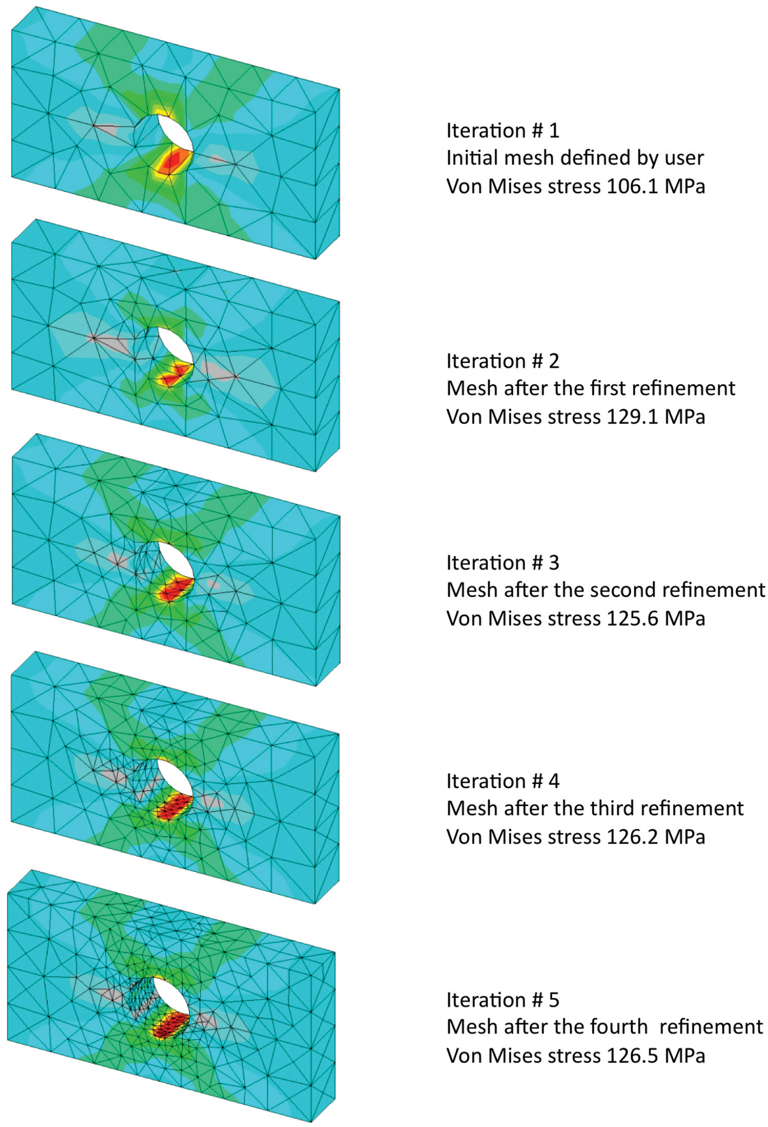


Figure 4.11 Von Mises stresses in five iterations of the *h* adaptive convergence process.

A summary of the von Mises stress results from Figure 4.11 is shown in graph in Figure 4.12, where von Mises stress is as a function of the iteration number.

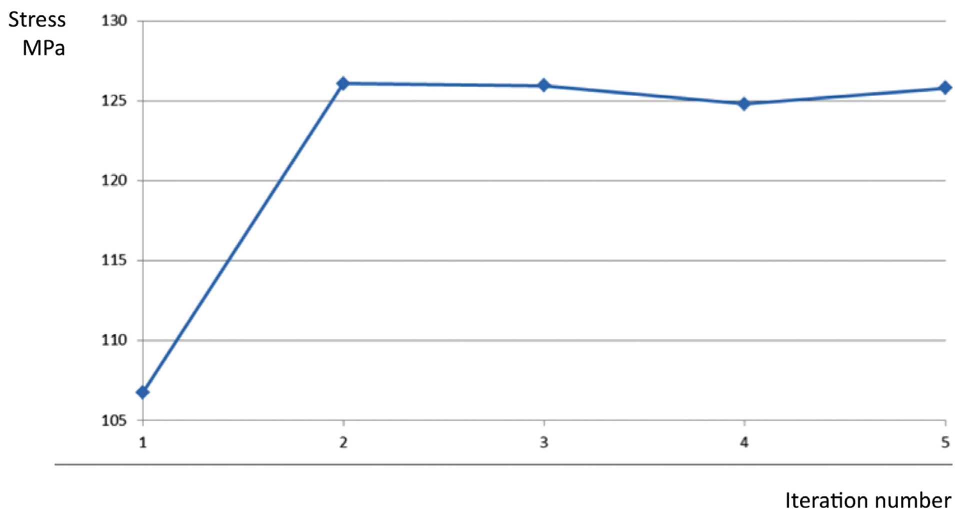


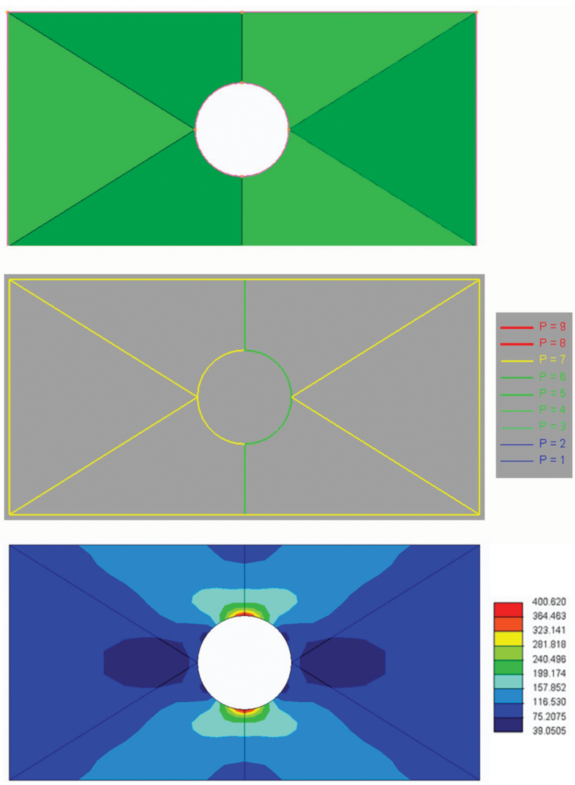
Figure 4.12 Convergence of the maximum von Mises stress in the *h* adaptive process; the maximum von Mises stress is shown as a function of the iteration number.

4.2.4 *p* Convergence Process

We have discussed three variations of the *h* convergence process: global, local, and *h* adaptive. In all cases, the element size is reduced, but the element order stays the same. Now, we will keep the same mesh, but we will upgrade the element order. Because the polynomial order of element displacement interpolation functions changes, this type of convergence process is called *p* convergence (*p* stands for polynomial). The *p* convergence process is possible only using elements capable of upgrading their order.

In a direct analogy to global mesh refinement (uniform refinement) and local mesh refinement (nonuniform refinement) used in the *h* convergence process, the *p* convergence process can also be uniform or nonuniform. In a uniform *p* convergence process, the order of all elements is upgraded until the desired accuracy is obtained. The advantage of uniform element upgrade is that mesh compatibility is automatically assured because the same displacement interpolation functions are used on all shared faces and shared edges of all elements. To assure mesh compatibility in a nonuniform (called adaptive) *p*-convergence process, different displacement interpolation functions must be used on different edges and faces of the same element; this is done to assure that displacements on the adjacent edges and faces of the neighboring elements are described by the same displacement interpolation functions. The face and edge order is "adapted" to the actual stress pattern and "quiet" portions of the model can be left at lower *p* orders for faster solution. To take a full advantage of *p* elements, a dedicated *p* element automesher is required. As a nonadaptive *p* convergence would not be numerically efficient, all commercial *p* element programs use *p* adaptive convergence.

The p element mesh shown in top illustration in Figure 4.13 is a result of discretization of a surface; it consists of only six shell elements: two triangular elements and four quadrilateral elements. It looks very different from the h element mesh. The model is subjected to a uniform tensile load applied to both vertical edges. The model is solved with program that can use element order anywhere between $p = 1$ and $p = 9$. In this case, the number between 1 and 9 refers to the order of stress interpolation function, not the displacement interpolation function. The final element order is found after a number of iterations during which the edge order is increased selectively based on errors found in the previous iteration; this happens in direct analogy to the h adaptive convergence process.



p-element mesh consists of six elements. The element shape may differ depending on the actual settings of the automesher.

The final edge order after five iterations. The highest order $p = 7$

Von Mises stress results produced in seven iteration of the p adaptive process. The maximum stress is 400.8 MPa.

Figure 4.13 The *p* element mesh (top), the final edge element order (middle), and the von MIses stress results found in a *p* adaptive solution process.

Figure 4.13 (middle) shows the final p order of element edges. Figure 4.13 (bottom) shows the von Mises stress plot; this result was produced in seven iterations. Given the accuracy requirements used in this solution, the solver did not have to reach the highest available p order and solution stopped at $p = 7$. The convergence of the maximum von Mises stress is shown in Figure 4.14, where the maximum von Mises stress is plotted for all seven iterations.

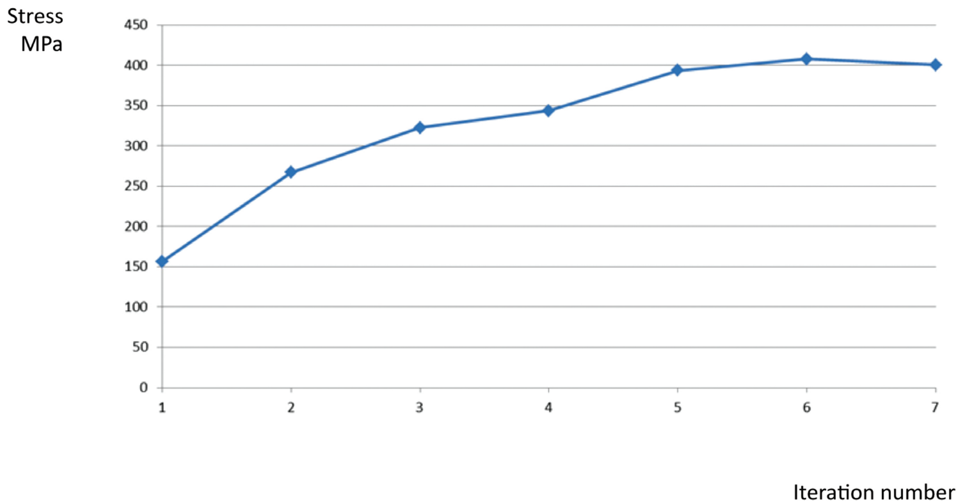


Figure 4.14 Convergence of the maximum von Mises stress in the *p* adaptive process; the maximum von Mises stress is shown as a function of the iteration number.

4.2.5 The Choice of Convergence Process

The majority of commercial FEM programs use *h* elements; they are native *h* element programs. For those programs, the *h* adaptive convergence, if available for the given type of analysis, offers advantages of an automatic detection of stress concentrations and mesh refinements adapted to the identified stress patterns. The users benefit from automatically generated convergence graphs of the data of interest. Some native *h* elements programs do support *p* elements even though those *p* elements are created using an *h* element automesher. This makes for a very useful learning tool that allows the users' familiarization with both *h* and *p* element technologies and leads to a full understanding of the convergence process. However, the best performance of native *h* element programs is usually achieved in the *h* element mode.

4.3 Discretization Error

All the graphs presented in this chapter show the data of interest (here displacements and stresses) converging to a finite value and indicate that discretization errors diminishes with the increase in the number of degrees of freedom in the model.

A simplified convergence curve shown in Figure 4.15 summarizes the results of the convergence process. For clarity of this illustration, only three steps are shown. This graph could have been produced by any type of convergence processes: *h* convergence global, *h* convergence local, *h* convergence adaptive, or *p* convergence adaptive. On the ordinate, there is the iteration number and on the abscissa, the selected data of interest, which is the convergence criterion. We will now present different ways of calculating the discretization errors in the FEM solution.

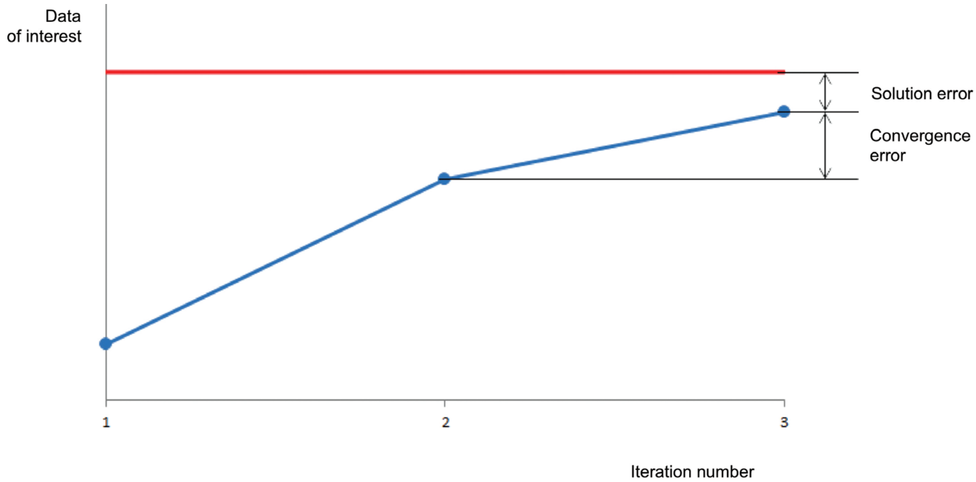


Figure 4.15 Definitions of convergence error and solution error.

4.3.1 Convergence Error

We can define convergence error as the difference in results between two consecutive mesh refinements. It is convenient to normalize the error to make it dimensionless. If we do that, the convergence error for iteration n is defined as

$$\text{Convergence error} = \left| \frac{\text{result}(n) - \text{result}(n-1)}{\text{result}(n)} \right|. \tag{4.1}$$

Figure 4.15 shows the convergence error of the last performed iteration, which is the third iteration.

Using the definition (4.1), the convergence error can be calculated for all steps of the convergence process except step 1. Convergence error for step 1 is unknown because no prior results exist. We can rephrase this important observation saying that a single run produces results with unknown discretization error.

4.3.2 Solution Error

With adding degrees of freedom to the model during the convergence process, we approach the exact solution of continuous mathematical. That unknown solution is the limit to which the data of interest converge. This asymptotic value may be used to define the discretization error another way; this will be called solution error. A solution error is the difference between the results produced by the FE model and the results that would be produced by a hypothetical FE model with an infinite number of infinitesimally small elements. This hypothetical FE model with an infinite number of elements would not be any different from continuous mathematical model:

$$\text{Solution error} = \left| \frac{\text{result}(n) - \text{asymptotic result}}{\text{result}(n)} \right| \tag{4.2}$$

where result(n) is the result of the last performed iteration.

As asymptotic solution is not known, we can only estimate the solution error. The convergence error and solution error are depicted in Figure 4.15.

4.4 Problems With Convergence

The finite-element solution converges to the exact solution of the mathematical model on which the FE model is based. Both *h* and *p* convergence processes demonstrated so far showed that the data of interest converge to finite values. But what happens if the solution fails to converge to a finite value?

4.4.1 Stress Singularity

Say we wish to find the maximum von Mises stress in a thin L-shape bracket. The bracket is modeled as a 2D plane stress problem and is meshed initially with large *h* elements as in Figure 4.16.

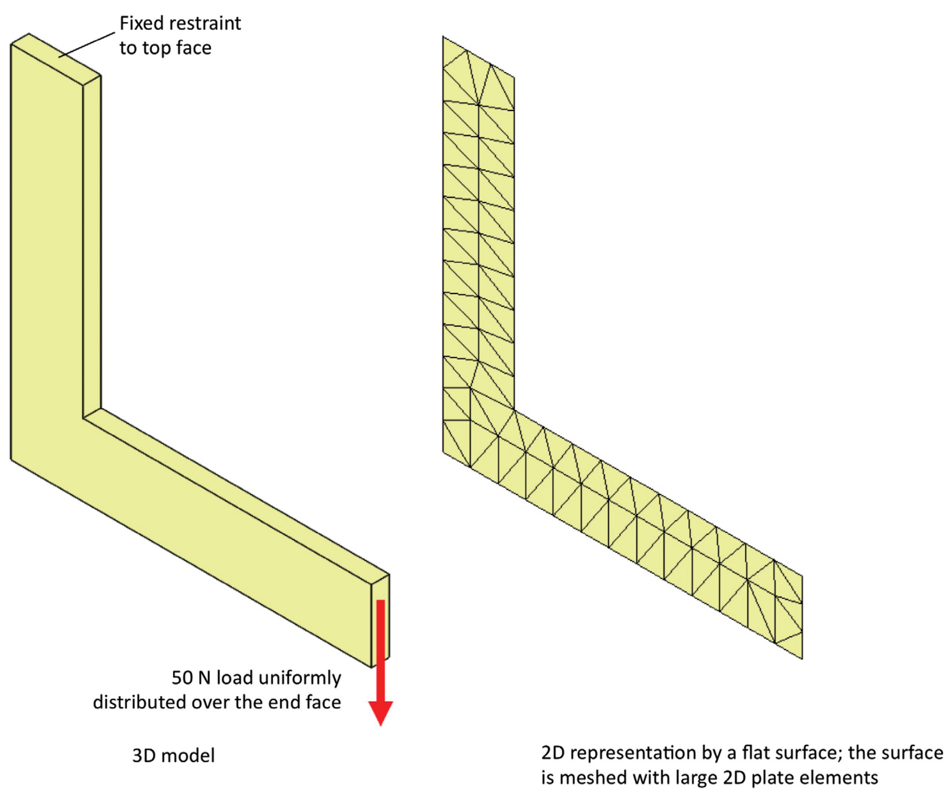


Figure 4.16 A thin L Bracket (left) may be represented by a flat surface and meshed with 2D plate elements (right). The mesh shown has an element size of 8 mm. Notice that 2D model may be used for stress analysis but not for buckling analysis.

We conduct the *h* convergence process by global mesh refinement starting with an element size of 8 mm as shown in Figure 4.16. The element size will be split in half in each iteration until it reaches 0.25 mm in the sixth and final iterations. A summary of the maximum von Mises stress results is shown in Figure 4.17. In all six iterations, the maximum stress is located in the sharp reentrant edge. The maximum stress 73.7 MPa (element size 0.25mm) is over 2.5 times higher than the yield strength of material (1060 aluminum alloy), which is 27.6 MPa. Figure 4.18 shows the results of the last performed iteration with an element size of 0.25 mm.

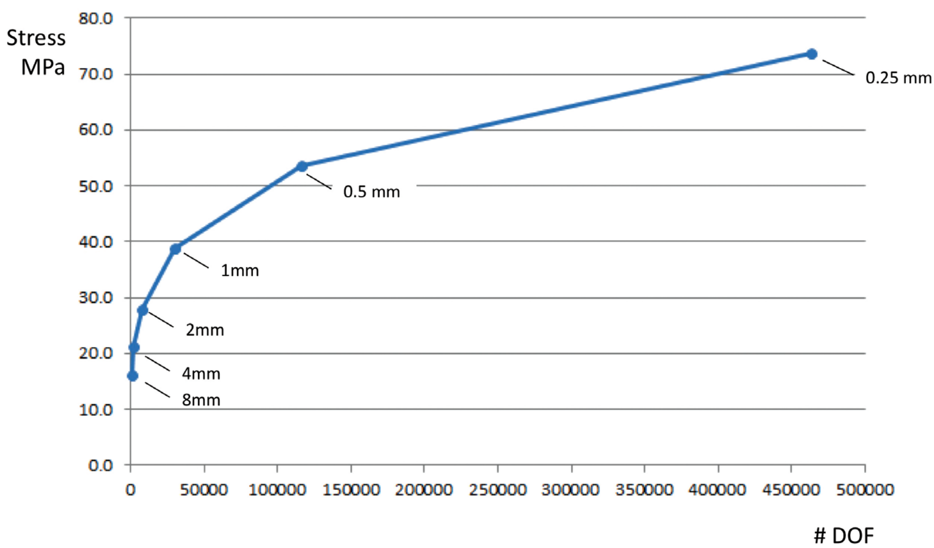


Figure 4.17 The maximum von Mises stress (located in the sharp reentrant edge) as a function of the number of degrees of freedom in the model with the element sizes of 8, 4, 2, 1, 0.5, 0.25 mm.

To find the accuracy of this result, we need to examine if the stress convergence curve converges to a finite value. However, a quick examination of the curve in Figure 4.17 reveals no sign of converging to a finite value; instead, each iteration brings ever higher stress results; the maximum stress diverges to infinity. The reason is that the mathematical model does not offer a solution for the maximum stress. The sharp reentrant corner constitutes a singularity where stress is infinite. As the objective of analysis is to find the maximum stress, we conclude that the FE model has been based on a wrong mathematical model. The mathematical model with stress singularity that coincides with the location of maximum stress cannot be used as a basis for the FE model if the analysis objective is to find the maximum stress.

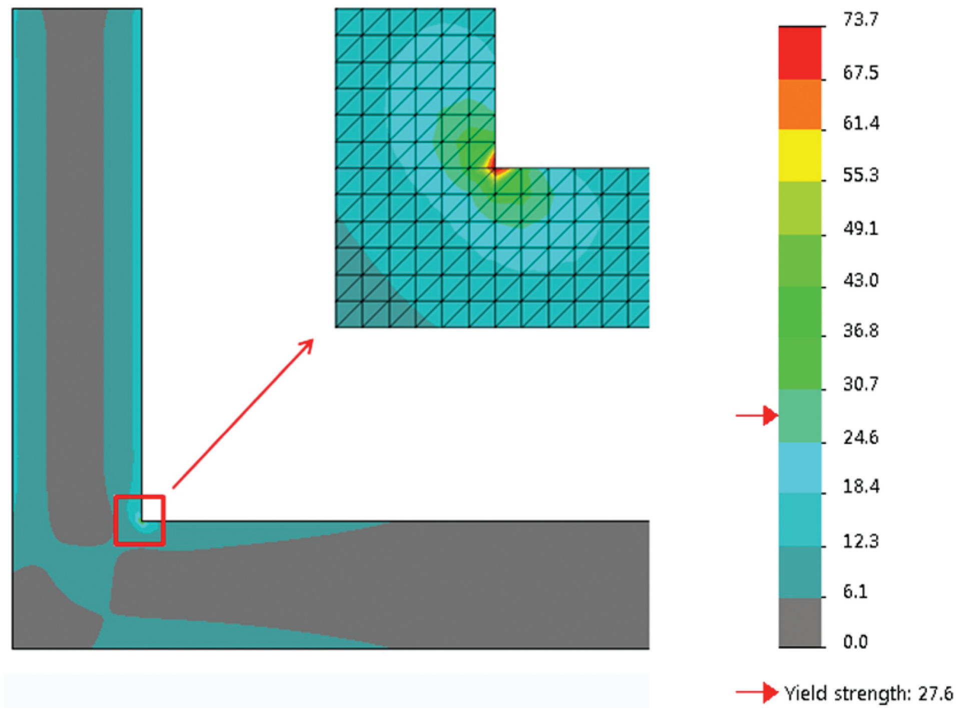


Figure 4.18 The maximum von Mises stress plot produced by mesh with an element size of 0.25 mm.

Why did the FE model produce a high but finite stress instead of infinite stress predicted by the mathematical model? This is because the modeling error (using wrong mathematical model) was masked by discretization error, which causes that finite stress is produced where mathematical model predicts infinite stress.

The stress singularity in the model in Figure 4.16 is caused by 270° sharp reentrant corner, but any reentrant corner (in 2D model) or a reentrant edge (in 3D model) produces stress singularity.

The strength of a singularity increases with the angle of the sharp reentrant edge. For example, a 270° edge causes a stronger singularity than a 225° edge and 340° causes stronger singularity than a 270° edge. Singularities manifest themselves as "hot spots"; the stronger the singularity, the easier it is to notice that "hot spot." To visualize singular stresses caused by the 225° edge, a more aggressive mesh refinement is required compared with the stress singularity caused by the 270° edge (Figure 4.19).

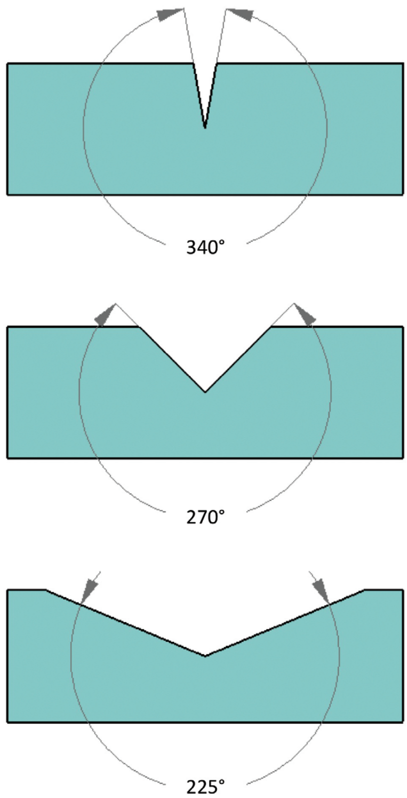


Figure 4.19 All sharp reentrant edges (or sharp reentrant corners in 2D) produce stress singularity. The closer is the angle to 360°, the stronger is the singularity. Comparing these three models, the singularity in the model with a 225° edge angle is the weakest and the singularity in the model with a 340° edge angle is the strongest. The strength of singularity indicates how fast stress diverges to infinity.

By showing the divergence of the data of interest (here the maximum von Mises stress), the convergence process revealed the modeling error. We can remedy this situation using a different mathematical model, one that does not have a stress singularity. The most obvious way is to model a fillet, which is always present in a real part even if the edge is very sharp. The result may show very high stress, but that result will be bounded; it will converge to a finite value.

Another way of eliminating stress singularity is to use material model capable of modeling plasticity. To illustrate this, we will use an elastic–perfectly plastic model characterized by a strain–stress curve in Figure 4.20. The elastic–perfectly plastic material model limits the value of the maximum von Mises stress to the plasticity limit; this eliminates stress singularity even though strain at the sharp reentrant corner continues diverging. The von Mises stress results using an elastic–perfectly plastic material and an element size of 0.5 mm is shown in Figure 4.21.

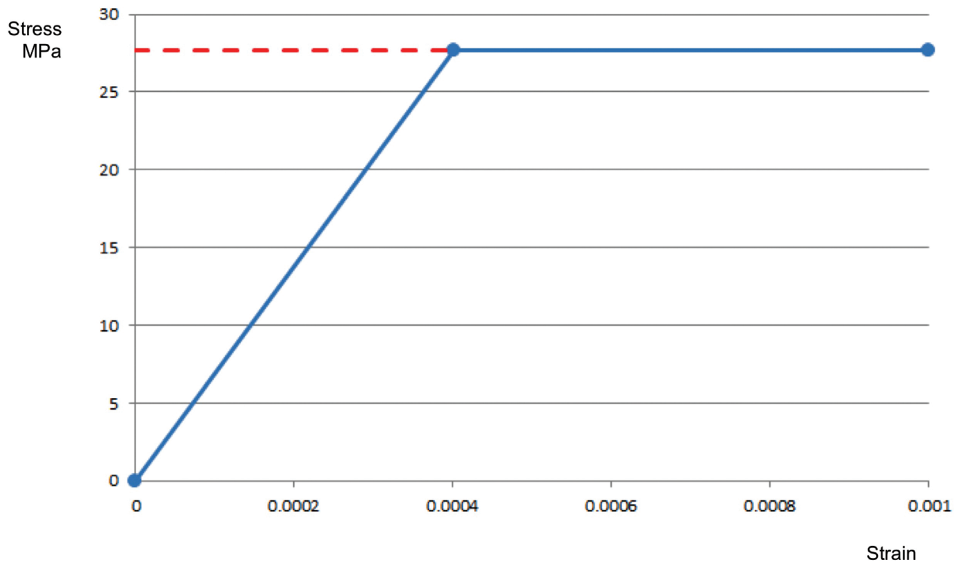


Figure 4.20 Elastic–perfectly plastic material, von Mises type; this illustration shows the strain–stress curve of 1060 aluminum alloy. The material behaves linearly until the von Mises stress reaches 27.6 MPa and then the modulus of elasticity becomes zero. Von Mises stress is used as a measure to control the switch of the modulus of elasticity to zero.

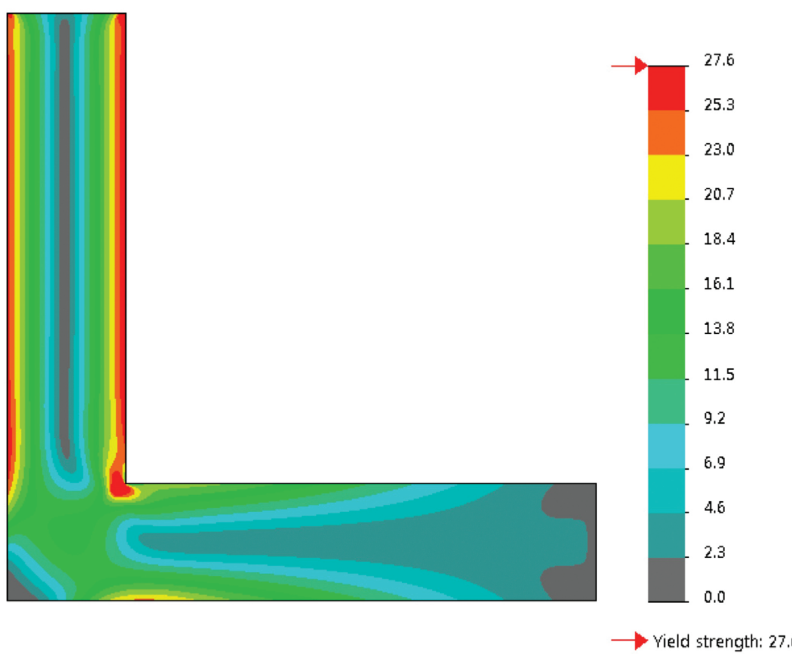


Figure 4.21 The von Mises stress solution using an elastic–perfectly plastic material and an element size of 0.5 mm. This illustration shows the results of step 14, which is the last step.

The solution shown in Figure 4.21 has been reached in 14 steps during which the load was being changed from 0 to 100%. The maximum von Mises stress for each step is shown in Figure 4.22.

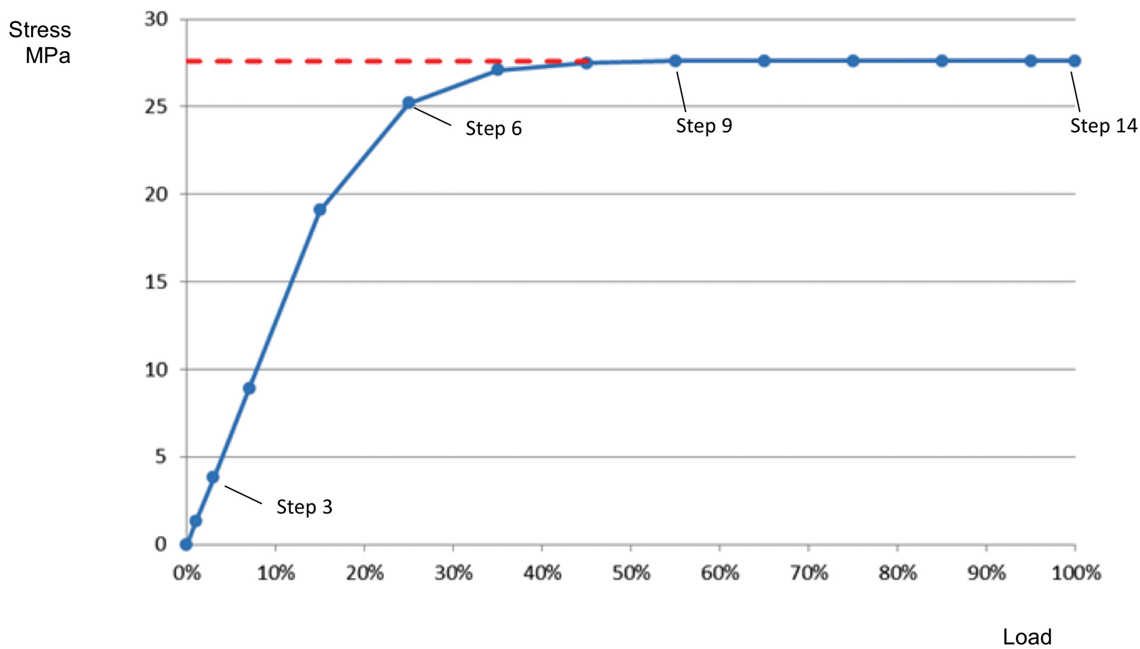


Figure 4.22 Changes of the maximum von Mises stress during 14 steps of the solution using elastic–perfectly plastic material and an element size of 0.5 mm.

Stress results in the vicinity of singularity are entirely dependent on the choice of discretization and, therefore, are meaningless. By manipulating the element size, element order or both, we can produce any stress result we want. Using the geometry with sharp reentrant corners in 2D models or sharp reentrant edges in 3D models, while the objective is finding stress in that location, is a severe yet common modeling error. An erroneous model is shown in Figure 4.23. This model can be used for displacement analysis because sharp reentrant edge does not pose displacement singularities. It can also be used for stress analysis in a location distant from sharp reentrant edges.

It is important to mention that fillets cannot be ignored even if stresses along the edge are not of interest, if removal of fillets changes model stiffness significantly.

If stresses along the edge are of interest, then fillets, no matter how small, must be modeled (Figure 4.24).

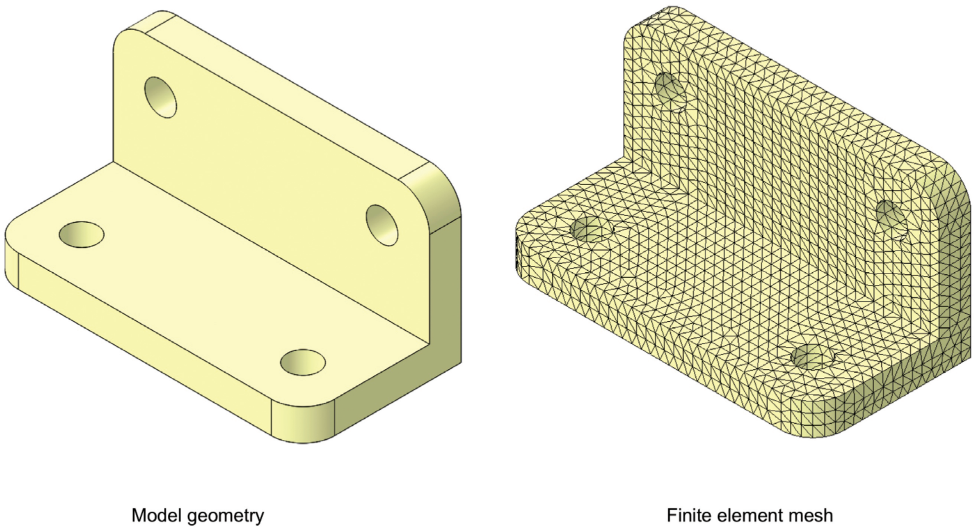


Figure 4.23 This model cannot be used for analysis of the maximum stress because those stresses (most likely) will coincide with the stress singularity caused by the sharp reentrant edge.

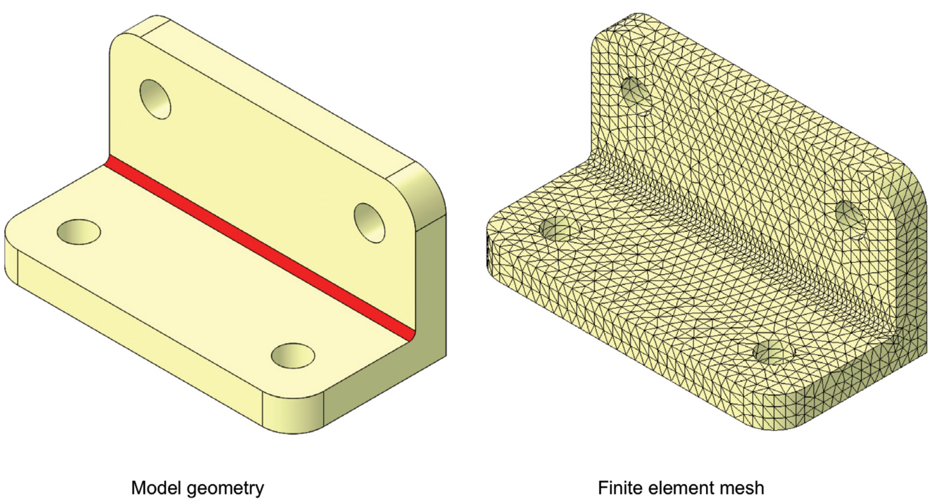


Figure 4.24 This model has a fillet (red) added in place of the former sharp reentrant edge. It can be used for analysis of the maximum stress. Stresses in the fillet may be very high, but during a convergence process, they will converge to a finite limit. Notice a mesh control defined on the fillet to produce correctly sized and correctly shaped elements.

4.4.2 Displacement Singularity

Now we wish to find displacements of a thin plate in bending supported by two spot welds (Figure 4.25). Model geometry and boundary conditions (restraints and loads) lend themselves to a 2D plane stress representation. Because the weld size is small

in comparison with the overall beam dimensions, we decide to model them as point supports; this is mistake, as we will soon find out. The stress results shown in Figure 4.26 indicate high stresses around supporting points. Divergence of the maximum stress reveals stress singularity as shown in Figure 4.27.

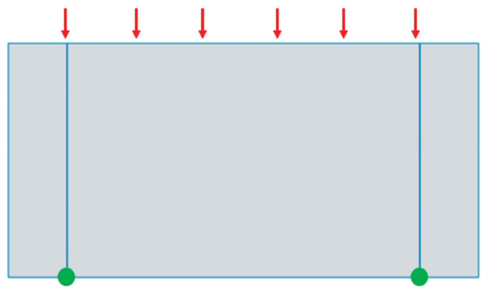


Figure 4.25 A thin plate supported in two points (green dots) and loaded along the top edge. This model lends itself to a 2D plane stress representation.

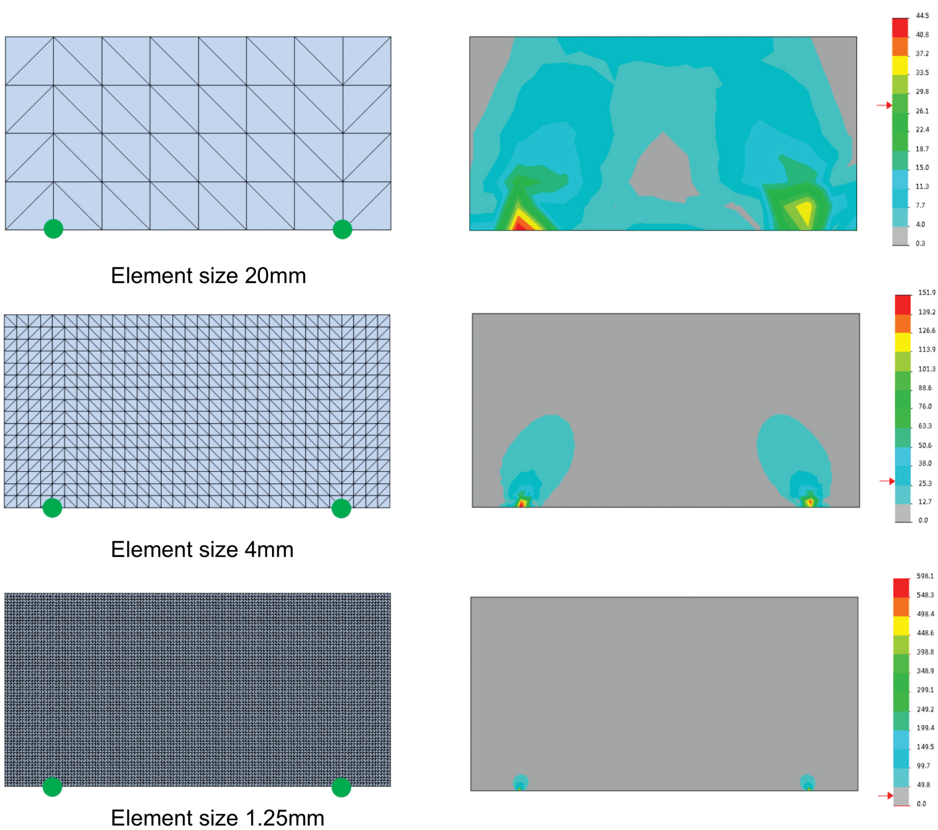


Figure 4.26 Three stress results in the global mesh refinement process produce diverging stress results caused by stress singularities at point supports. Undeformed stress plots are shown.

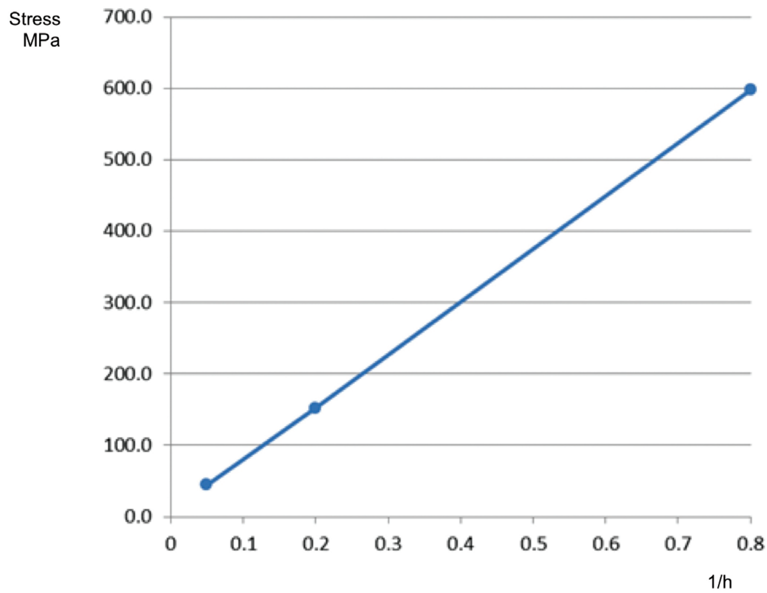


Figure 4.27 Summary of the maximum stress results from three iterations shown in Figure 4.26. The maximum von Mises stress is plotted as a function of $1/h$, where h is the characteristic element size. The stress at the point support diverges.

Can we still rely on this model to produce meaningful displacement results? To answer this question, we need to examine the convergence of displacements. The displacement results for three different meshes are shown in Figure 4.28 and are summarized in Figure 4.29. Examination of graph in Figure 4.29 proves that displacement results diverge.

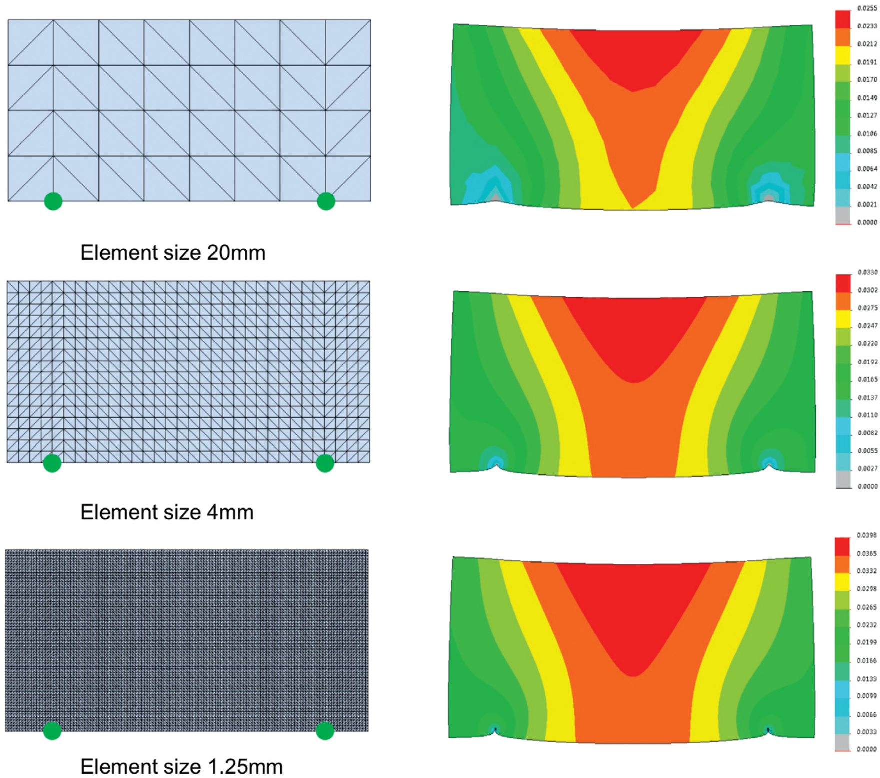


Figure 4.28 Three displacement results in the global mesh refinement process produce diverging displacement results caused by displacement singularities at point supports. Deformed displacement plots are shown.

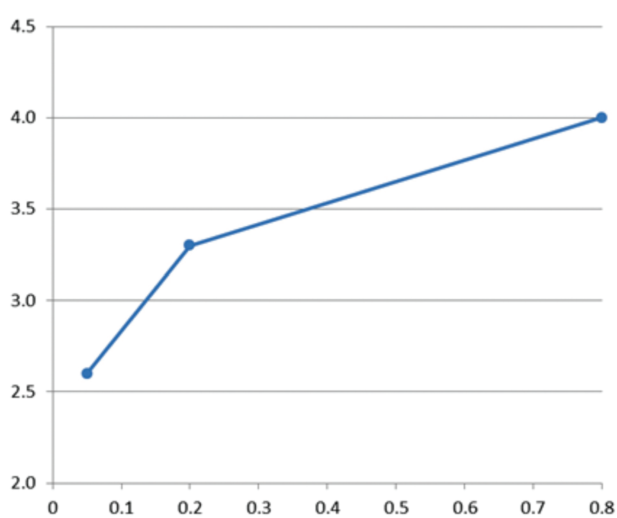


Figure 4.29 Summary of the maximum displacement results from three iterations shown in Figure 4.28. The maximum resultant displacement is plotted as a function of 1/*h*, where *h* is the characteristic element size. Displacement results diverge.

The maximum displacement tends to infinity with mesh refinement and finite displacement results reported by the FE model are because of discretization errors that conceal both displacement and stress singularities. Displacement and stress singularities are summarized in Table 4.2.

Table 4.2 Types of singularities encountered in finite-element models

	Type 1	Type 2
Stress	Infinite	Infinite
Strain energy	Finite	Infinite
Displacement	Finite	Infinite
Examples	Point load in 2D Line load in 3D Sharp reentrant corner in 2D Sharp reentrant edge in 3D	Point support in 2D Line support in 3D

Point support is a mathematical abstract and just like a sharp reentrant corner, it can never exist in a real structure. Point supports can be correctly used in FEA only if they are not expected to generate nonzero reactions like in the case of restricting rigid body motions (RBMs). Figure 4.30 illustrates eliminating RBMs for a 2D model, Figure 4.31 for a 3D solid element model and Figure 4.32 for a 3D shell element model. All these three cases prepare the plate model to be loaded with tensile load applied to both short edges without the use of fixed restraints applied to a face (3D model) or edge (2D model). This is often done to eliminate stress singularities created by fixed restraints also called rigid supports.

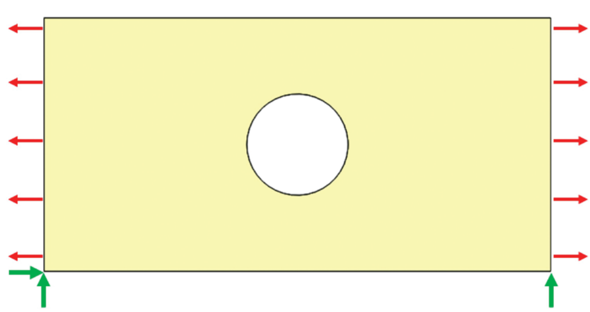


Figure 4.30 Point supports applied to two corners of this 2D model prevent RBMs but do not generate any reaction forces. Restraints shown in the lower left corner (green arrows) eliminate two translational degrees of freedom leaving the model free to rotate about this corner. Restraint eliminating one degree of freedom, shown in the lower right corner, eliminates that rotation. Load on one side may be replaced by restraint acting in horizontal direction only. 2D elements in this model can be plane stress or plane strain elements; they have 2 DOF/ per node; both are translations.

2D model in Figure 4.30 is loaded with tensile load applied to both short sides.

The following two 3D models, one using solid elements and the other one using shell elements, have restraints applied to one side and tensile load to the opposite side.

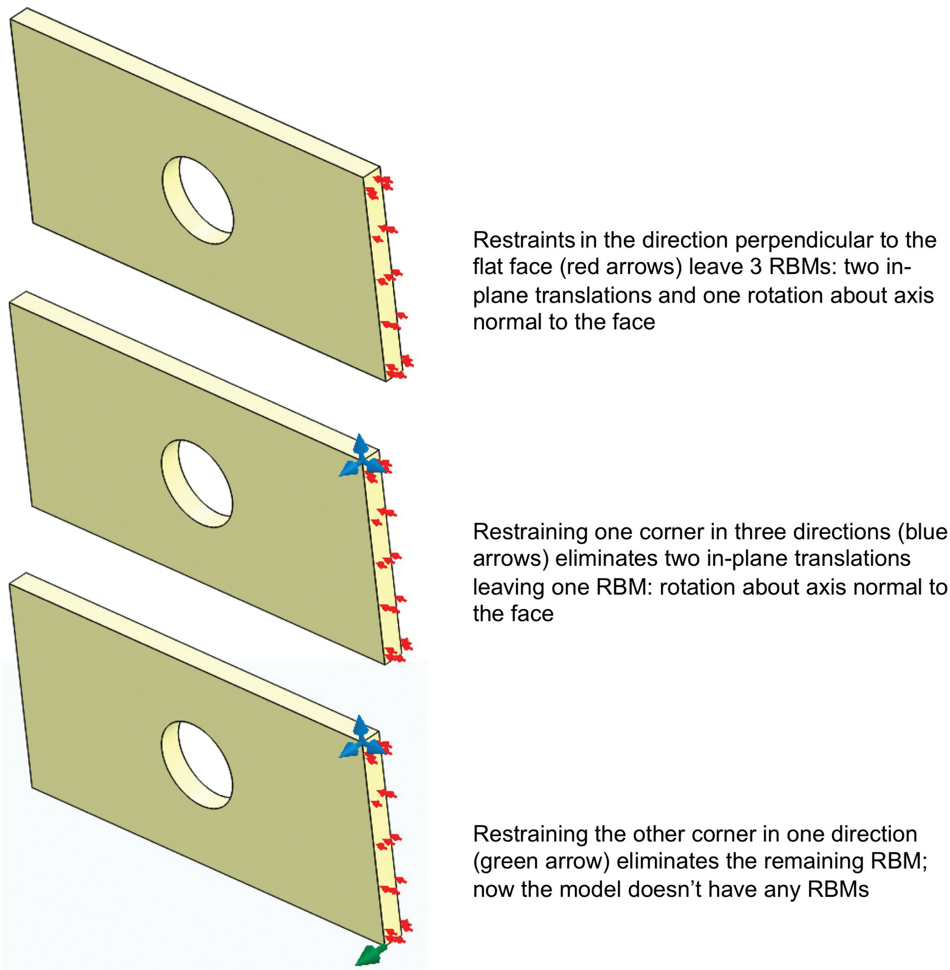


Figure 4.31 Point restraints used to eliminate RBMs in model meshed with solid elements. Solid elements have 3 DOF/ per node; all three are translations. Notice that applying fixed (rigid) restraints to the end-face would cause stress singularities in the four corners. Tensile load is applied to the opposite short face.

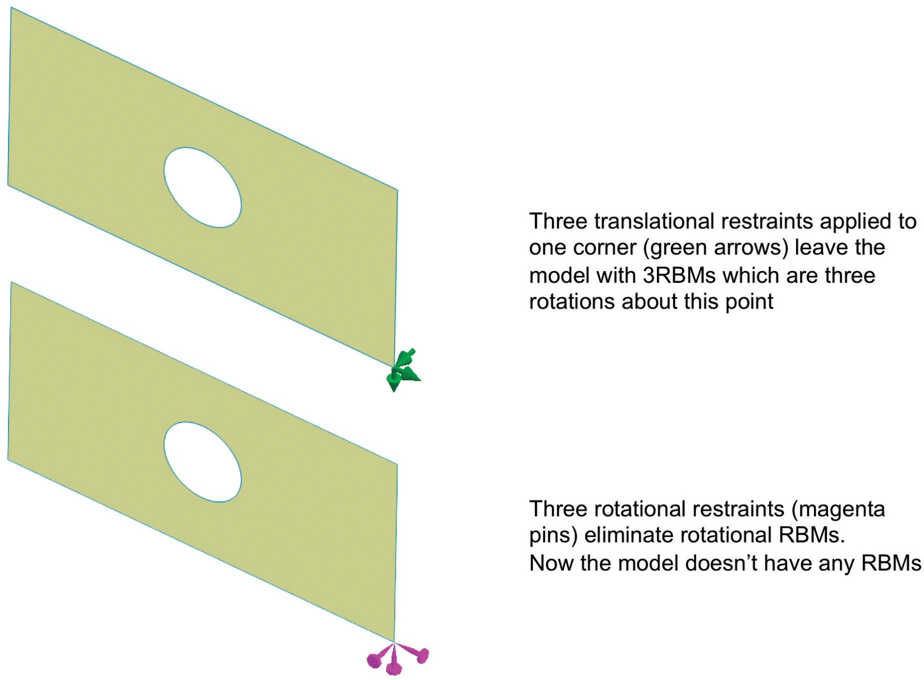


Figure 4.32 Point restraints used to eliminate RBMs in model meshed with shell elements. Shell elements have 6 DOF/ per node: three translations and three rotations. Tensile load is applied to the opposite short edge.

Singularities cannot be "fixed" using FEM methods. Singularities are modeling errors introduced by the formulation of mathematical model, and not by the finite-element approximation. Singularities, like other modeling errors, are introduced before the FEM enters the stage. Singularities can be revealed in convergence process, but no corrective action is possible unless mathematical model is changed.

The presence of singularity does not necessarily invalidate the FE model or make results incorrect, but we must be aware of the limitations imposed by the existence of singularity. For example, a model with sharp reentrant edges is incorrect if the analysis objective is to find the maximum stress or stress close to that edge. However, if the data of interest are displacements or modes of vibration, then the model may be used and acceptance of sharp reentrant edges in model geometry allows us to simplify the model. This is valid as long as the elimination of sharp reentrant edges from model geometry does not change stiffness significantly.

4.5 Hands-On Exercises

4.5.1 Hollow Plate (Figure 4.33)

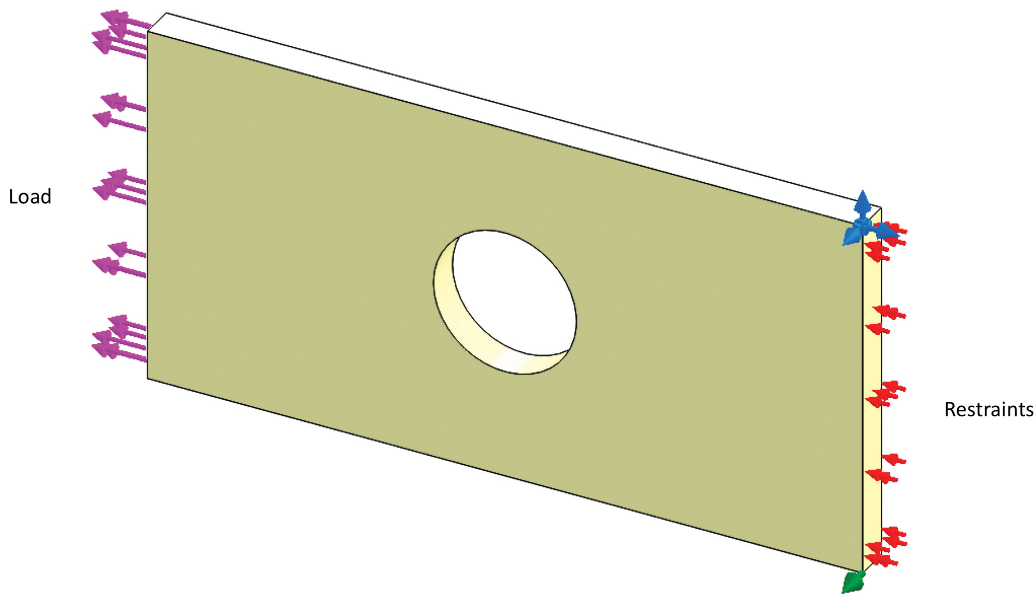


Figure 4.33 Hollow plate exercise. Restraints defined on the flat end face (red arrows) generate reaction forces and restraints defined on two corners (blue and green arrows) eliminate RBMs.

Model name

- 4.01.HOLLOW_PLATE.x_t
- 4.01.HOLLOW_PLATE.sldprt

4.5.1.1 Objective: Demonstrate the convergence of displacements and stresses using *h* and *p* convergence processes and first- and second-order elements.

4.5.1.2 Description: A plate with a circular hole is loaded with 100000-N tensile load uniformly distributed to one ends and supported as shown in Figure 4.33; restraints are identical to those shown in Figure 4.31. The geometry is suitable for meshing with solid elements. This exercise illustrates the *h* and *p* convergence processes and demonstrates how data of interest (maximum displacement in the direction of load and maximum von Mises stress) change with the element size during several steps of mesh refinement (*h* convergence) or several steps of element order upgrade (*p* convergence).

The following are the required steps:

1. Apply the material properties (steel).
2. Apply sufficient restraints to balance the load and eliminate RBMs but not to create stress singularities; follow the steps shown in Figure 4.31.
3. Apply a 100000-N uniformly distributed tensile load to the opposite face.
4. Mesh using first-order solid elements.
5. Obtain displacement and stress solutions.
6. Repeat steps 4 and 5 each time meshing with smaller elements, this is the *h* convergence process by global mesh refinement.
7. Plot the displacement and stress results as a function of the number of degrees of freedom in the model. Each curve will consist of three points corresponding to three steps in the mesh refinement process. If your software does not provide information of the number of degrees of freedom in the model, use the umber of nodes or number of elements or the inverse of the characteristic element size.
8. Repeat the exercise using second-order solid elements, and note that the results converge faster when using second-order elements.
9. Repeat the exercise using the *h* adaptive solution with second-order elements if your program supports this option; only one mesh will be required and this initial mesh may be a coarse one. The number of automatic steps will depend on the accuracy requirements specified in the analysis definition.
10. Repeat the exercise using *p*-type elements if your software supports this option; only one mesh will be required and this initial mesh may be a coarse one. The number of automatic steps will depend on the accuracy requirements specified in analysis definition.
11. Observe the convergence of displacement and stress in all solutions.
12. Delete all restraints and define a fixed restraint applied on the face opposite to where the load is applied. Notice that stress singularities are now present in the corners of the supported face. These singularities are difficult to spot when global mesh refinement is used because stress concentration becomes visible only when a very fine mesh is used. In this exercise, they may show in the *h* adaptive and *p* adaptive convergence processes.

4.5.2 L Bracket (Figure 4.34)

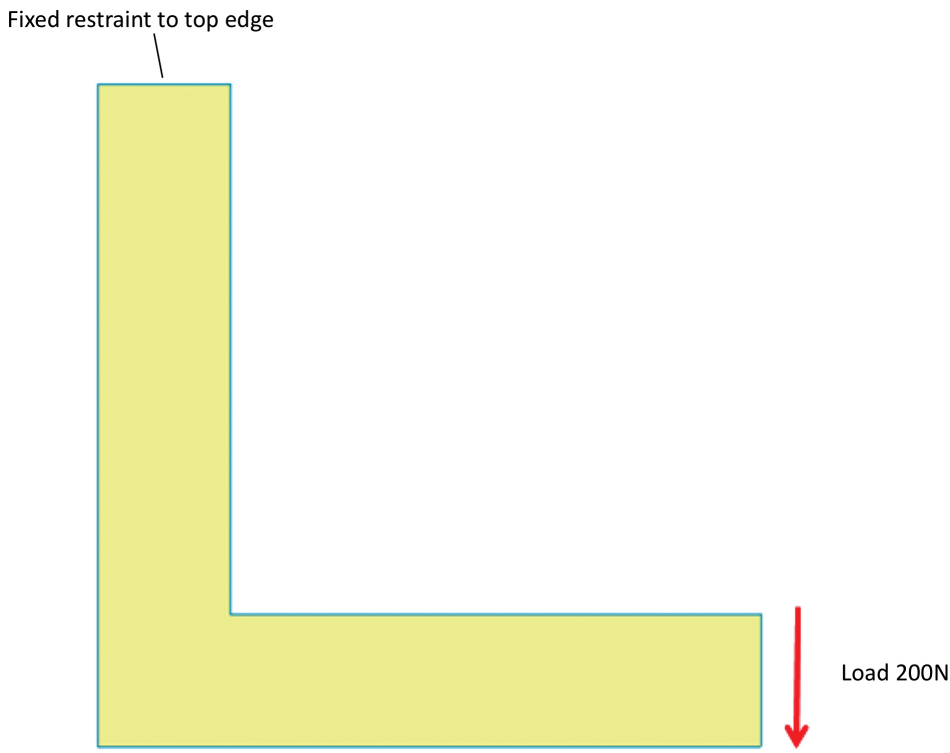


Figure 4.34 L bracket exercise. Restraints are defined on the top edge; the bending load is evenly distributed over the right vertical edge.

Model name

- 4.02.L_BRACKET.x_t
- 4.02.L_BRACKET.sldprt

4.5.2.1 Objective: Demonstrate the convergence of displacements and the divergence of stresses in the *h* and *p* convergence processes in the model with stress singularities.

4.5.2.2 Comment: The L shape bracket is represented by a surface intended for meshing with 2D plane stress elements or 3D shell elements. This exercise illustrates the lack of convergence of the maximum stress because of stress singularity in the sharp reentrant corner. The stress singularity makes the model useless if the objective is to find the maximum stress because the location of the maximum stress is coincident with stress singularity.

The following are the required steps:

1. Apply the material properties (1060 aluminum alloy).

2. Apply the shell properties (thickness 10mm).
3. Define a fixed support to the top edge.
4. Define a uniformly distributed bending load of 200 N, as shown in Figure 4.34.
5. Use second-order 2D plate elements or 3D shell elements.
6. Mesh with coarse, medium, and fine meshes.
7. Plot the displacement and stress results as a function of the number of degrees of freedom in the model. Each curve will consist of three points corresponding to three steps in the mesh refinement process.
8. Repeat the exercise using the *h* adaptive solution.
9. Repeat the exercise using the *p* adaptive solution.

4.5.3 2D Beam (Figure 4.35)

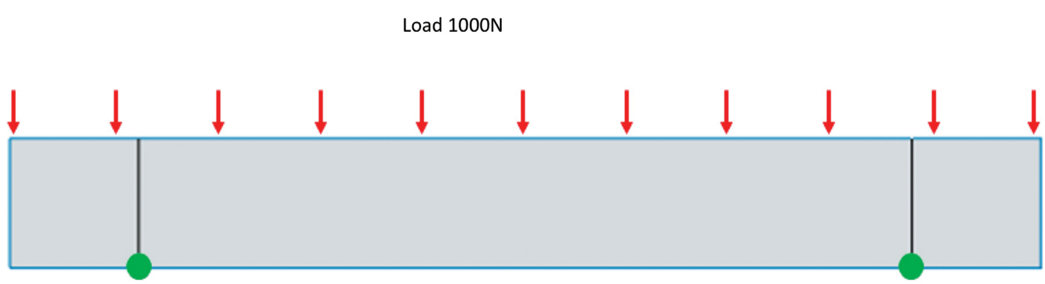


Figure 4.35 2D beam exercise; two point restraints (green dots); bending load (red arrows) evenly distributed over the top horizontal edge.

Model name

- 4.03.2D_BEAM.x_t
- 4.03.2D_BEAM.sldprt

4.5.3.1 Objective: Demonstrate the divergence of displacements and stresses in model with displacements and stress singularities.

4.5.3.2 Comment: A thin beam is supported by two spot welds. This model is intended for meshing with 2D plane stress elements or 3D shell elements. Because of the small size of spot welds, compared with the overall model size, somebody decided to model spot welds as point supports, which was a bad mistake as we soon will discover.

The following are the required steps:

1. Apply steel material properties.
2. Define thickness (10 mm).
3. Apply fixed supports to two points.

4. Apply the load of 1000 N as shown in Figure 4.35.
5. Use the second 2D plate elements.
6. Mesh with coarse, medium, and fine meshes decide on the actual element size.
7. Plot the displacement and stress results as a function of the number of degrees of freedom in the model. Each curve will consist of three points corresponding to three steps in the mesh refinement process.
8. Repeat the exercise using the *h* adaptive solution; only one mesh will be required (it may be the coarse one).
9. Repeat the exercise using the *p* adaptive solution; only one mesh will be required (it may be the coarse one).
10. Observe the divergence of displacements and stresses as demonstrated by the global *h* convergence, *h* adaptive convergence, and *p* adaptive convergence processes.

Chapter 5
Finite Element Mesh

We have already introduced several types of finite elements when discussing convergence process. Now we will focus on how are elements assembled into a mesh. We will review different methods of creating the mesh as well as the issues of mesh compatibility, mesh quality, and mesh adequacy.

5.1 Meshing Techniques

5.1.1 Manual Meshing

The finite element analysis (FEA) was used as a tool of engineering analysis long before CAD was introduced and before graphic user interfaces became available. In order to create a mesh, one had to input the coordinates of all nodes and then construct elements by defining element connectivity to selected nodes. Later, the manual meshing process was improved by rudimentary geometry creation capabilities so that positions of nodes could be defined with the help of underlying geometry. That eventually led to mapped meshing requiring user to define key points and lines and the number of nodes along lines connecting key points. Surfaces and volume defined by those key points and connecting curves could then be meshed automatically; this was called mapped meshing (Figure 5.1).

Manual meshing and mapped meshing were very time consuming and prone to errors. These techniques are now of historical importance only and a few commercial FEA programs still support manual and mapped meshing techniques.

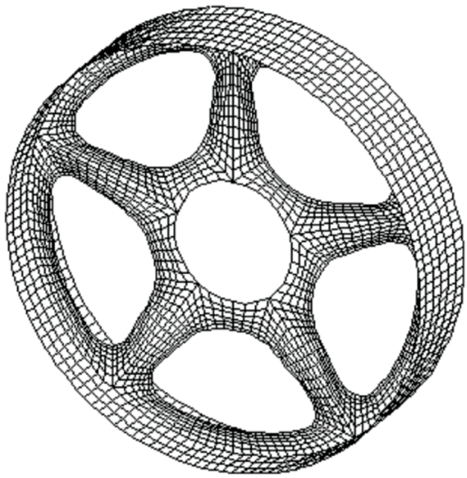


Figure 5.1 A sprocket model meshed with shell elements using mapped meshing technique.

5.1.2 Semiautomatic Meshing

Semiautomatic meshing techniques can be explained with concepts borrowed from CAD. Just like a solid feature can be created in CAD by extruding or rotating a flat sketch, a mesh can be created by extruding or revolving a planar surface. That surface needs to be meshed first with 2D elements and extrusion (Figure 5.2) or revolution (Figure 5.3) is done in several steps depending on how many layers of elements we need to create. Semiautomatic meshing is a powerful technique but, by its underlying principle, is limited to a narrow class of shapes. Just like manual meshing, it is now of historical importance only.

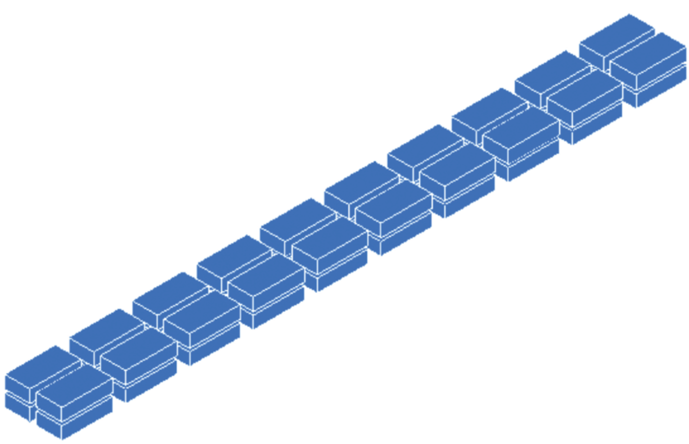


Figure 5.2 A mesh created with *p* elements by extruding a meshed flat surface in ten steps to create ten layers of elements. The mesh is shown with element shrinkage applied.

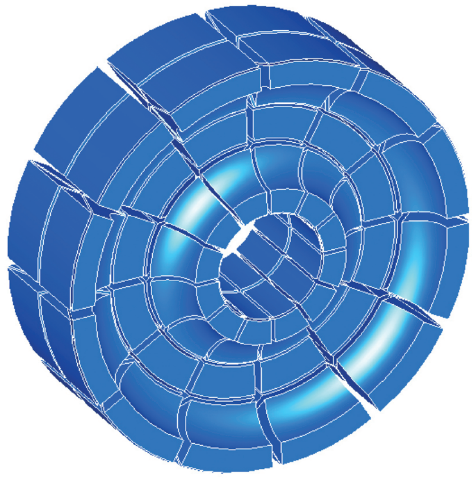


Figure 5.3 A mesh created with *p* elements by revolving the meshed radial cross section in 12 steps along 360° arc. The mesh is shown with element shrinkage applied.

5.1.3 Automeshing

Automatic meshing commonly called automeshing is usually the only meshing technique available in modern FEA programs. In addition, automatic meshing is the only practical meshing choice for complex models. The actual implementation of automeshing depends on particular FEA software. In some cases meshing is done in the background and is invisible to the user. More often, the FEA programs offer some control over the meshing process by allowing users to control characteristic element size (Figure 5.4) and mesh bias (Figures 5.5–5.7).

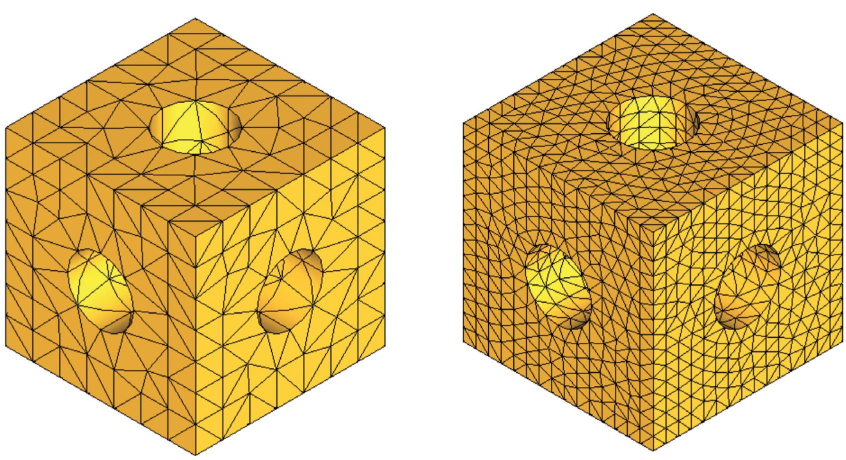


Figure 5.4 FE model meshed with coarse (left) and fine (right) elements. This is what we used in the convergence process by global mesh refinement. Note that second-order elements are used and they map precisely onto the cylindrical faces of all holes in both coarse and fine meshes.

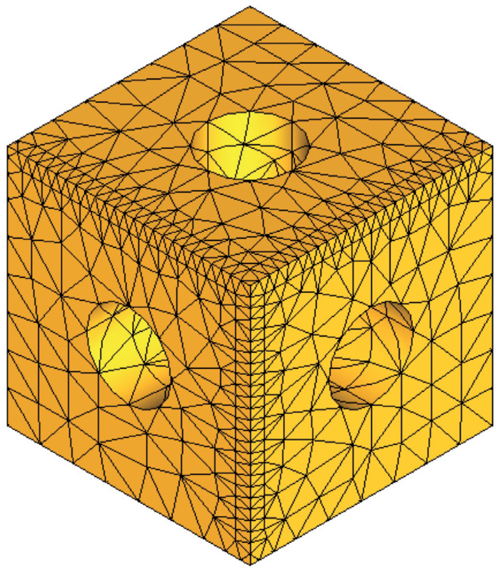


Figure 5.5 Mesh control (mesh bias) applied to selected edges produces smaller elements along these edges.

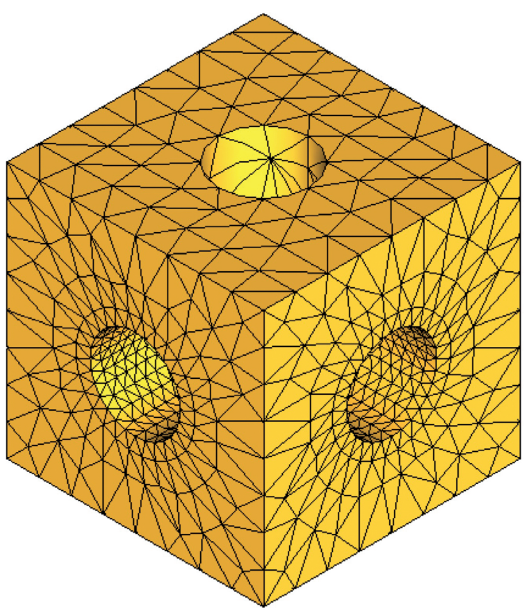


Figure 5.6 Mesh control applied to selected faces (two cylindrical holes) produces smaller elements on these faces.

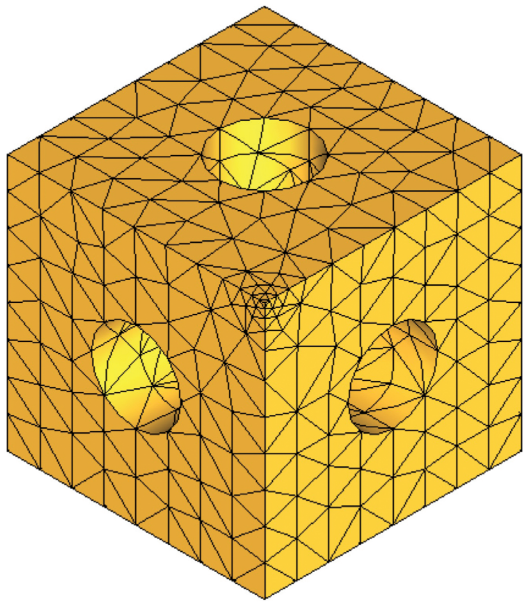


Figure 5.7 Mesh control applied to a vertex produces smaller elements around the selected corner.

The type of elements created by an automesher depends on what kind of geometry is submitted to meshing. Meshing volumes (solid CAD geometry) creates solid elements, meshing 3D surfaces creates shell, or membrane elements; meshing curves produces beams; meshing 2D surfaces (planes) creates 2D plate elements. Elements created by meshing different types of geometry are summarized in Table 5.1.

Table 5.1 Element type depends on what geometric entity is meshed. Note that what in CAD is called "solid" is called "volume" in FEA; volumes are meshed to create solid elements

Model dimensionality	Geometric entity meshed	Elements created
3D	Volume	Solid elements
	Surface	Shell elements Membrane elements
	Curve	Beam elements
2D	Flat surface	Plate elements

Hexahedral (brick) solid element mesh is difficult to create for automeshers; the same applies to a lesser degree to quadrilateral shell element meshes. For this reason, automeshers found in commercial FEA software are often limited to creating tetrahedral solid elements and triangular shell elements.

5.2 Mesh Compatibility

Sometimes, it is advantageous or necessary to combine different types of elements in one mesh. For example, "bulky" shapes may be meshed with solid elements and thin walls with shell elements. However, using different types of elements in the same mesh brings up the issue of mesh compatibility.

5.2.1 Compatible Elements

Two elements are compatible if they produce continuous displacement field across elements border. Depending on element type this border can be face (solid element), edge (3D shell and 2D plate), or point (beam element). Note that "continuous displacement field" means that all six displacement components (three translations and three rotations) are continuous. This requires that displacement interpolation functions describing displacements on neighboring element edges or faces are identical. If the elements are of the same type and they share corresponding nodes, this condition is automatically satisfied (Figure 5.8); otherwise, displacement is discontinuous across the border of connecting elements.

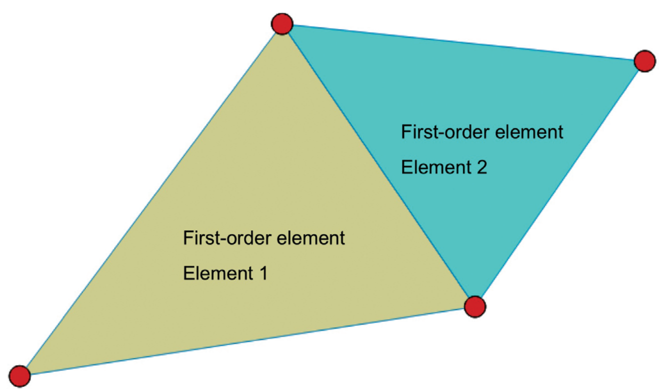


Figure 5.8 Two elements joined along an edge are shown as if they shared one edge, but these are two coincident edges. For two elements to be compatible, the displacement interpolation functions describing displacements of the edge belonging to element 1 and the displacement interpolation function describing displacement of the edge belonging to element 2 must be identical. This requirement is automatically satisfied if the elements are of the same type and order as the above two first-order triangular shell elements.

5.2.2 Incompatible Elements

If connecting elements are not of the same type and/or order or are not connected to corresponding nodes of the other element, then compatibility conditions are not automatically satisfied and mesh compatibility must be enforced by imposed compatibility requirements called links.

Elements shown in Figure 5.9 are not compatible. The displacement interpolation function along the edge of element 1 is of the second order, while the displacement

interpolation function along the edge of element 2 is of the first order. There is nothing "telling" the mid-side node on edge belonging to element 1 to follow the displacements of edge 2. As a result, a gap or an overlap will form between the two elements when they deform under load. Similarly, there is an incompatibility in mesh shown in Figure 5.10 even though the mesh consists of only one element type. A closer look at the transition between small and large elements reveals that the refined part of mesh is connected to the coarse part of mesh only by two nodes. This results in a gap forming under tensile load.

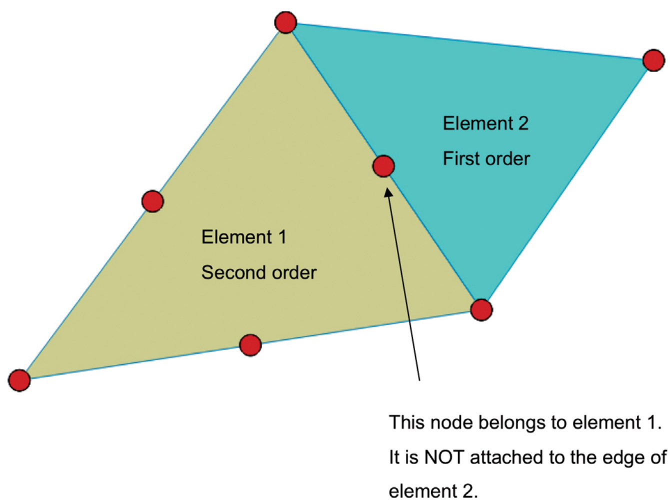


Figure 5.9 Because the mid-side node on the edge of element 1 is not attached to the edge of element 2, a gap or overlap will form when a load is applied. The mesh incompatibility occurs because of different element orders.

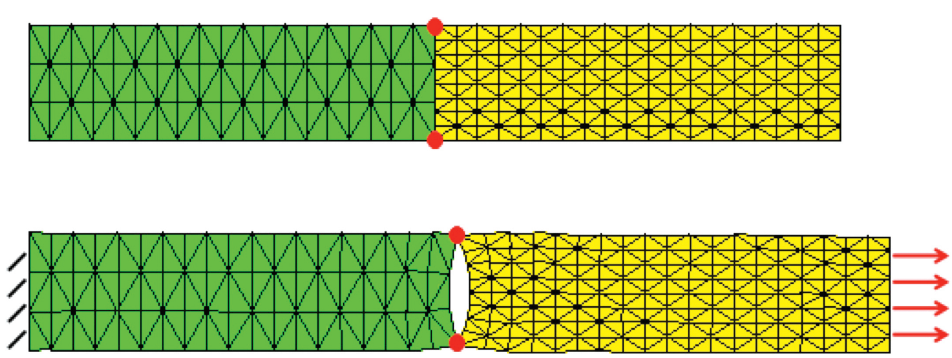


Figure 5.10 The left and the right sides of the mesh share only two nodes shown in red. This is not evident if the mesh is shown in the undeformed shape (top) but clearly shows under tensile load (bottom). This illustration was created in an old FEA program; it would take some effort to intentionally create this incorrect mesh using a modern automesher.

5.2.3 Forced Compatibility

The continuity of displacement field and thus the mesh compatibility may be enforced in an incompatible mesh. This is done by linking all displacement components of entities (points, edges, and faces) belonging to the first group of elements to the corresponding entities of the second group of elements as illustrated in Figures 5.11 and 5.12 (linking edges and linking faces, respectively). When displacements of two entities are linked, one entity becomes “master,” the other becomes “slave”. Displacements of the slave entity must then follow the corresponding displacements of the master entity.

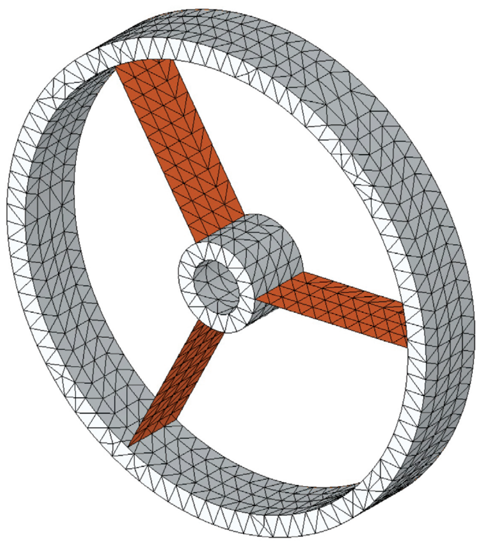


Figure 5.11 The spokes, meshed with shell elements, are connected to the hub and rim meshed with solid elements. The mesh is incompatible because the nodes of shell elements have 6 DOFs, while the nodes of solid elements have 3 DOFs. The rotational degrees of freedom along the edges of shell elements that connect to edges of solid elements are not restrained and unintentional hinges are created. Links must be defined to constrain translations and rotations of the edges of the shell element to translations of the corresponding edges of the solid elements. The edges of solid elements become “masters” and the edges of shell elements become “slaves”. Notice that links suppress rotations of the edges of the shell elements because rotations of the solid element are not defined. The rim and the hub are meshed with large elements for clarity of this illustration. An analysis would require much smaller elements.

While mesh compatibility enforced by linking does assure a continuous displacement field, linking is often referred to as a “quick fix” approach to meshing problems. Links should not be used in locations where accurate stress results need to be produced.

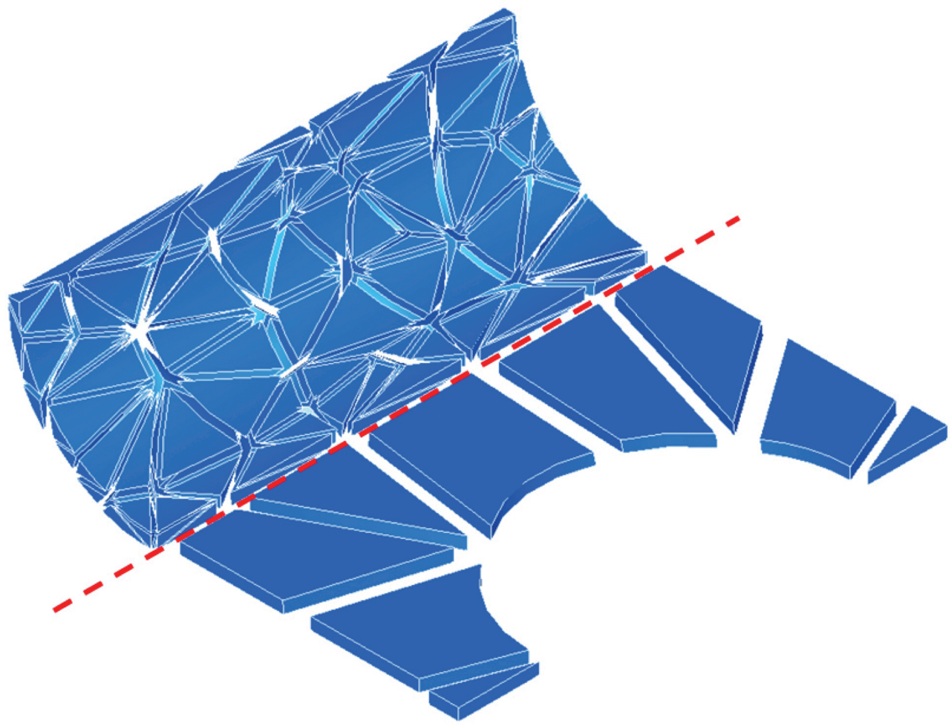


Figure 5.12 The flat portion of the model is meshed with solid hexahedral and pentahedral *p* elements; the curved one is meshed with solid tetrahedral *p* elements. The edges of tetrahedral elements running diagonally across faces of hexahedral and pentahedral elements are not connected to the corresponding faces of elements. To eliminate this incompatibility, links are required between the corresponding faces of tetrahedral and pentahedral and hexahedral elements. The location of links is indicated above by a dashed line.

5.3 Common Meshing Problems

New FEA users often expect that meshing should be a fully automated process requiring little, in any, input from the user. With experience comes a realization that meshing is not a "hands-off" task; it is often a difficult and time-consuming process; the user's input into meshing is often required. Leaving the automesher on the default settings may lead to severe meshing errors and this is why most FEA programs provide users with meshing controls.

5.3.1 Element Distortion

The shape of a finite element must be distorted from the ideal shape (e.g., equilateral triangle for shell element or cube for solid hexahedral element) when it is assembled in a mesh representing the actual model geometry. This happens during the meshing process when elements are mapped onto the model geometry. The distortion from the ideal shape results can take different forms are shown in Figures 5.13–5.16. Aspect

ratio and curvature distortion (high turn angle) are two most common types of element distortions.

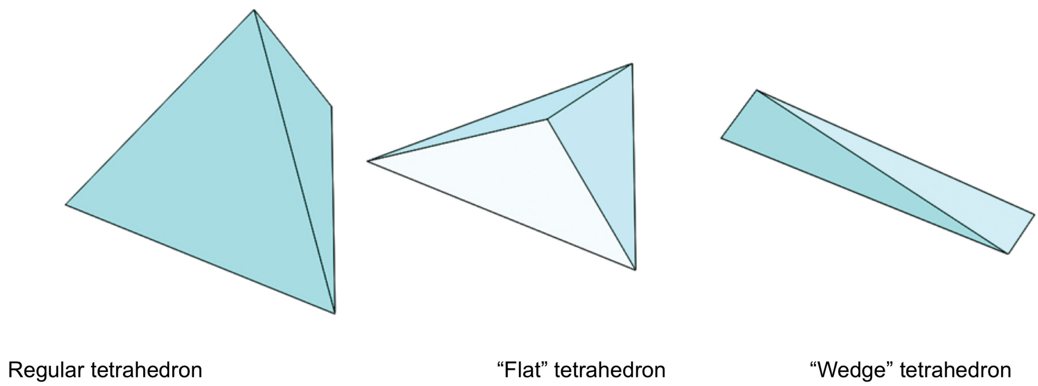


Figure 5.13 A solid tetrahedral element in its ideal shape (left) and two distorted tetrahedral elements with high aspect ratio (right).

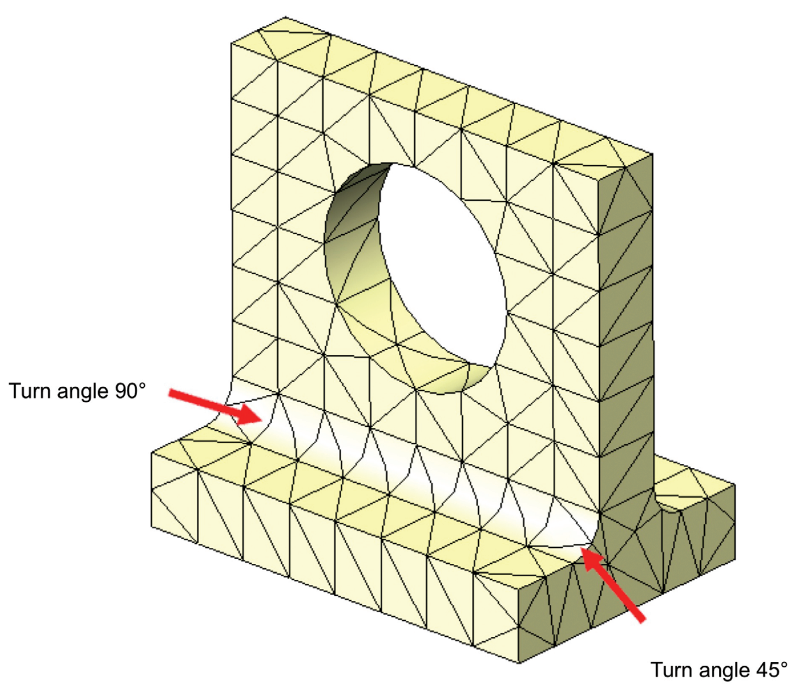


Figure 5.14 Elements in the fillet have high curvature distortion. Note that curvature distortion applies only to second order and higher; first-order elements cannot have curved edges.

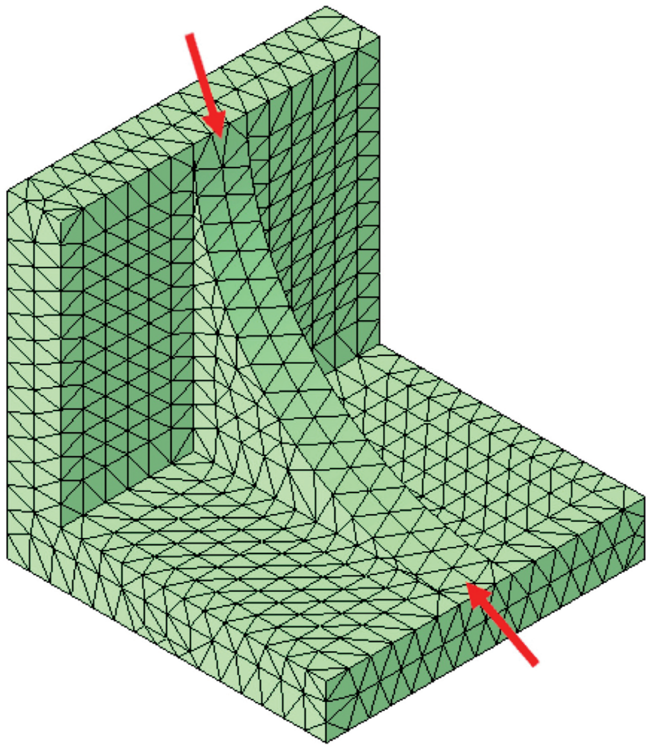


Figure 5.15 Elements with tangent edges as indicated by arrows.

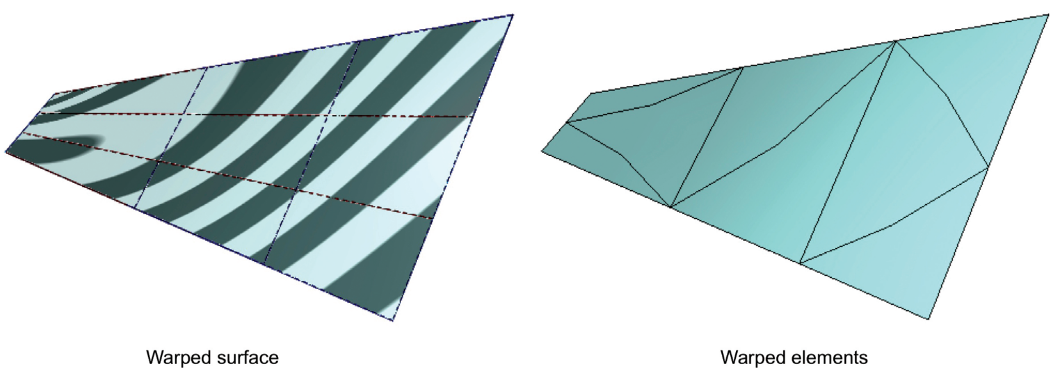


Figure 5.16 The warped surface shown with zebra stripes (left) and the warped shell element mesh (right).

Every element is designed to work properly within a certain range of shape distortion. Exactly how much distortion and what type of distortion is allowed before element degenerates depends on factors like element type and numerical procedures used in the

element design. Distorted elements can be detected using mesh quality tools available in most commercial FEA programs and with some training can be spotted "by eye".

While element distortion is easy to detect, it may be difficult to control. Recall that automeshing is basically a process of filling up a given volume or surface with certain geometric shapes, most often with triangles in 2D models and with tetrahedrals in 3D models. The automesher often finds it impossible to accomplish its task while keeping the elements within the allowable range of distortion and resorts to using excessively distorted elements. This is how element distortion enters the mesh.

5.3.2 Mesh Adequacy

A pure moment is applied to a beam; Figure 5.17 shows a model erroneously meshed with one layer of first-order elements across the beam thickness. Modeling a beam in bending with one layer of first-order elements does not capture the mechanics of bending as explained in Figure 5.18.

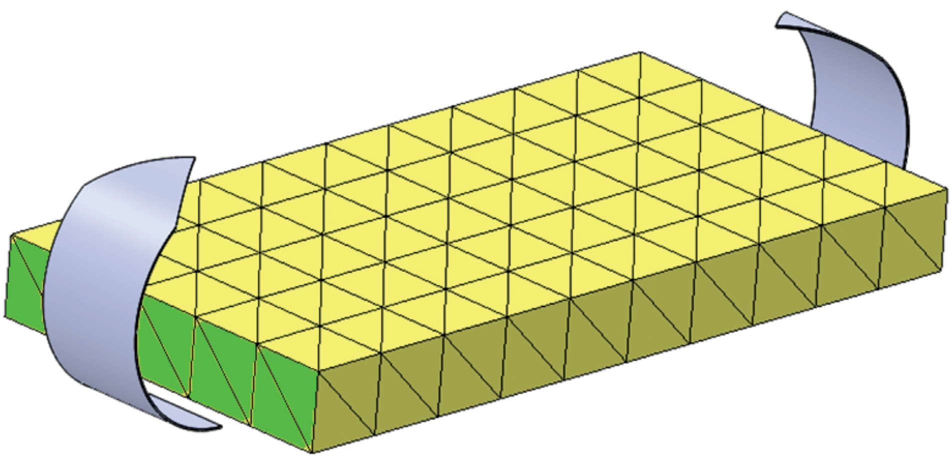


Figure 5.17 Beam loaded with a pure bending moment is meshed here with one layer of first-order element across thickness. The arrows indicate moment applied to two end faces; the visible end face is shown in green.

Modeling bending requires higher order elements. If only bending and not a more complex stress pattern needs to be modeled, then one layer of second-order elements suffices because it can represent linear distribution of bending stress across the thickness. However, it is generally recommended to use at least two layers of second-order elements in order to be able to capture more complex stress patterns. Solve some simple problems to establish how many layers of elements are required. Also, refer to your software manual for detailed recommendations. Thin features (e.g., thin walls) are well modeled with shell elements (Figure 5.19).

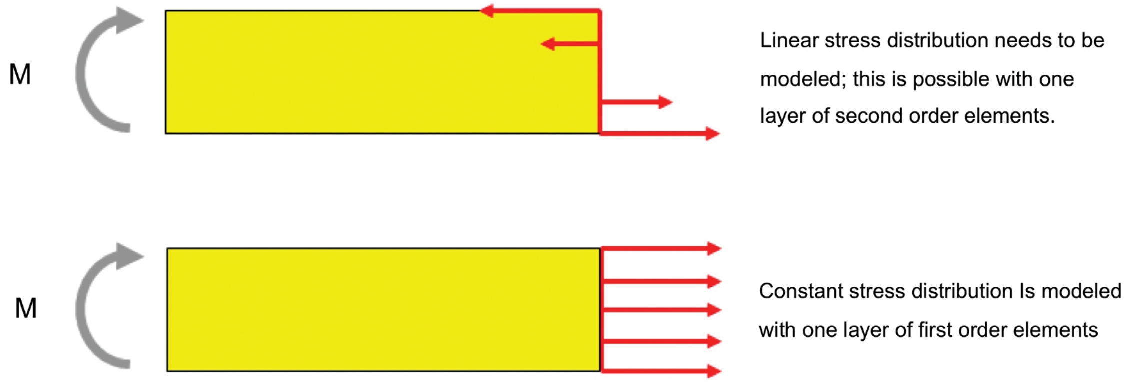


Figure 5.18 Pure bending produces a linear distribution of bending stresses across the beam thickness. The finite-element mesh must be capable of modeling this linear distribution, but this cannot be done with one layer of the first-order elements across the beam thickness. Using one layer of the first-order elements across the thickness models constant stress and does not capture the mechanics of bending. One layer of second-order elements, capable of modeling linear stress distribution, is required to model pure bending.

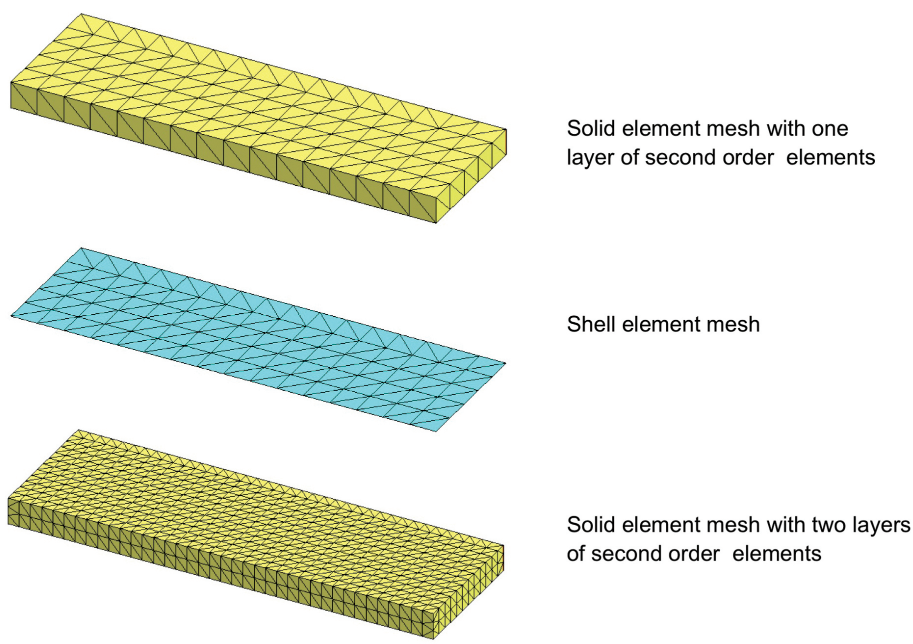


Figure 5.19 Beam in pure bending is correctly modeled with one layer of second-order elements (top). It can be also modeled with shell elements (middle) because shell elements model linear distribution of in-plane (bending) stresses. A more complex state of stress requires two (or more) layers of second-order elements.

5.3.3 Element Mapping to Geometry

The elements should be small enough to capture important details of geometry; too large elements "skip" geometry details as shown in Figure 5.20. Problems shown in Figure 5.20 are purposely exaggerated using large first-order elements, but models with similar mapping errors are often found in everyday practice. Meshing of even relatively simple geometry often requires a well-refined second-order element mesh (Figure 5.21). Notice that meshing gets easier when *p* elements are used. *p* elements map better onto model geometry because they are allowed higher distortion. Consequently, a fewer elements are required to mesh a model (Figure 5.22).

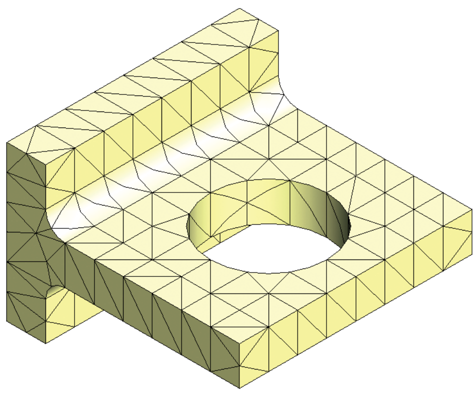


Figure 5.20 Large first-order elements are unable to map to geometry. Fillets are changed into chamfers and round hole into a polygon. The inability of the first-order elements to map onto curvilinear geometry combined with modeling constant stress make the first-order elements practically useless.

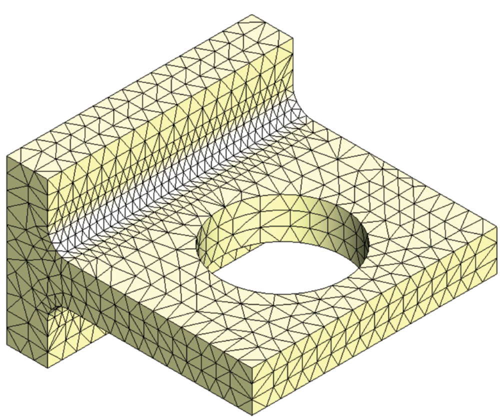


Figure 5.21 A second-order solid element mesh is required to model this bracket; the mesh is locally refined in the fillet area.

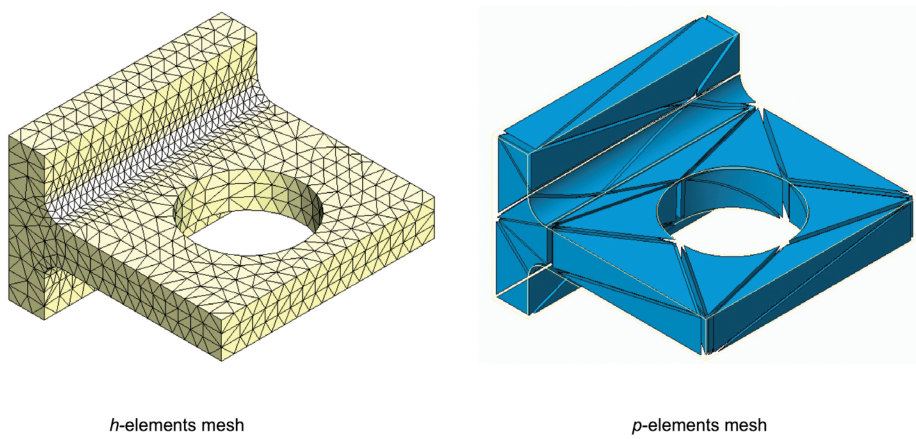


Figure 5.22 The same model meshed with 12980 second-order solid *h* elements (left) can be meshed with only 60 *p* elements (right). A fewer *p* elements are required because these elements are designed to reach higher order during solution; no local mesh refinement is required in the *p* element mesh. The *p* element mesh is shown with element shrinkage applied.

5.3.4 Incorrect Conversion to Shell Model

When representing thin wall models with shell element mesh, it is necessary to convert the geometry into mid-plane surfaces for meshing with shell elements. Even for simple geometry collapsing thin solids into mid plane surfaces often produces unattached surfaces and, consequently, unintentional gaps in the shell element mesh. These gaps are easily visible in simple models (Figure 5.23), but may be missed if more complicated geometry is analyzed. Conversion of solid geometry to surface geometry suitable for meshing with shell elements is a tedious process. Often, it is better to construct FEA-specific surface geometry from scratch.

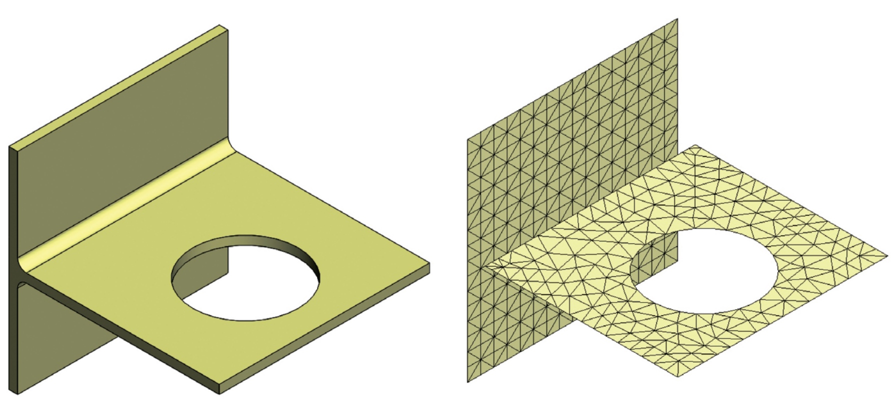


Figure 5.23 Automated processing of thin solid geometry (left) to create surface geometry suitable for shell element meshing may produce disjoint mid-plane surfaces useless for meshing with shell elements (right).

5.4 Hands-On Exercises

5.4.1 BRACKET01 (Figure 5.24)

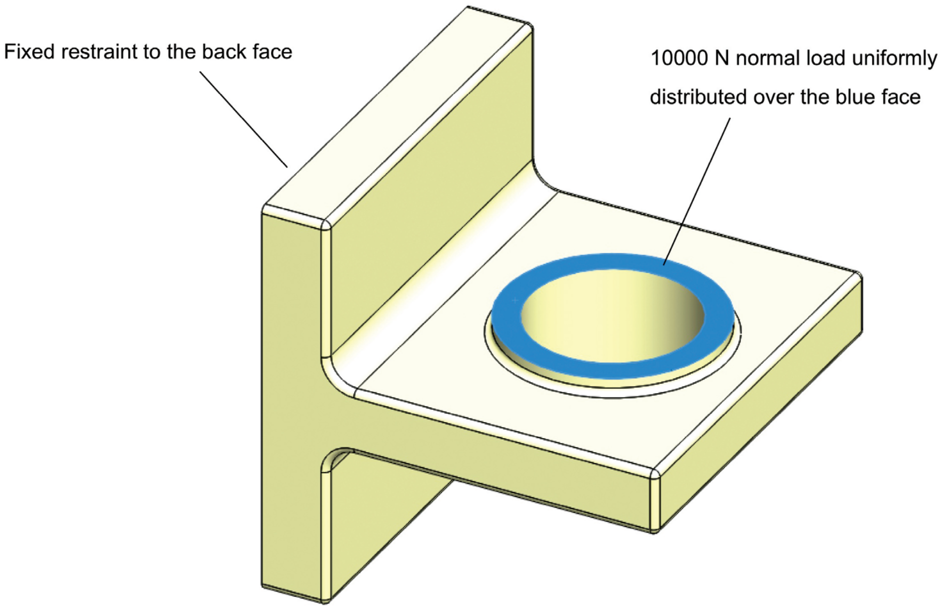


Figure 5.24 BRACKET01 exercise.

Model name

- 5.01.BRACKET01.x_t
- 5.01.BRACKET01.sldprt

5.4.1.1 Objective: Demonstrate difficulties caused by using first-order elements; demonstrate difficulties with meshing small geometric features.

5.4.1.2 Comment: The hollow cantilever bracket has fixed support at the back and is loaded with a 10000-N bending load uniformly distributed over the blue face (Figure 5.24). The objective is to find the location of the maximum von Mises stress.

The following are the required steps:

1. Apply material properties (steel).
2. Apply support to the back face.
3. Apply uniformly distributed 10000 N as indicated in Figure 5.24.
4. Using first-order tetrahedral solid element mesh with coarse, medium, and highly refined meshes, observe the quality of mapping to model geometry.
5. Plot maximum von Mises stress results as a function of the number of degrees of freedom in the model.

6. Observe the change in location of the maximum von Mises stress as produced by increasingly refined meshes.
7. Repeat the exercise using second-order elements as produced by increasingly refined meshes.
8. Repeat the exercise using the *h* adaptive solution; notice that the mesh refinement process is not necessary when the *h* adaptive method is used.
9. Repeat the exercise using *p* elements; notice that mesh refinement process is not necessary when the *p* element method is used.
10. Repeat the exercise after removing small outside rounds that have no structural significance; compare mesh complexity before and after the removal of rounds. DO NOT remove the big round which is structurally important.

5.4.2 Cantilever Beam (Figure 5.25)

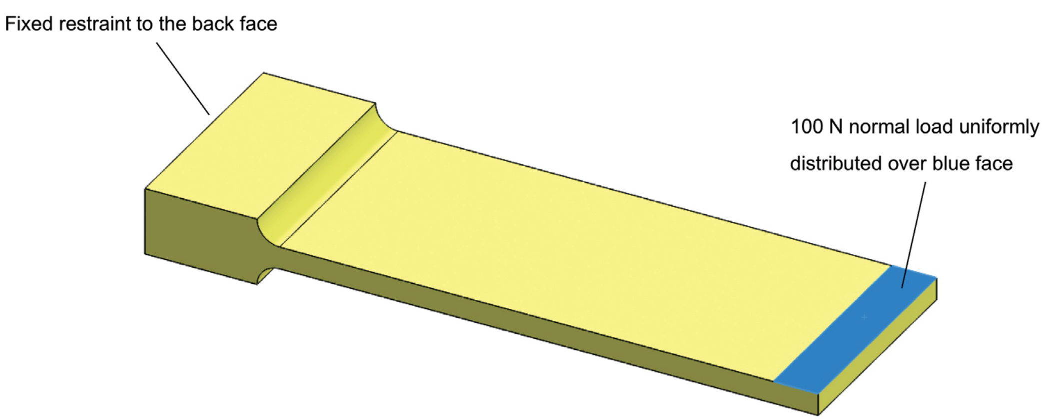


Figure 5.25 Cantilever beam exercise.

Model name

- 5.02.CANTILEVER_BEAM.x_t
- 5.02.CANTILEVER_BEAM.sldprt

5.4.2.1 Objective: Demonstrate the effect of an inadequate mesh on displacement and stress results.

5.4.2.2 Comment: The beam is rigidly supported and loaded with 1000 N of bending load uniformly distributed on the free end. The objective is to find the maximum displacement and the maximum von Mises stress.

The following are the required steps:

1. Apply material properties (steel).

2. Apply loads and restraints.
3. Mesh with first-order tetrahedral solid elements.
4. Adjust mesh density so that beam is meshed with only one layer of elements. This purposely creates an erroneous mesh.
5. Record the erroneous results for the maximum displacement and the maximum von Mises stresses and compare with the analytical result.
6. Repeat the exercise with second-order elements and/or *p* elements.
7. Compare the results with the analytical results and with those produced using first-order elements.

Chapter 6
Modeling Process

In just about any FEA, most effort is spent creating the mathematical model that is the basis for the finite element (FE) model. When the mathematical model is ready, it needs to be discretized to turn it into FE model. The discretization process, commonly called meshing, is the only FE specific task in the process of FE model creation.

We discuss issues related to the mathematical model having just finished discussing mesh-related issues. This order is reversed to how they appear in the FE modeling process. We do that because we are now better prepared to appreciate the differences among the CAD model, the mathematical model, and the FE model. We can better appreciate that most topics typically discussed in FEA textbooks or FEA training courses are not specific to the FEA. Instead, they deal with mechanics of materials and with techniques used for creation of the mathematical model. Decisions on the type of analysis (linear or nonlinear), on the dimensionality of analysis (2D or 3D), idealization, and defeaturing of the CAD model, applying loads, restraints, and so forth all belong to the process of creating a mathematical model and not an FE model. Still, in everyday FEA practice, the tasks of creating a mathematical model and the task of creating a FE model are tightly knitted together. This justifies our emphasis on mathematical models and modeling techniques used in their creation.

No matter how much time and effort we spend on modeling, a mathematical model can never be accepted as a final and true description of the system. Rather, it can at best be regarded as a good enough description of certain aspects of the system that are of particular interest to us. As the ultimate objective is to make a design decision, a mathematical model should only be good enough to allow making that decision with a reasonable confidence. From this point of view, the best mathematical model is the simplest one that still provides desired results. We will review typical steps in creating the mathematical

model and offer modeling tips helpful in creating models that provide the data of interest without unnecessary complexity.

6.1 Modeling Steps

Many FEA users start an FEA project by creating model geometry suitable for analysis. While this approach is common and seems natural, it is not necessarily correct. There are lots of things to do before working on the geometry.

6.1.1 Definition of the Objective of Analysis

This is, of course, the most fundamental issue. A model intended for modal analysis may be very different from the one intended for stress analysis. We need to know what we wish to learn from results and then construct the simplest possible model that will still provide reliable data of interest. This applies to all stages of the modeling process. For example: do not use a nonlinear material if linear model will do; do not use time-dependent loading if static load will do just fine; do not used fully featured geometry if all you need to find are global displacements and so forth.

6.1.2 Selection of the Units of Measurement

We can use any consistent system of units for FE model, but, in practice, the choice of the system of units is dictated by what units are used by CAD model. Trouble is that the system of units in CAD is not always consistent; for example length can be expressed in millimeters, while mass density can be expressed in kilogram per cubic meter. Contrary to CAD, in FEA, all units must be consistent. Inconsistencies, which are easy to overlook, may lead to serious errors.

In the SI system, based on meter [m], kilogram mass [kg], and second [s], all other units are easily derived from these three basic units. The situation gets more complicated if we use a system based on derived units. In mechanical engineering, length is commonly expressed in millimeters [mm], force in newton [N], and time in seconds [s]. All other units must then be derived from basic units: millimeter, newton, and second. Consequently, the unit of mass is a mass, which when subjected to the unit force of 1 N will accelerate with the unit acceleration of 1 mm/s^2. Therefore, the unit of mass, in the system using millimeter, newton, and second, is equivalent to 1000 kg or to 1 metric ton. Mass density is therefore expressed in tons per cubic millimeter. This is critically important to remember when defining material properties used in FE model as well as when assigning mass properties to mass elements. For example, the element with the mass of 5 kg must be assigned the mass of 0.005 ton. Notice that the erroneous definition of mass density in kilograms per cubic meter rather than in tons per cubic millimeter will result in mass density being one trillion (10^{12}) times higher! The same applies to other derived systems of units as shown in Table 6.1. Many commercial FEA programs offer unit managers where data may be entered and results be displayed in any units.

Table 6.1 Mass density of aluminum in the three systems of units. Comparison of numerical values of mass densities of 1060 aluminum alloy defined in the SI system of units with the system of units derived from SI and with the English (IPS) system of units

System SI	[m] [N] [s]
Unit of mass	kg
Unit of mass density	kg/m^3
Density of aluminum	2794 kg/m^3
System of units derived from SI	[mm] [N] [s]
Unit of mass	tonne
Unit of mass density	$tonne/mm^3$
Density of aluminum	2.794×10^{-9} ton/mm^3
English system (IPS)	[in] [lbf] [s]
Unit of mass	slug/12
Unit of mass density	$slug/12/in^3$
Density of aluminum	2.614×10^{-4} $slug/12/in^3$

6.1.3 Geometry Preparation

Geometry preparation means converting CAD specific geometry into FEA specific geometry, that is, one that captures all important model features but avoids unnecessary complexity and, at the same time, also meshes correctly.

A common mistake is an attempt to use fully featured, manufacturing-ready CAD geometry for analysis. Meshing such a complex CAD geometry often turns out to be impossible or the created mesh comes out too large to be solved or it is erroneous because of not having enough elements to capture the essential behavior, or it has excessively distorted elements. Even if you can create correct mesh based on the fully featured CAD geometry model, a very detailed representation of geometry results in an expensive FE model with little, if any, benefits to the quality of results.

When preparing analysis specific geometry, we should use Saint-Venant's principle and concentrate on modeling details in the regions of interest. Solid elements should not be necessarily our first choice. Instead, using shells, beams, or 2D representations should be considered.

The distinction between manufacturing-specific CAD geometry and analysis-specific FEA geometry is one of the challenges facing design engineers who use FEA and one of the most important topics covered in this book. We will further discuss it in chapter 13.

6.1.4 Definition of Material Properties

In most cases, material properties can be found in a material library that comes with the FEA program, but sometimes, we will need to create a custom material. Some programs offer a choice of assigning material properties either to the CAD model or to the FEA model.

If your FEA program offers this choice, it is more convenient to assign material properties to the CAD model, especially if a CAD model consists of many parts with different material properties. This reduces material assignment errors.

Notice that none of the geometric entities used for mesh creation becomes a part of FE model; geometry only serves as an aid in creating the FE mesh. Therefore, all properties assigned to CAD entities will be eventually transferred to respective element entities once geometry has been meshed. As explained in Table 5.1, volumes are meshed with solid elements, surfaces with shell elements, and curves are meshed with beam elements.

6.1.5 Definition of Boundary Conditions

Defining boundary conditions includes the definitions of loads and restraints. In FEA terminology, restraints are called essential boundary conditions and loads are called natural boundary conditions. Volume loads like gravity or inertial loads do not belong to the class of boundary conditions. Taking that into consideration, the popular term used in the FEA literature: "loads and boundary conditions" should be rephrased to either "volume loads and boundary conditions" or "natural and essential boundary conditions." Regardless of this fact, in many software manuals, the term "boundary conditions" is reserved for restraints. Restraints themselves are referred to as constraints, supports, fixtures, and so forth. As a side note, the FEA terminology is often inconsistent between different commercial programs leading to confusion and mistakes. Be sure to understand the meaning of all terms used by your program.

Boundary conditions are assigned to geometric entities of CAD model; after meshing they are transferred to nodes. Defining loads is usually easier than defining restraints. Loads are vectors; they have direction, sense and magnitude; magnitude is expressed in numbers. The definition of loads should be verified by examining the total load on the model and reaction forces. The definition of restraints is where severe errors are often made. The most common error is overrestraining the model. The correctness of restraint definition can be verified by examining reaction forces and comparing them with free-body diagrams that should always be prepared prior to analysis. Animation of displacement plots is a very useful technique when reviewing model restraint for correct definition.

The relative level of uncertainties in defining geometry, material, loads and restraints is qualitatively shown in Figure 6.1. The level of difficulty has no relation to time required for each step, so the message in Figure 6.1 may be counterintuitive. In fact, preparing analysis specific geometry may take hours or days, while applying restraints takes only a few mouse clicks.

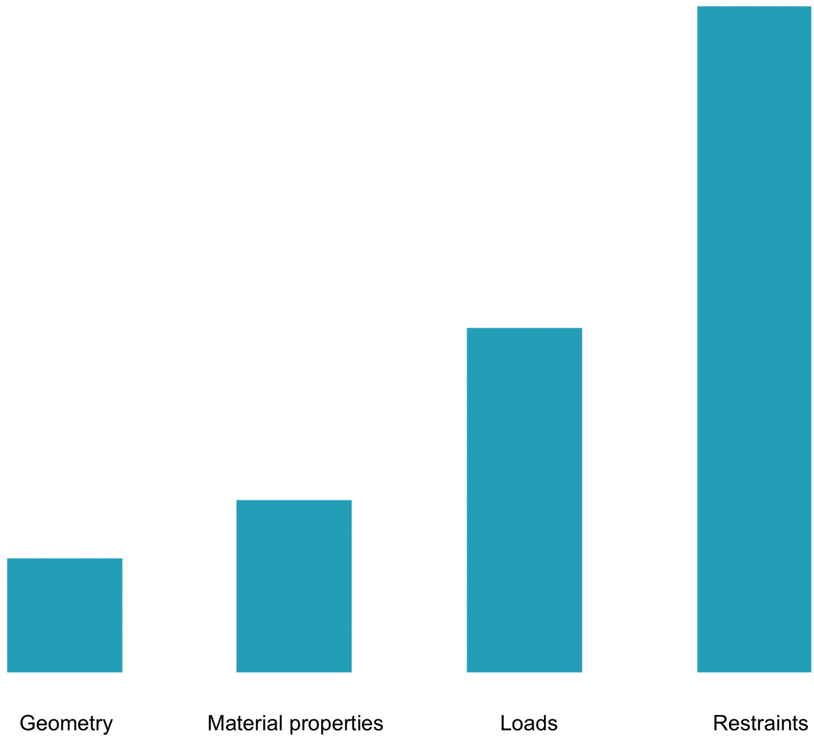


Figure 6.1 Qualitative comparison of uncertainty in defining geometry, material properties, loads, and restraints.

6.2 Modeling Techniques

6.2.1 Mirror Symmetry and Antisymmetry Boundary Conditions

The bracket shown in Figure 6.2 is subjected to bending load uniformly distributed over the cylindrical face. The load is pointing down; the back face is rigidly supported.

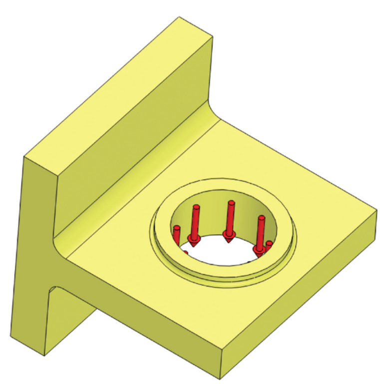


Figure 6.2 A symmetric bracket is loaded with tractions evenly distributed over the cylindrical face.

The model is characterized by mirror symmetry; we will refer to it as symmetry. Let us consider how the model will deform under this load. Because of the symmetry of geometry as well as the symmetry of loads and supports, an imaginary cross section along the plane of symmetry will remain flat in the deformed model. Having noticed that, we may simplify the problem by cutting the model in half along the plane of symmetry and removing one half of the model. One half of the original load needs to be applied to the remaining half of the model. However, the load needs to be cut in half only if it is defined as the total load; the load defined as force per area (pressure) remains of course the same. To make the remaining half of the model behave as if the other half was still there, we must define certain displacement boundary conditions to the cross face shown in blue in Figure 6.3. Those boundary conditions are necessary to keep the face flat and coincident with the plane of symmetry while the model deforms. Such conditions are called symmetry boundary conditions. Working with a model intended for meshing with solid elements whose nodes have 3 DOF, the symmetry boundary conditions are defined in terms of translations only; translations in the direction normal to the face are restrained and translations in in-plane directions are permitted. Working with a model intended for meshing with shell elements whose nodes have 6 DOF, the symmetry boundary conditions must be also defined in terms of rotations. Rotations in the direction normal to the plane of symmetry is permitted, and both in-plane rotations are eliminated.

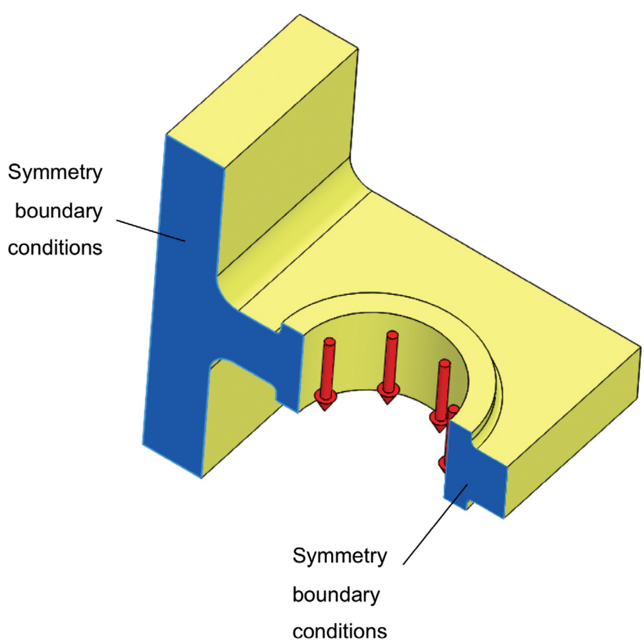


Figure 6.3 To make this half of the original geometry represent the whole model, the symmetry boundary conditions must be applied to the faces (blue) created by the symmetry cut along the vertical plane. One half of the total load is applied to this model. Either the right or left half of the model may be analyzed.

The model shown in Figure 6.2 can be also simplified by taking advantage of antisymmetry present in this problem as shown in Figure 6.4. The antisymmetry boundary conditions about certain plane apply when geometry and restraints are symmetric about that plane and loads are antisymmetric as shown in Figure 6.5. The antisymmetry boundary conditions are exactly opposite to the symmetry boundary conditions.

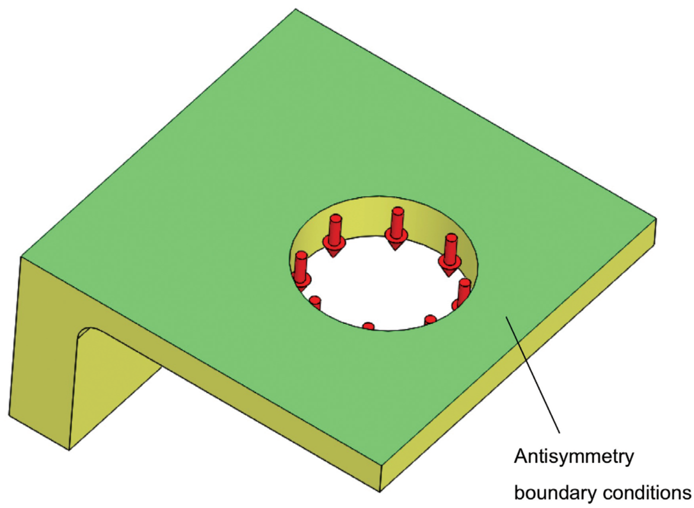


Figure 6.4 To make this half of the original geometry represent the whole model, the antisymmetry boundary conditions must be applied to the face (green) created by the symmetry cut along the horizontal plane. One half of the total load is applied to this model. Either the top or bottom half of the model may be analyzed.

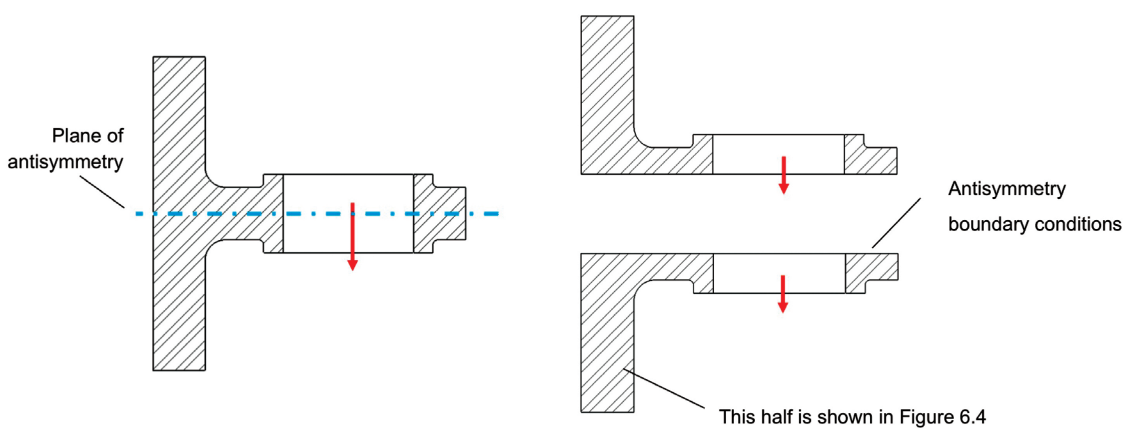


Figure 6.5 The plane of antisymmetry refers to the symmetry of geometry and the antisymmetry of loads. This illustration shows the cross section along the plane of symmetry only to show the hole. The model intended for analysis with antisymmetry boundary conditions is obtained by cutting the original model along the plane of antisymmetry only. Two halves of the model shown on the right serve to visualize the antisymmetry of loads.

Tables 6.2 and 6.3 explain how symmetry and antisymmetry boundary conditions are defined in each of the three principal planes. The comparison between the two tables shows that the antisymmetry boundary conditions can be seen as exactly opposite to the symmetry boundary conditions. The antisymmetry boundary conditions are less intuitive than the symmetry boundary conditions, but are easy to define simply by reversing the symmetry boundary conditions. Displacement components allowed in symmetry boundary conditions are restrained in antisymmetry boundary conditions and the other way around.

Table 6.2 Definition of symmetry boundary conditions in three planes of the global coordinate system

	Symmetry Boundary Conditions		
	Plane of symmetry		
	xy	*yz*	*xz*
Translation *x*	Free	Restrained	Free
Translation *y*	Free	Free	Restrained
Translation *z*	Restrained	Free	Free
Rotation *x*	Restrained	Free	Restrained
Rotation *y*	Restrained	Restrained	Free
Rotation *z*	Free	Restrained	Restrained

Table 6.3 Definition of antisymmetry boundary conditions in three planes of the global coordinate system

	Antisymmetry Boundary Conditions		
	Plane of antisymmetry		
	xy	*yz*	*xz*
Translation *x*	Restrained	Free	Restrained
Translation *y*	Restrained	Restrained	Free
Translation *z*	Free	Restrained	Restrained
Rotation *x*	Free	Restrained	Free
Rotation *y*	Free	Free	Restrained
Rotation *z*	Restrained	Free	Free

Using both symmetry and antisymmetry boundary conditions leads to further simplification, because now only one quarter of the bracket geometry needs to be modeled. This, of course, requires that only one quarter of the original load be applied; this is because the load is defined as a total load, not as load per unit of area. Different model simplifications are shown in Figure 6.6.

Antisymmetry boundary conditions are applicable to structures in bending; they are also applicable to analyzing symmetric structures under torsion, as shown in Figures 6.7 and 6.8.

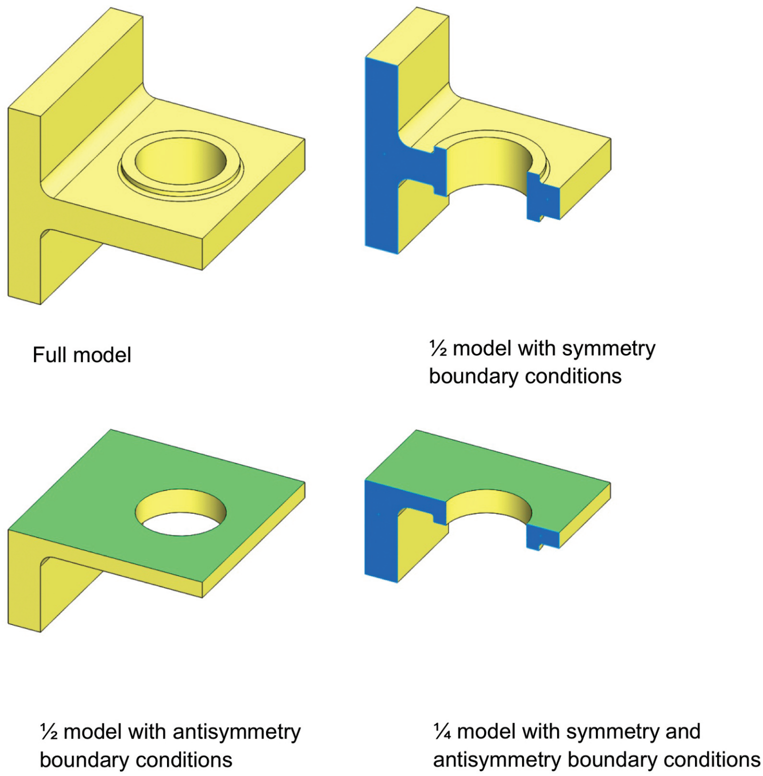


Figure 6.6 Faces with symmetry boundary conditions are colored blue, faces with antisymmetry boundary conditions are colored green. Either of the above four models can be used for analysis.

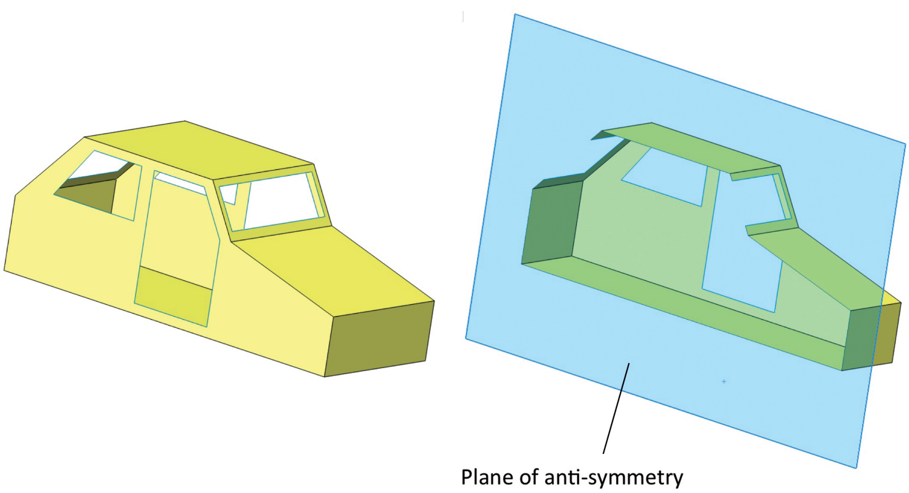


Figure 6.7 The use of antisymmetry boundary conditions allows modeling one half of an automotive body under torsion along the longitudinal axis.

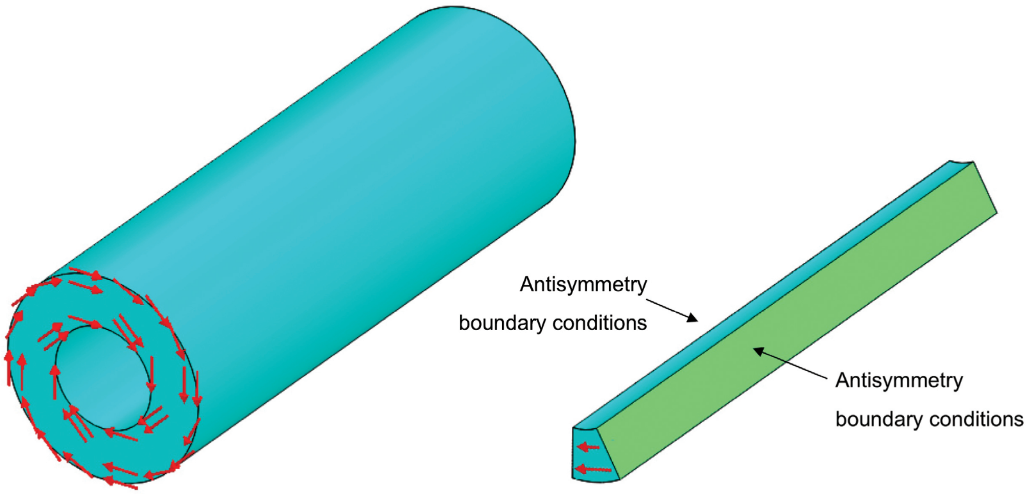


Figure 6.8 Antisymmetry boundary conditions are applied to the shaft under a torsional load. Because of the axial symmetry, an arbitrary slice (top right) may be analyzed by defining antisymmetry boundary conditions on two faces in the radial planes. Restraints applied to the back face are not shown.

Symmetry and antisymmetry boundary conditions are very useful techniques used to reduce the model size in structural analysis but should be used only very carefully in modal analysis. We will discuss this issue in chapter 9.

6.2.2 Axial Symmetry

If geometry and boundary conditions (restraints and loads) display the axial symmetry, then FEA model can be greatly simplified. Unlike symmetry and antisymmetry boundary conditions where the user needs to define boundary conditions along the planes of symmetry, the axial symmetry is included in the formulation of the 2D element used for axisymmetric analysis. An axisymmetric model in shown in Figure 6.9. Taking advantage of axial symmetry, only a planar axial cross section needs to be meshed and analyzed, as shown in Figure 6.9.

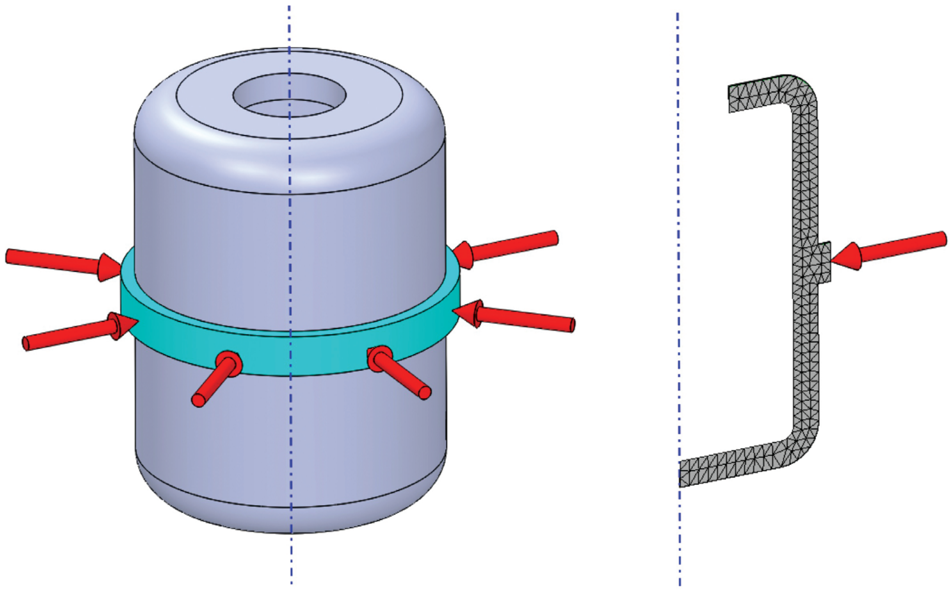


Figure 6.9 An axisymmetric tank, loaded with pressure distributed over 360° angle can be represented by a 2D radial cross section. This cross section is meshed with 2D axisymmetric elements. A load applied to a 2D axisymmetric model is interpreted as a load applied over 360° angle. Geometry details such as rounds are not modeled; this would limit the usefulness of this model.

6.2.3 Cyclic Symmetry

The other type of symmetry is cyclic symmetry also called repetitive symmetry. Cyclic symmetry may be of angular or linear type. Angular cyclic symmetry, often called circular symmetry, is applicable to shapes that may be created by an angular pattern of one segment. The repeatability must include not only geometry but also loads and restraints. Circular symmetry enforces the same displacements on the corresponding faces shown in Figure 6.10. A linear cyclic symmetry works with repeatable shapes that can be generated with linear patterns; an example is shown in Figure 6.11.

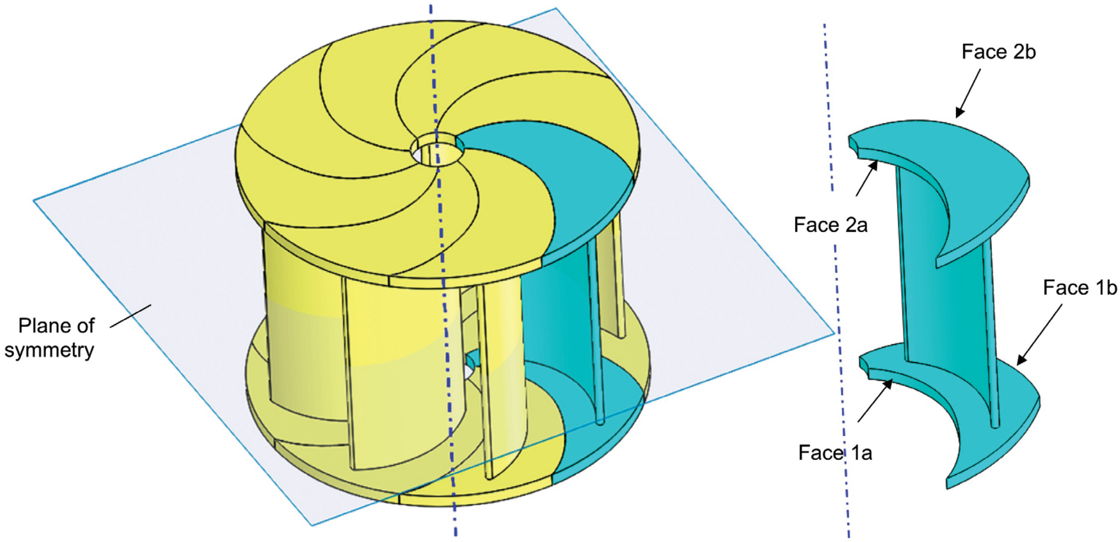


Figure 6.10 A centrifugal blower has circular symmetry. The section on the right is repetitive; it may be copied in a circular pattern about the axis of circular symmetry to create the complete model shown on the left. The blower is loaded with a centrifugal load, which is not shown in this illustration. By the virtue of definition of the circular symmetry, faces “a” and “b” of the section must have the same shape. Circular symmetry enforces the same displacements in the corresponding points of “a” and “b” faces. Geometry details such as rounds are not modeled; this would limit the usefulness of this model.

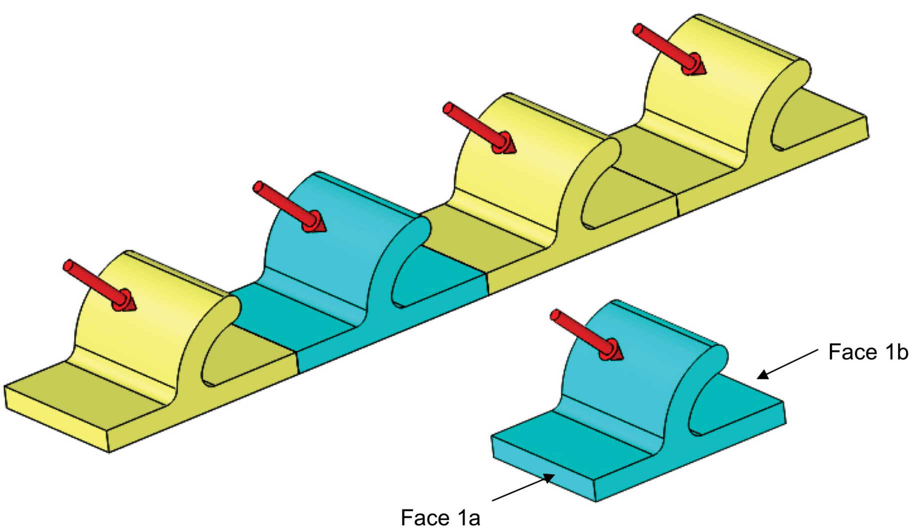


Figure 6.11 The repetitive patter can be obtained by copying one segment in a linear pattern. The repetitiveness must include geometry, loads, and supports, which are not shown in this illustration. By the virtue of definition of the cyclic symmetry, faces “a” and “b” of the section have the same shape. Cyclic symmetry enforces the same displacements in the corresponding points of “a” and “b” faces.

Combinations of different forms of symmetry can also occur. Figure 6.6 demonstrates the use of symmetry and antisymmetry in one model. The blower model in Figure 6.10 could be further simplified by taking advantage of symmetry about the midplane in-between the endplates.

6.2.4 Realignment of Degrees of Freedom

Each element node has a certain number of degrees of freedom (DOF) depending on the element type. As already discussed, nodes of solid elements have typically three translational DOF because element deformation can be fully described with nodal translations only; there is no need to introduce rotations. Nodes of 3D shell elements have 6 DOF: three translational and three rotational. Nodes of 2D plate elements have two translational DOF.

By default, the directions of DOF are aligned with the global coordinate system of the model. On certain occasions, it is convenient to realign those directions with other coordinate systems, either for easier definition of boundary conditions or for easier interpretation of results.

The realignment of the directions of DOF with a local cylindrical coordinate system, shown in Figure 6.12, allows modeling of a hinge support. As we will explain it in chapter 7, the analysis of this model requires careful consideration of nonlinear effects. Here, we assume that one of the supporting pins has the ability to "float" meaning that distance between pins is allowed to change.

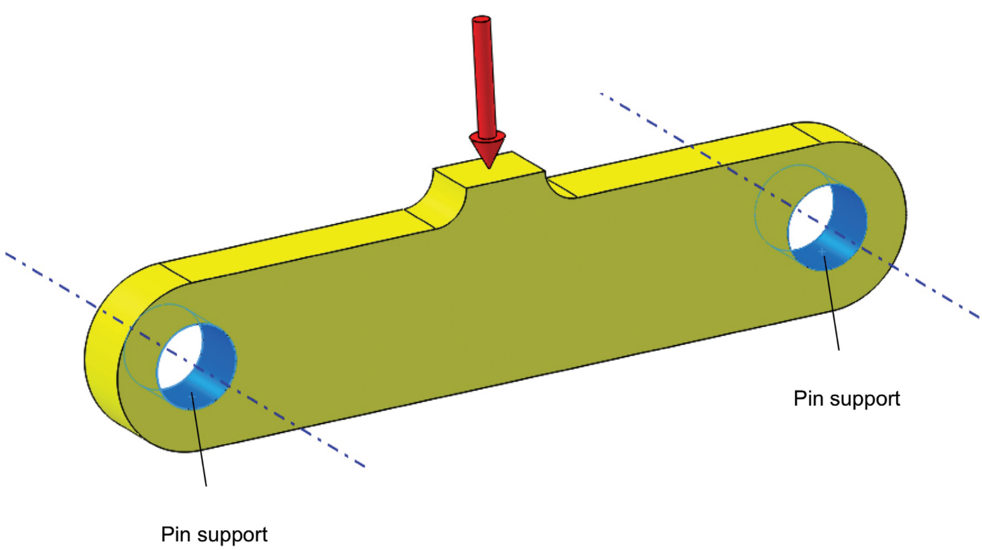


Figure 6.12 DOF on cylindrical surfaces are aligned with respective directions of cylindrical faces in the radial, tangential, and axial directions. Restraining radial translations simulate the pin support. Notice that the model as shown has one rigid body motion (translation in axial direction) that still needs to be restrained. Because geometry is intended for meshing with solid elements, there is no need to do anything about rotational DOF.

The realignment of the directions of DOF with a local spherical coordinate system, shown in Figure 6.13, allows modeling of a spherical bearing support. Notice that this realignment is made possible by modeling bearing with spherical faces, which, of course, is different from real bearing geometry.

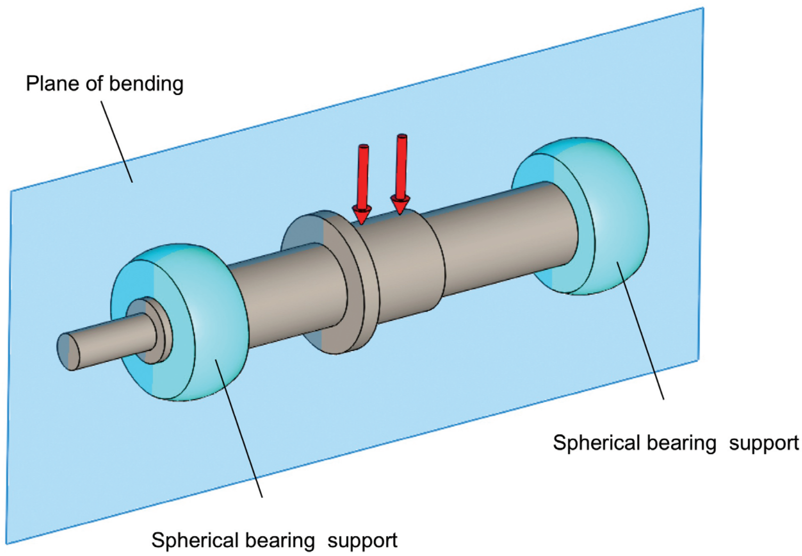


Figure 6.13 The shaft is supported by two spherical bearings that allow some rotations in the plane of bending. To model the spherical bearing support, the DOF on spherical faces are aligned with respective directions of spherical faces: radial, longitudinal and latitudinal directions. The support offered by the spherical bearing is then modeled by restraining translations in the radial direction. Notice that model restrained in such a way will have one rigid body motion (rotation about the shaft axis) that still needs to be restrained. Because geometry is meshed with solid elements, there is no need to do anything about rotational DOF.

6.3 Hands-On Exercises

6.3.1 BRACKET02-1 (Figure 6.14)

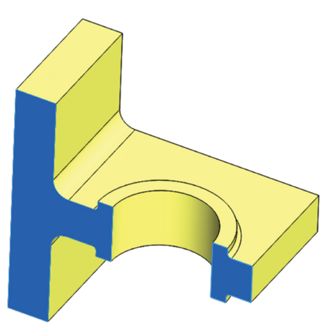


Figure 6.14 Define symmetry boundary conditions on the blue faces. Apply half of the total load to the cylindrical face.

Model name

- 6.01.BRACKET02.x_t
- 6.01.BRACKET02.sldprt

6.3.1.1 Objective: Demonstrate the use of symmetry boundary conditions.

6.3.1.2 Description: The bracket is supported along the back side and loaded with a 10000-N load uniformly distributed as tractions to the cylindrical face, as shown in Figure 6.2. A fixed restrain is applied to the back face.

The following are the required steps:

1. If you use Parasolid geometry, cut the model in half as shown in Figure 6.14; if you use SOLIDWORKS geometry, switch to configuration *02 half sym*.
2. Apply material properties (steel).
3. Apply the fixed restraint to the back face and apply symmetry boundary conditions to two faces located in the plane of cut (Figure 6.14).
4. Apply half of the total load to the cylindrical face; refer to Figure 6.3 for load direction.
5. Mesh.
6. Analyze the animated displacement plots and stress plots.

6.3.2 BRACKET02-2 (Figure 6.15)

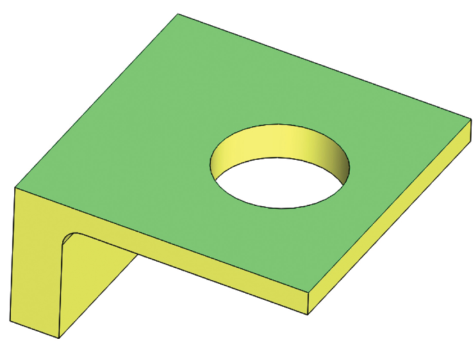


Figure 6.15 Define antisymmetry boundary conditions on the green face. Apply half of the total load to the cylindrical face.

Model name

- 6.01.BRACKET02.x_t
- 6.01.BRACKET02.sldprt

6.3.2.1 Objective: Demonstrate the use of antisymmetry boundary conditions to model bending.

6.3.2.2 Description: The bracket is supported along the back side and loaded with a 10000-N bending load. Only one half is modeled, the other half is to be simulated with antisymmetry boundary conditions.

The following are the required steps:

1. If you use Parasolid geometry, cut the model in half as shown in Figure 6.15; if you use SOLIDWORKS geometry, switch to configuration *03 half anti sym*.
2. Apply the material properties (steel).
3. Apply the fixed restraint to the back face and define the antisymmetry boundary conditions (only out-of-plane displacements are allowed).
4. Apply half of the total load to the cylindrical face.
5. Mesh.
6. Analyze the animated displacement plots and stress plots compared with the exercise in Section 6.3.1.

6.3.3 BRACKET02-3 (Figure 6.16)

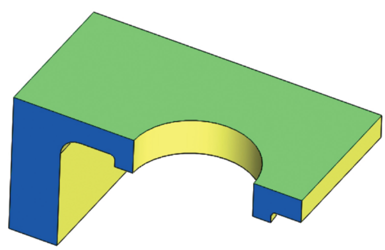


Figure 6.16 Define symmetry boundary conditions on the blue faces; define antisymmetry boundary conditions on the green face. Apply one-fourth of the total load to the cylindrical face.

Model name

- 6.01.BRACKET02.x_t
- 6.01.BRACKET02.sldprt

6.3.3.1 Objective: Demonstrate the use of symmetry and antisymmetry boundary conditions to model bending.

6.3.3.2 Description: This exercise combines the modeling techniques practiced in exercises 6.3.1 and 6.3.2. If you use Parasolid geometry, cut the model in quarter as shown in Figure 6.16; if you use SOLIDWORKS geometry, switch to configuration *04 quarter*. Remember to apply one-fourth of the total load to the cylindrical face. Compare the displacement and stress results produced in exercises 6.3.1–6.3.3. The results will be slightly different only because of different discretization errors in each solution. Once convergence analysis is conducted, the results of all three models should converge to the same values.

6.3.4 Shaft (Figure 6.17)

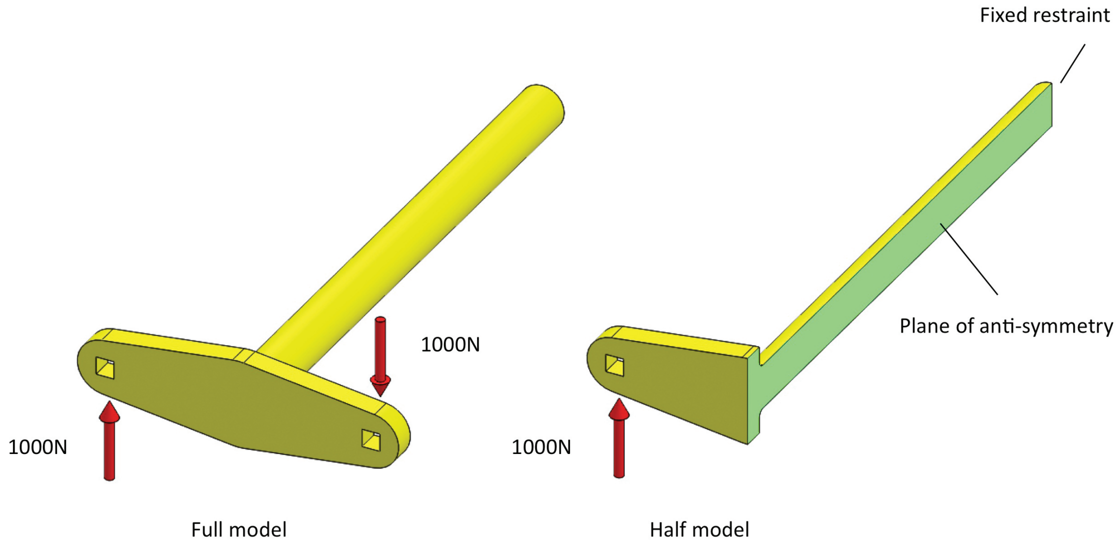


Figure 6.17 The shaft is supported at the back and loaded with a couple of 1000 N forces applied to square holes. Load is applied to holes in the direction of arrows; it is uniformly distributed over flat face that forms a part of hole. Only one half of the shaft needs to be analyzed if antisymmetry boundary conditions are used.

Model name

- 6.02.SHAFT.x_t
- 6.02.SHAFT.sldprt

6.3.4.1 Objective: Demonstrate the use of antisymmetry boundary conditions to model torsion.

The following are the required steps:

1. If you use Parasolid geometry, cut the model in half as shown in Figure 6.17; if you use SOLIDWORKS geometry, switch to configuration *02 half*.
2. Apply the material properties (steel).
3. Apply the fixed restraint to the end face.
4. Apply antisymmetry boundary conditions (only out-of-plane displacements are allowed).
5. Apply a load as shown in Figure 6.17.
6. Mesh.
7. Analyze the animated displacement plots and stress plots.
8. Repeat the analysis using full geometry; compare the displacement and stress results.

6.3.5 Pressure Tank (Figure 6.18)

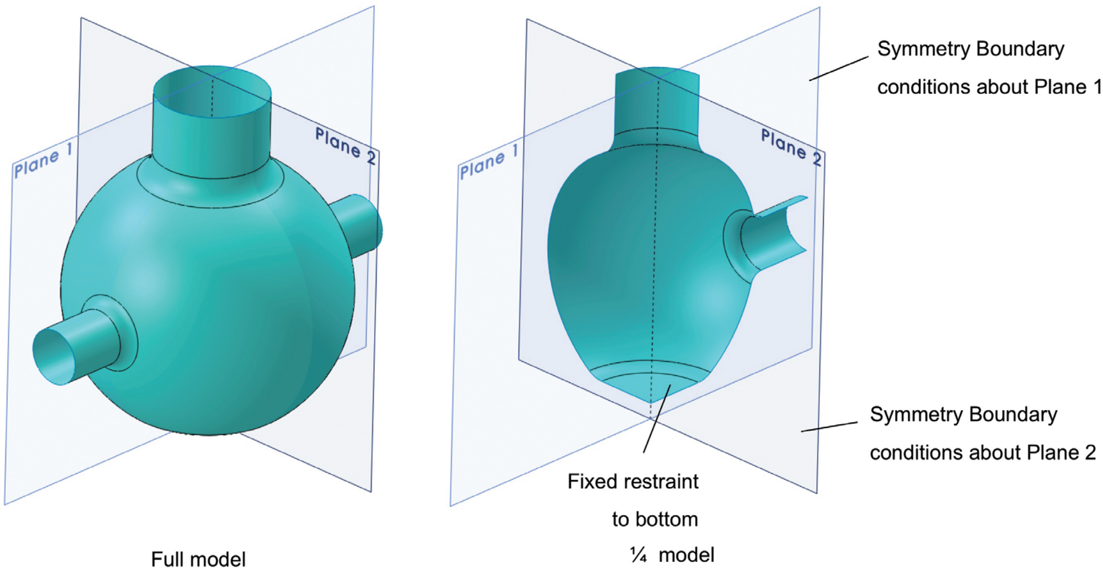


Figure 6.18 A pressure vessel is subjected to a 1-MPa internal pressure. Taking advantage of double symmetry, only one-fourth of the model may be analyzed. The geometry is represented by surfaces and this necessitates the use of shell elements. Because shell elements are used, symmetry boundary conditions must be defined in terms of all 6 DOF. Pressure and symmetry boundary condition symbols are not shown.

Model name

- 6.03.PRESSURE_TANK.x_t
- 6.03.PRESSURE_TANK.sldprt

6.3.5.1 Objective: Demonstrate the use of symmetry boundary conditions in a shell element model where all 6 DOF must be dealt with. Use the quarter model geometry.

6.3.5.2 Comment: Shell elements are used because the pressure vessel is thin walled; this gives an opportunity to work with symmetry boundary conditions defined in terms of all 6 DOF. You may want to repeat this exercise using a similar model of a pressure tank represented by a solid geometry.

The following are the required steps:

1. If you use Parasolid geometry, cut the model twice to work with a quarter geometry; if you use SOLIDWORKS geometry, switch to configuration *02 quarter*.
2. Define the material properties (steel).
3. Define the plate thickness (3 mm).
4. Define the fixed restraints to the flat bottom.

5. Define the symmetry boundary conditions along the edges in Planes 1 and 2. Translation in the direction normal to the plane of symmetry is restrained and two in-plane rotations are restrained; consult Table 6.2.
6. Define a pressure load of 1 MPa; As this is a pressure and not the total load, the pressure magnitude does not change when working with one-fourth of the model.
7. Mesh and use a fine mesh.
8. Analyze the animated displacement plots and stress plots.
9. Repeat the analysis using the full model.
10. Compare the results of analyzes of the quarter model and the full model.

6.3.6 RING (Figure 6.19)

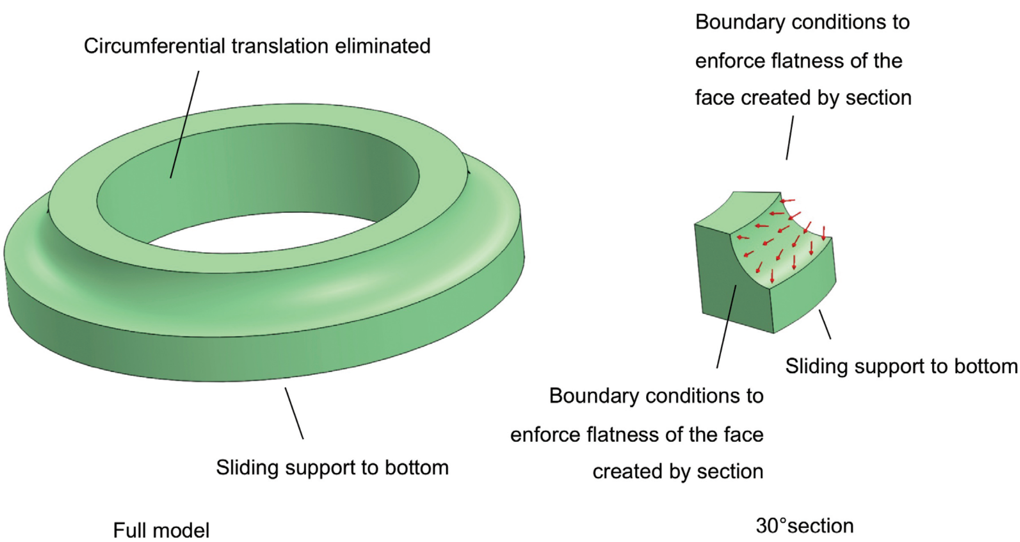


Figure 6.19 An axisymmetric ring is subjected to pressure applied to the toroidal face. If the full model is analyzed, then restraints must be defined in the cylindrical coordinate system associated with the inner face and circumferential displacements must be restrained. Taking advantage of axial symmetry, the analysis may be conducted on an arbitrary slice (here 30°) with boundary conditions enforcing flatness of faces in the radial planes. The sliding support to the bottom applies both to the full model and the section model.

Model name

- 6.04.RING.x_t
- 6.04.RING.sldprt

6.3.6.1 Objective: Demonstrate the use of symmetry boundary conditions in a solid element model with axial symmetry.

6.3.6.2 Comment: The ring has sliding support applied to the bottom face. Restraints to the cylindrical face are defined in a cylindrical coordinate system; restraints in circumferential directions are restrained (Figure 6.19); notice that this set of restraits fully restraints the model.

A 100MPa pressure is applied to the toroidal face. Because of the axial symmetry, the problem can be represented either by a flat radial cross section requiring the use of 2D axisymmetric elements or by an arbitrary slice suitable for meshing with solid elements; here we use a 30° section. Notice that the term "symmetry boundary condition" may be confusing when applied to the 30° section. In this case, the term "symmetry boundary condition" means that the out-of-plane displacements are not allowed; this is done to enforce the flatness of faces created by two radial cuts.

The 30° section with boundary conditions enforcing flatness of radial faces would still have one rigid body motion. To complete the definition of restraints, define the sliding support to the bottom; this restraint eliminates translations normal to the face just like the other two restraints, which we call here "symmetry boundary conditions."

The following are the required steps:

1. If you use Parasolid geometry, cut the model to create a 30° section, if you use SOLIDWORKS geometry, switch to configuration *02 section*.
2. Apply the material properties (steel).
3. Apply the restraints and pressure load as shown in Figure 6.19.
4. Mesh and solve.
5. Analyze animated displacements and stresses.
6. Repeat the analysis using the full model. Notice that sliding support to the bottom will leave the model with three rigid body motions. To eliminate these three rigid body motions, define restraints in the cylindrical coordinate system associated with the inner cylindrical face and restrain circumferential displacements (Figure 6.19).

6.3.7 Link (Figure 6.20)

Model name

- 6.05.LINK.x_t
- 6.05.LINK.sldprt

6.3.7.1 Objective: Demonstrate the use of symmetry boundary conditions in a solid element model. Demonstrate how the realignment of DOF allows modeling of a hinge support.

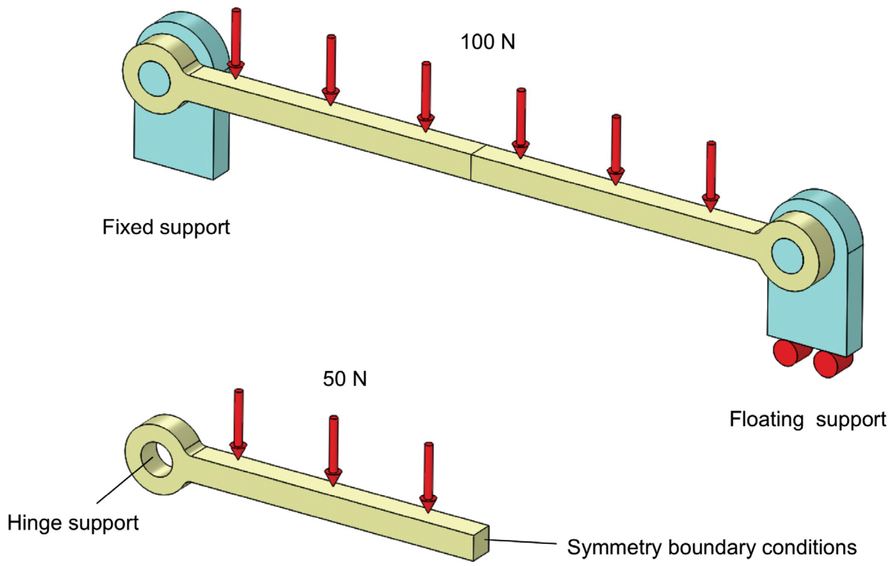


Figure 6.20 A pin supported link is loaded with a 100-N load uniformly distributed over the top face. The left pin is fixed and the right one has a floating support represented here by rollers. If contact stresses between pins and link "eyes" are not of interest, then the analysis may be conducted on the link only, rather than on the assembly. Restraints must be defined in the cylindrical coordinate systems associated with respective holes. Taking advantage of mirror symmetry, the analysis may be conducted on one half of the model with symmetry boundary conditions defined as shown and half of the load applied to the top face.

6.3.7.2 Comment: Assuming that contact stresses developing between the link and pin are not of interest, we will define link support as a hinge that allows rotation of the hole. Because of model symmetry, the problem can be represented by one half of the model. We assume that one of hinge supports is "floating" meaning that the distance between hinges may change. This prevents the link form developing membrane stresses; the analysis of membrane stresses would require a nonlinear geometry analysis. This will be discussed in chapter 7.

The following are the required steps:

1. If you use Parasolid geometry, cut the model in half; if you use SOLIDWORKS geometry, switch to configuration *02 half*.
2. Apply the material properties (Nylon 101).
3. Align the DOF of the hole face with a local cylindrical coordinate system. After realignment, the directions of translational DOF are radial, circumferential, and axial. There are no rotational DOF because the model will use solid elements.

4. Restrain radial and axial DOF.
5. Apply symmetry boundary conditions.
6. Apply half load (50 N) to the top face as shown.
7. Mesh and solve.
8. Analyze the animated displacement plots and stress plots.
9. Repeat the analysis using the complete model and compare the results.

Chapter 7
Nonlinear Static Structural Analysis

This chapter starts a review of different types of analyses performed with finite element analysis (FEA). The review starts with nonlinear geometry analysis; other types of analyses are covered in subsequent chapters.

The most commonly performed type of analysis, the linear static analysis, has been already introduced in the preceding chapters.

7.1 Classification of Different Types of Nonlinearities

Before we discuss different types of nonlinear static structural analyses, we need to review some terms.

- *Structural Analysis:* An analysis that deals with deformable bodies; it finds displacements, strains, and stresses. The opposite of a deformable body is a rigid body that does not deform under the applied load. Rigid bodies are universally used in mechanism analysis, but are useless in structural analysis.
- *Linear Structural Analysis:* A structural analysis of a body who's stiffness does not change during the process of load application.
- *Static Analysis:* Structural analysis with loads that do not change with time.

Recall the fundamental FEA equation

$$[K] * [d] = [F] \tag{2.1}$$

where $[K]$ is the known stiffness matrix, $[d]$ is the unknown vector of nodal displacements, and $[F]$ is the known vector of nodal loads. Until now, we have always assumed that the stiffness matrix $[K]$ is during the process of load application. The only exception was the introduction of an elastic–perfectly plastic material with a strain–stress curve shown in Figure 4.20, which was used to discuss stress singularities.

The assumption that stiffness does not change during the process of load application is what defines linear analysis. If stiffness remains constant, then stiffness matrix $[K]$ needs to be calculated only once.

If stiffness does change during the process of load application, then analysis is nonlinear and it is necessary to update stiffness matrix $[K]$ while loads are being applied.

Nonlinear analysis may be classified based on the source of nonlinear behavior. In this chapter, we will review nonlinear effect caused by changes in model geometry. This analysis is called nonlinear geometry analysis.

We will discuss three subcategories of nonlinear geometry analysis: large displacements, membrane stress stiffening, and contact, along with other classification.

7.2 Large Displacement Analysis

In nonlinear geometry analysis, displacements change the structure's shape and, consequently, structure stiffness. The stiffness matrix must then be updated during the process of load application.

Obvious candidates for large displacements analysis are structures that visibly experience large displacement. Therefore, let us start with a problem where displacements are clearly large and cause a significant change in model shape. Such a problem is a prismatic cantilever beam subjected to pressure (Figure 7.1). It takes 1s to change pressure from 0 to 0.03MPa as shown by the load time history graph in Figure 7.2. Time steps are, in this case, automatically selected by solver. It is important to remember that time on the ordinate is a pseudotime; it is used only to define the shape of the load time history curve and no inertial effects are considered; remember that analysis is static. It could have been 10 or 0.1 s just as well. The red markers indicate steps 0 (no load), 4, 7, and 13; the corresponding pressure magnitude and beam displacements are shown in Figure 7.3.

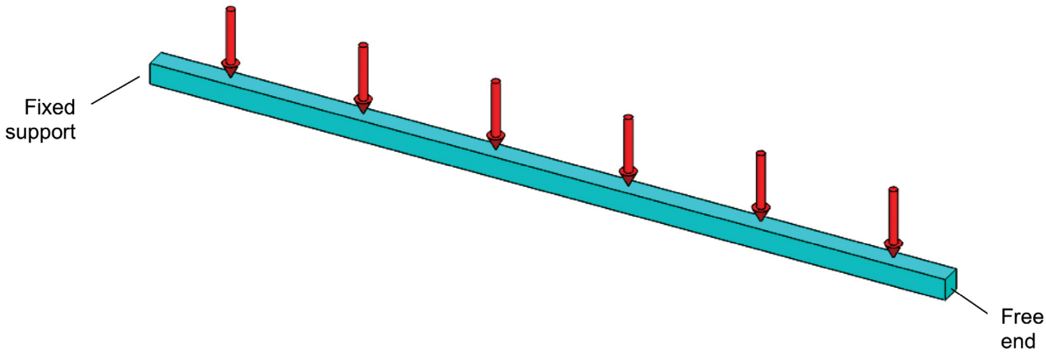


Figure 7.1 A cantilever beam subjected to pressure shown here in the undeformed shape.

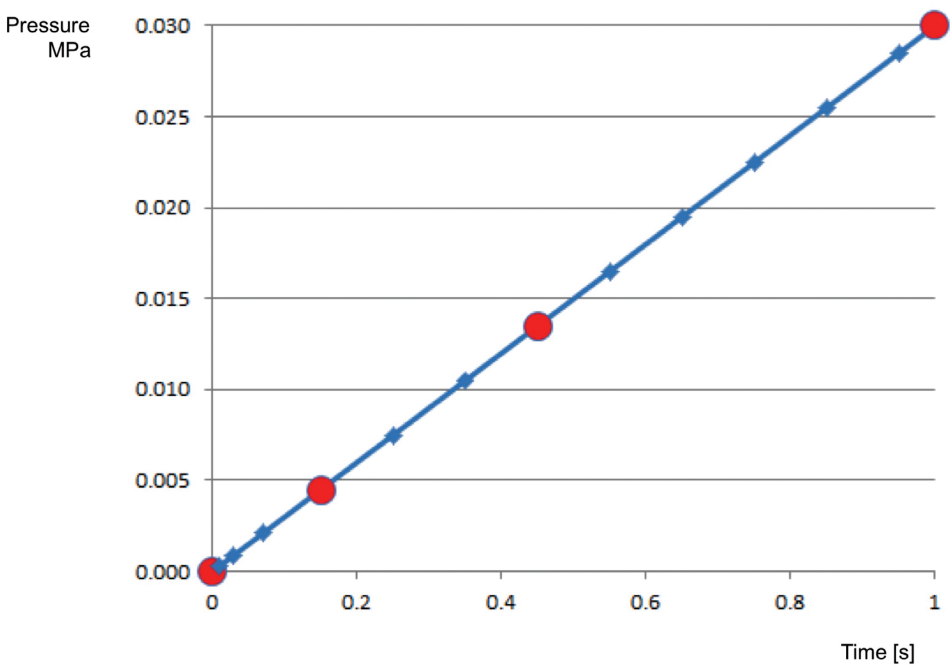


Figure 7.2 The time history of pressure applied to the cantilever beam in Figure 7.1.

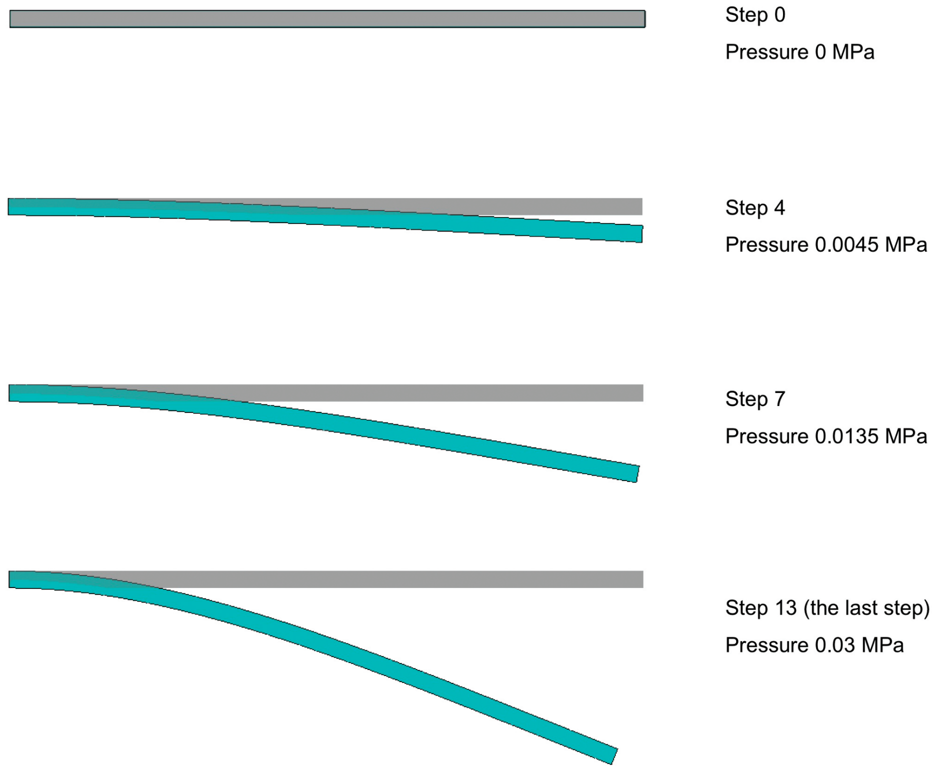


Figure 7.3 Progressive deformation of the cantilever beam subjected to pressure. Pressure remains normal to the top face while the model deforms and the pressure increases from 0 to 0.03 MPa in 13 automatically selected increments. The 1:1 scale of displacement is recommended for presenting the results of large displacement analysis as it is done above.

Model stiffness is updated during the process of load application; it is done before every next load increment. The load direction is also updated because in this example, we assume that pressure follows the deforming beam, meaning that pressure remains normal to the deforming face (Figure 7.4). Another possibility would be a nonfollowing load that retains its original direction relative to global coordinate system (Figure 7.5). The nonfollowing load could be produced by weight added to the top face. Let us repeat the above definitions: load retaining its direction in relation to the current configuration of model is called a nonconservative or following load; the load retaining its absolute direction in relation to the global coordinate system is called a conservative or nonfollowing load.

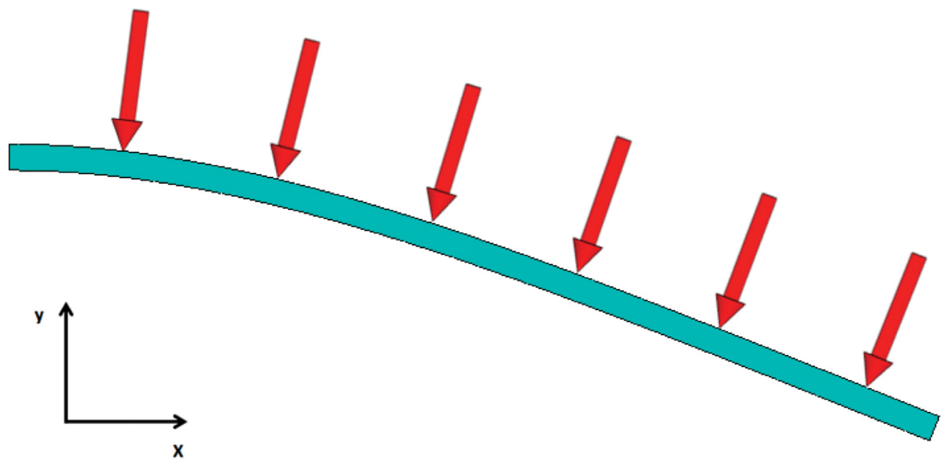


Figure 7.4 The following load keeps its original orientation relative to the model. Here, the pressure load remains normal to the top face. The following load is also called nonconservative load because it does not retain the original direction.

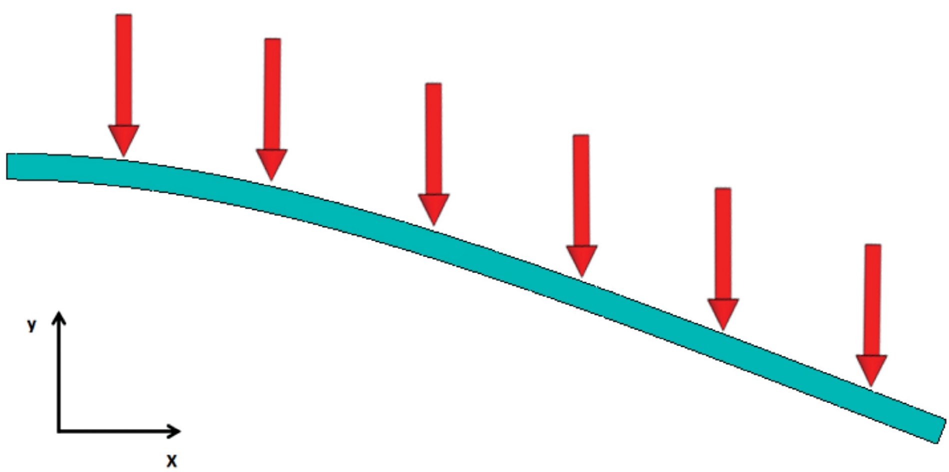


Figure 7.5 A nonfollowing load retains its original orientation relative to a global coordinate system. The nonfollowing load is also called conservative load because it retains the original direction relative to a global coordinate system.

It is interesting to compare the pattern of deformation of the cantilever beam in linear and nonlinear analyses. In linear analysis (Figure 7.6), the end of beam travels along a straight line. This would be acceptable only if displacements were small. An apparent rotation of beam end face is caused by elongation of the top layers because of elongation

of top layers and contraction of bottom layers in the beam experiencing bending. Notice that the midpoint on the free end of the beam still travels along a straight dashed line shown in Figure 7.6. Compare this incorrect pattern of deformation produced by linear analysis with the results of nonlinear geometry analysis in Figure 7.3.

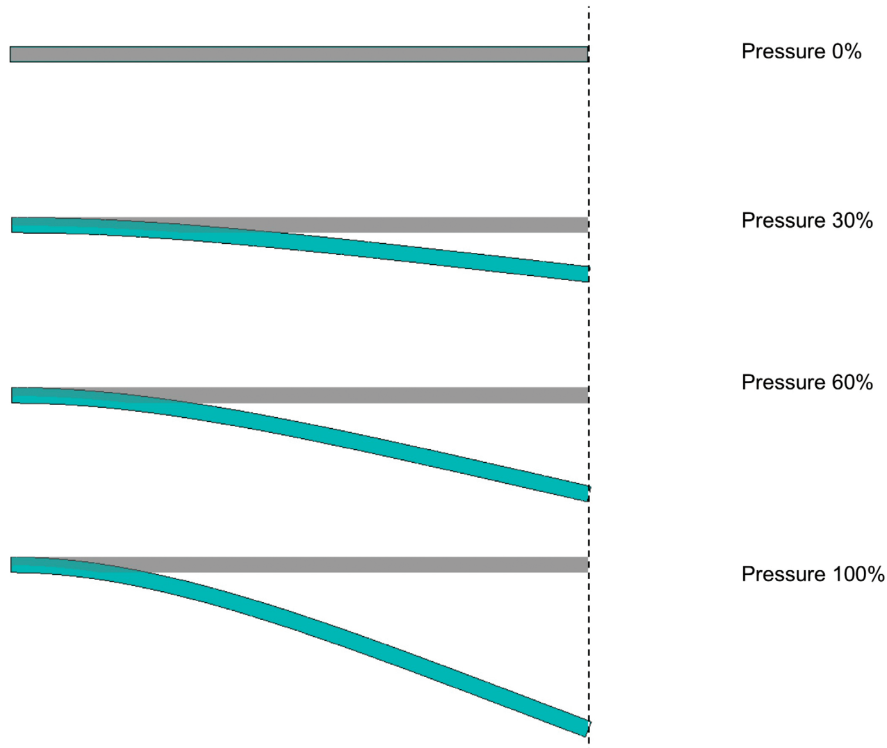


Figure 7.6 Appearance of model deformation in linear analysis.

For another comparison between incorrect results of linear and correct results of large displacement analysis, consider a round shaft loaded with a couple of forces (Figure 7.7). An apparent stretch of lug and growth in shaft diameter observed in the results of linear analysis, shown in Figure 7.8, is a manifestation of exceeding the limits of linear analysis. In linear analysis, each point on the shaft surface travels along the circumferential direction, normal to the radius, and this produces the apparent “swelling” of the shaft explained in Figure 7.9.

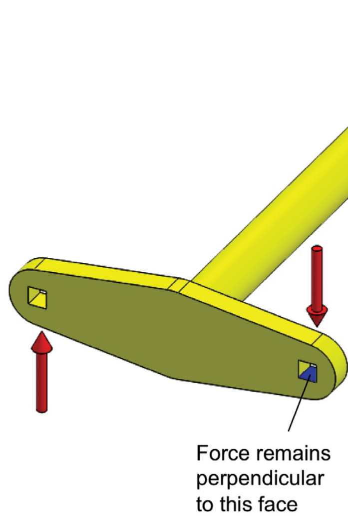


Figure 7.7 The torsion bar loaded with a couple of forces. The load retains its direction relative to the deforming model. The force load is perpendicular to the blue face. The far end of the round shaft is fixed.

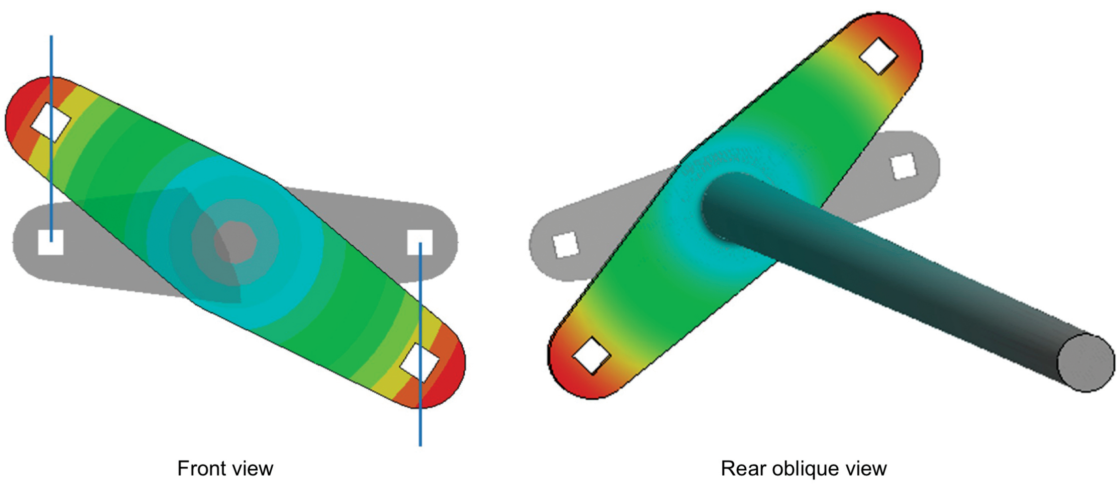


Figure 7.8 Results of linear analysis. Notice that holes travel along straight lines and shaft "swells" seemingly increasing its diameter. This is because in linear analysis, load is applied in just one step and model shape is not updated during the process of load application. Displacements are superimposed on the undeformed model and there are no intermediate shapes between the undeformed model and deformed shape.

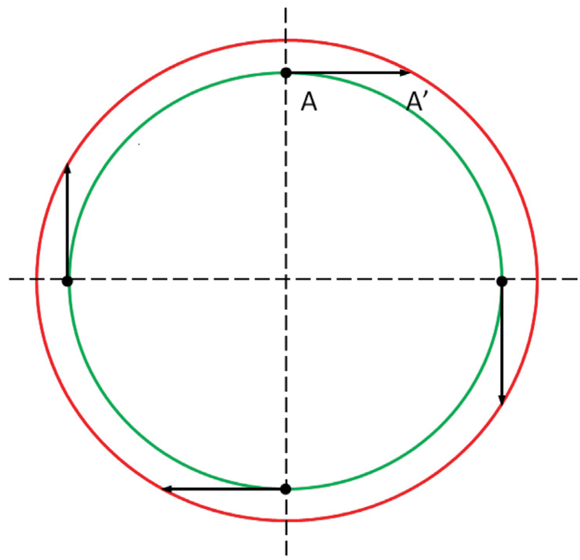


Figure 7.9 The green circle is the shaft diameter before torque load has been applied; the red circle in the shaft diameter when loaded with torque in linear analysis. Because the analysis is linear, trajectories of points traveling from undeformed to deformed position are straight lines; point A displaces along a straight line to position A’ This is repeated for all points on the circumference of the green circle and produces an apparent increase of shaft diameter.

Considering the magnitude of displacements, the shaft problem requires nonlinear geometry analysis; the results of nonlinear analysis are shown in Figure 7.10. A similar misapplication of a linear analysis is shown in Figure 7.11. Whenever you see model “stretching” (beam in bending) or “swelling” (shaft in torsion), go back to problem definition and specify a nonlinear geometry analysis, specifically large displacement analysis.

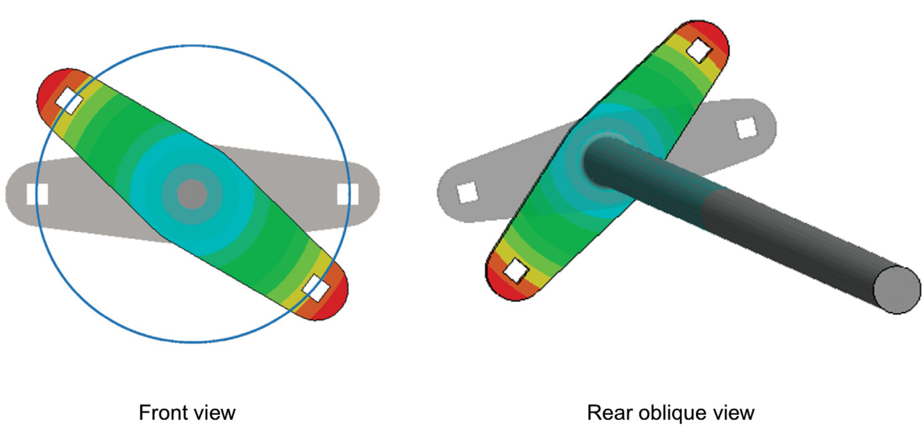


Figure 7.10 Results of nonlinear geometry analysis; notice that holes travel along an arc as they should and shaft diameter does not increase.

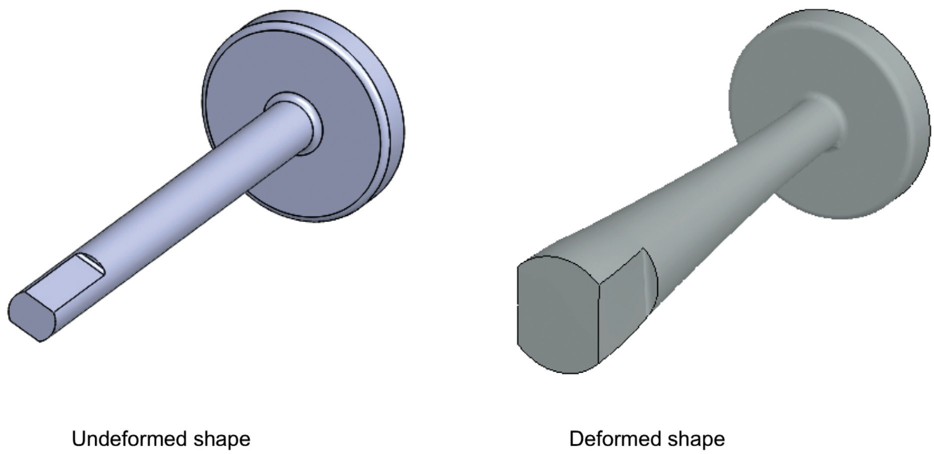


Figure 7.11 Model of shaft under torsion before deformation (left) and after deformation (right). The deformed shape, produced by a linear analysis shows "swelling" of shaft in torsion. Notice that such a deformed shape can be easily produced with correct results of a linear analysis using a sufficiently large scale of deformation. However, if the deformed shape above uses 1:1 scale of deformation, then this is a clear indication that a nonlinear large displacement analysis should have been used in place of a linear analysis.

The above problems of beam in bending and shaft in torsion required nonlinear geometry analysis because large displacements were changing stiffness significantly.

7.3 Membrane Stress Stiffening

A classic example of problem requiring nonlinear geometry analysis, even though displacements are small in relation to overall model size, is a thin flat plate under pressure (Figure 7.12). Initially, the only mechanism available to resist the load is bending stiffness produced by bending stresses. During the process of deformation, the plate acquires membrane stiffness (Figure 7.13) in addition to the original bending stiffness.

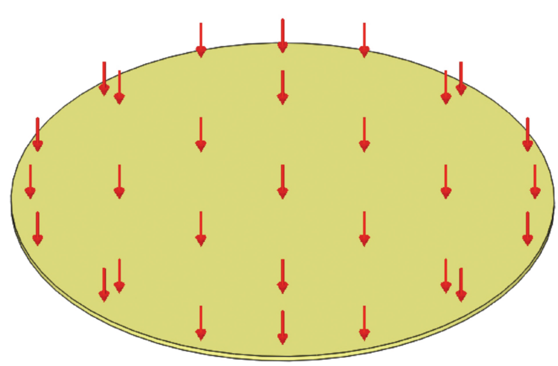


Figure 7.12 A thin flat plate of diameter 500 mm and thickness 5 mm is rigidly supported along the circumference and subjected to pressure slowly changing from 0 to 0.3 MPa. The plate material is steel.

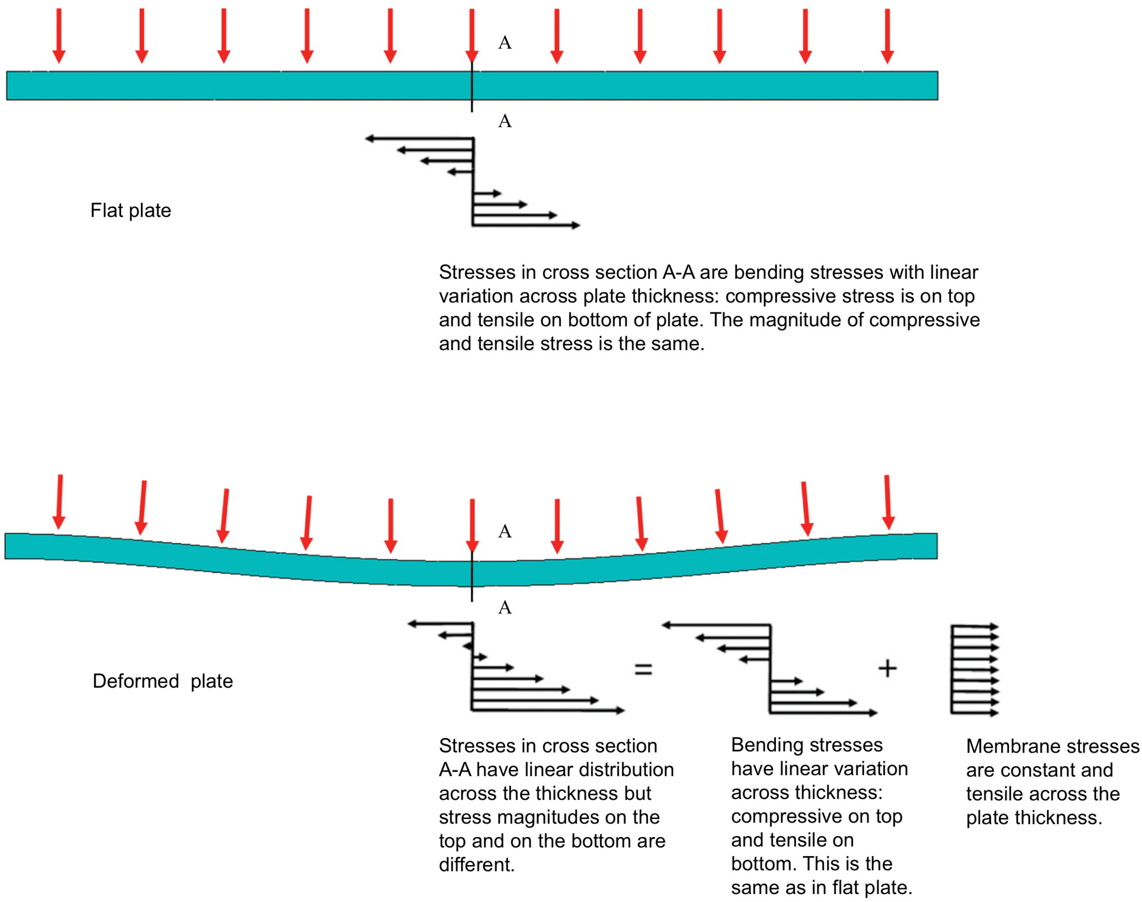
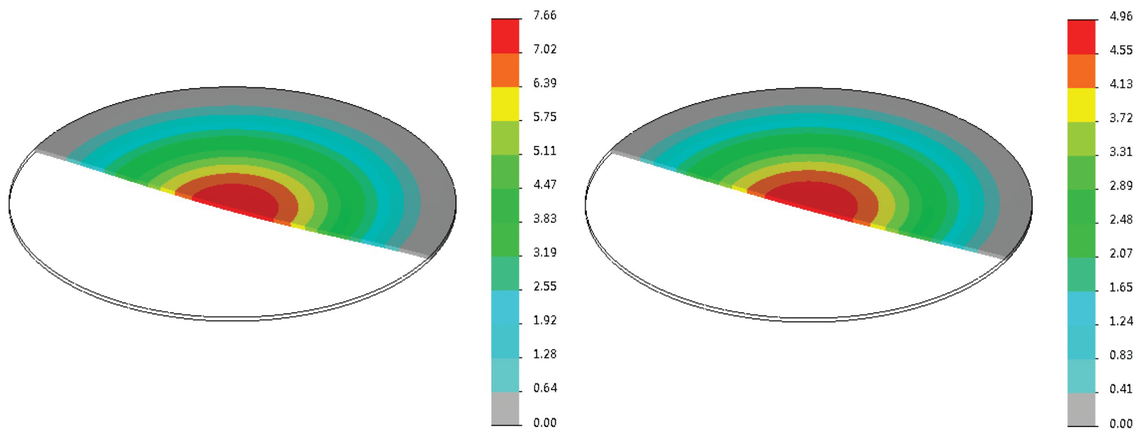


Figure 7.13 A flat plate responds to pressure with bending stresses only; it has bending stiffness only. The deformed plate responds to pressure with bending stresses and tensile (membrane) stresses and has bending and membrane stiffness. Membrane stiffness has been created during the process of load application.

Neglecting to account for nonlinear effect in the model defined in Figure 7.12 causes 55% error in displacements results. This huge error occurs despite the fact that the plate diameter is 500 mm and the maximum plate displacement is only 5 mm when accounting for membrane stiffness effect and 7.7 mm as erroneously reported by the linear analysis (Figure 7.14).

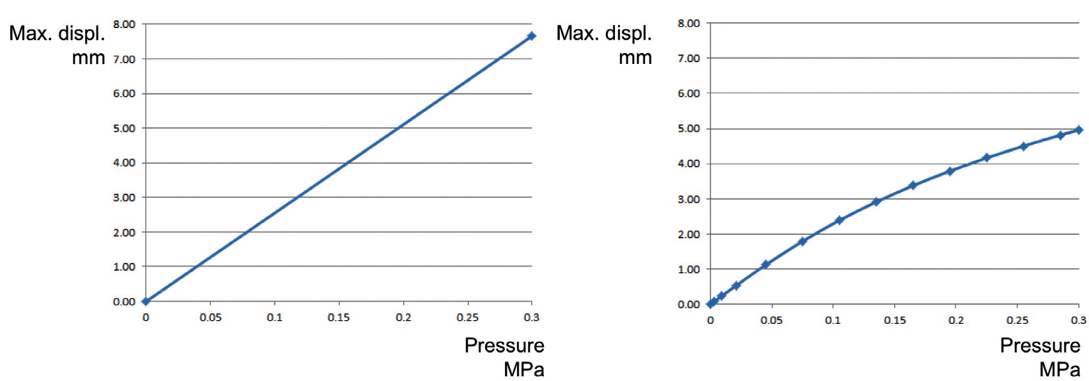
The comparison between the load–displacement curves produced by linear and nonlinear solutions is shown in Figure 7.15. The curved load–displacement graph clearly demonstrates that stiffness is increasing during the process of load application. The round plate problem illustrates that linear analysis is unable to model membrane stiffness, which does not exist prior to load application.



Linear solution; maximum displacement 7.66 mm

Nonlinear solution; maximum displacement 4.96 mm

Figure 7.14 Displacement results from the linear and nonlinear solutions are compared here. The deformation scale is 1:1. Even though section clipping is used to visualize deformation better, the deformation is barely visible. Neither the displacement of 7.66 mm or 4.96 mm qualifies as a "large displacement," still the problem requires nonlinear geometry analysis.



Linear solution found in one step;
maximum displacement 7.66 mm

Nonlinear solution found in 13 steps;
maximum displacement 4.96 mm

Figure 7.15 The load-displacement curve in a linear solution is a straight line. Plate stiffness is calculated for the original (flat) shape before any load has been applied. Stiffness is not updated and load is applied in one step. Results of the linear analysis can be freely scaled to find results for different load magnitudes. The load displacement curve in nonlinear solution is a curve built on 13 points, and the solver performed 13 load increments and 13 stiffness updates. The shape of load–displacement curve demonstrates that plate stiffness is increasing, while it deforms under pressure.

Consider a link supported by two hinges in Figure 7.16. If linear analysis is used on this model, it will always assume that one of hinges is floating (notice roller support in top illustration). When hinges are in a fixed position (Figure 7.16, bottom), pressure applied to the top face deforms the link that develops both bending and membrane stresses. However, linear analysis cannot model that. Linear analysis finds the original stiffness (before any deformation has taken place) and that stiffness is based on bending stresses only; membrane stiffness is not accounted for (Figure 7.13). Linear analysis cannot distinguish between fixed and floating supports of a hinged link shown in Figure 7.16. When analyzed with linear analysis, both configurations shown in Figure 7.16 produce the same displacements in the direction of load; the floating hinge will not move at all. Therefore, the analysis of configuration with both hinges fixed requires nonlinear geometry analysis.

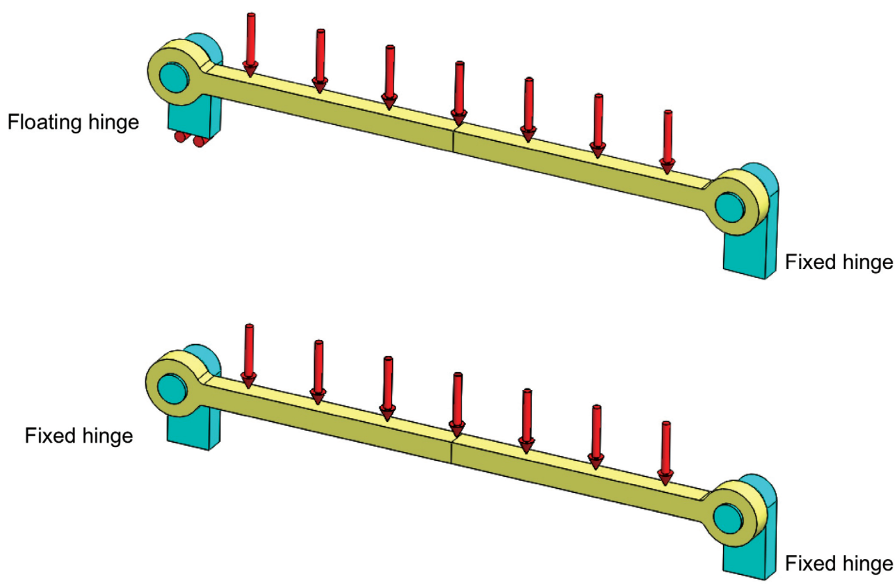


Figure 7.16 The configuration with floating hinge (top) may be analyzed with linear analysis if displacements are small and horizontal displacement of the floating hinge (bottom) is not of interest. If horizontal displacement of the floating hinge is not of interest, then nonlinear geometry analysis is required. The configuration with both hinges in fixed position always requires nonlinear geometry analysis.

Configuration with one floating hinge may be solved with linear analysis if displacements are small and horizontal displacement of the floating hinge is not of interest. However, if the horizontal displacement of the floating hinge is of interest, then finding it requires nonlinear geometry analysis. This is because the horizontal displacement is caused by model deformation that happens during the process of load application.

Both linear and nonlinear analyses do not have to be run on assembly; they may be conducted on the part model as shown in Figure 7.17.

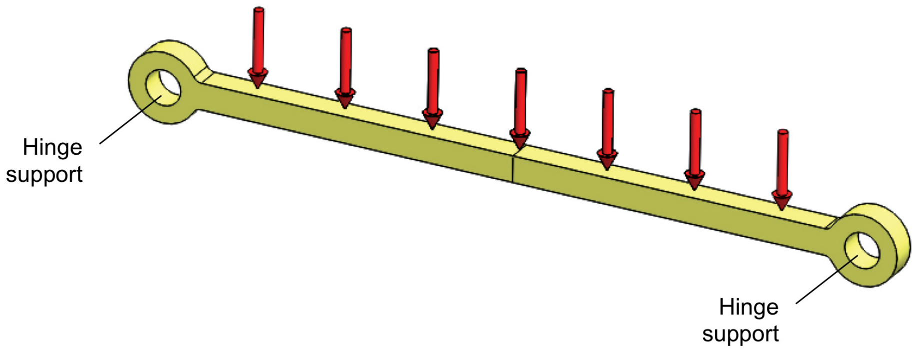


Figure 7.17 Analysis of the assembly in Figure 7.16 may be simplified to analysis of a part. If the above link is analyzed with linear analysis, then results are valid for configuration with one floating hinge. If it is analyzed with a nonlinear geometry analysis, then results are valid for configurations with both hinges in fixed positions.

Another problem similar to the hinge supported link (Figure 7.16) is shown in Figure 7.18. The wide end of the beam is held in a fixed position; the narrow end (shown in blue) can slide in the horizontal position. A pressure is applied to the top face. What type of analysis is required to find displacements and stresses? Should it be linear analysis or nonlinear geometry analysis? Let us review the results of both analyses.

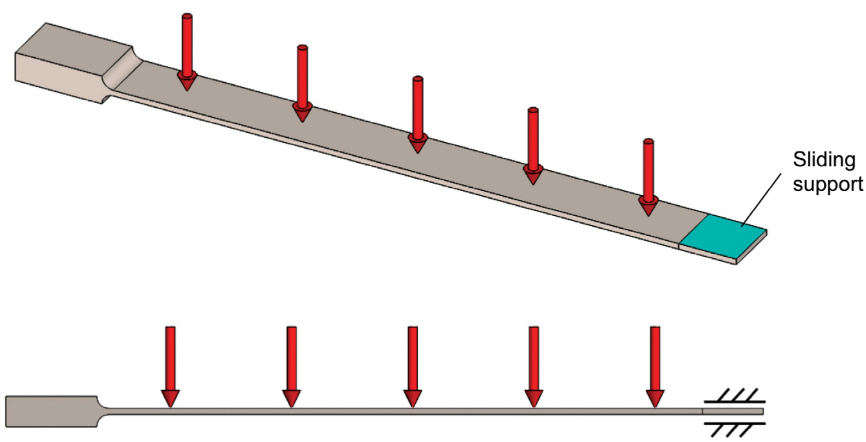


Figure 7.18 A thin beam under pressure has one end fixed; the other end may slide in the horizontal direction.

The displacement plots in 1:1 scale found in linear and nonlinear analyses are shown in Figure 7.19. The maximum displacement that takes place in the direction of the load is the same in both analyses. Displacements are small, and therefore, if the maximum displacement is all we are interested in, then linear analysis suffices.

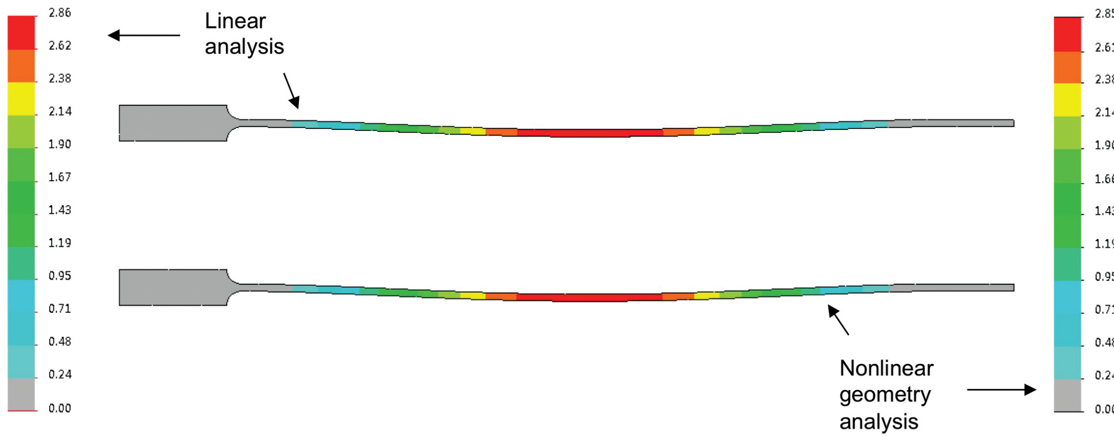


Figure 7.19 Linear displacement analysis (top) and nonlinear geometry analysis (bottom) both report the same resultant displacements. If displacement in the direction of load is all what we need to find, then linear analysis may be used.

Figure 7.20 shows the displacement plots in 10:1 scale. This exaggerated scale of deformation reveals that in linear analysis, the sliding end does not slide at all. In nonlinear geometry analysis, the sliding is correctly modeled. Therefore, if displacements of sliding end are of interest, then this problem requires nonlinear geometry analysis.

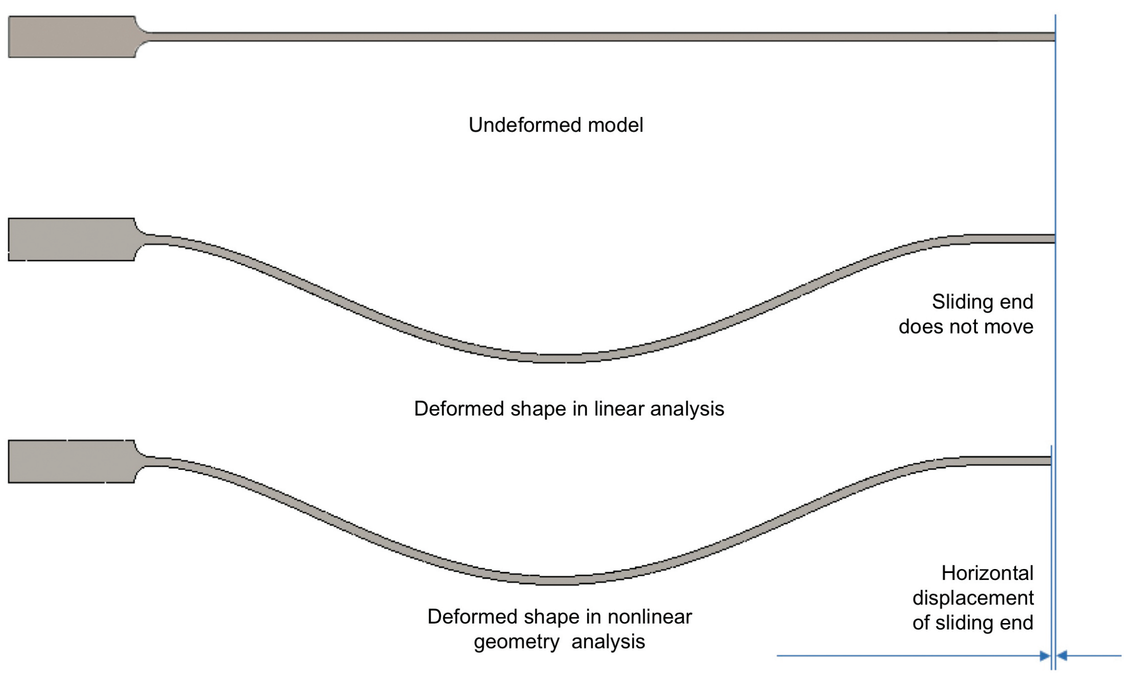


Figure 7.20 Deformed shape of the beam found by linear analysis (top) and nonlinear geometry analysis (bottom). Linear analysis does not show any displacements of the sliding end. Nonlinear analysis shows displacements of the sliding end. The scale of deformation is 10:1.

It is difficult to establish a "rule of thumb" that would say when it is OK to run linear analysis and when nonlinear geometry analysis is required. A commonly used threshold value of displacements not exceeding 5% of a characteristic model dimension can only be considered as a rough guidance.

The magnitude of displacement does not differentiate between linear geometry and nonlinear geometry analysis. What does differentiate between linear geometry and nonlinear geometry analysis is whether or not displacements cause significantly change stiffness.

For summary, let's clearly differentiate between terms "large displacement analysis" and "nonlinear geometry analysis". These two terms are often confused because in some commercial FEA programs nonlinear geometry analysis is called large displacement analysis.

Large displacement analysis is a sub-category of nonlinear geometry analysis and applies to cases where stiffness changes because displacements are large. However, we need to remember that displacements don't have to be large to cause significant change of stiffness as illustrated by membrane stress stiffening.

7.4 Contact

Analysis of contact is another type of nonlinear geometry analysis. The source of nonlinear behavior is the change of contact area, and consequently, of contact stiffness with the load magnitude. Even though displacements may be very small in comparison with the overall model size, they significantly change the model stiffness and, therefore, require a nonlinear treatment.

Most often, analysis of contact between two or more bodies is conducted to find contact stresses. Proper meshing is very important modeling consideration in contact stress analysis, but often it is difficult to satisfy. For accurate stress results, elements in and around the contact zone must be sized accordingly to the expected size of contact area. However, automeshers size the mesh accordingly to geometric features and not accordingly to the size of contact area which is unknown when meshing is created. Therefore, automatically generated mesh is rarely adequate for contact stress analysis. Mesh controls must be used to assure that elements are small enough to be able to model contact stress distribution properly. We will illustrate this with an example of contact stress analysis between two cylindrical faces of a clamp shown in Figure 7.21. There is a small gap between those faces; it closes under the applied load.

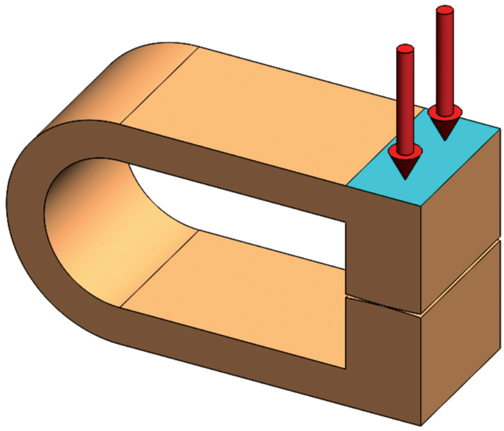


Figure 7.21 The load applied to the top face closes the gap between two cylindrical faces where contact stresses develop. The bottom face of the clamp is restrained.

A mesh with the default element size does not come even close to correct modeling of contact stress. As shown in Figure 7.22, contact is modeled as a line with several nodes along its length. A very fine mesh is required to model contact stresses correctly. In this case, it is easiest to accomplish with a 2D representation as shown in Figure 7.23. The 2D plane strain element mesh is shown in Figure 7.24 and the contact stress results in Figure 7.25.

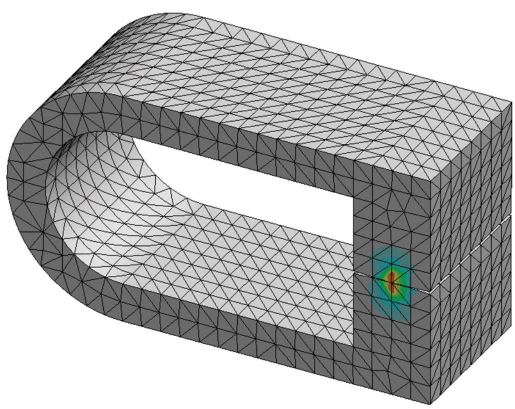


Figure 7.22 A mesh using default element size is totally inappropriate to model contract stresses between two cylindrical faces. Notice that when using these large elements, the contact is incorrectly modeled as a line contact between two rows of nodes.

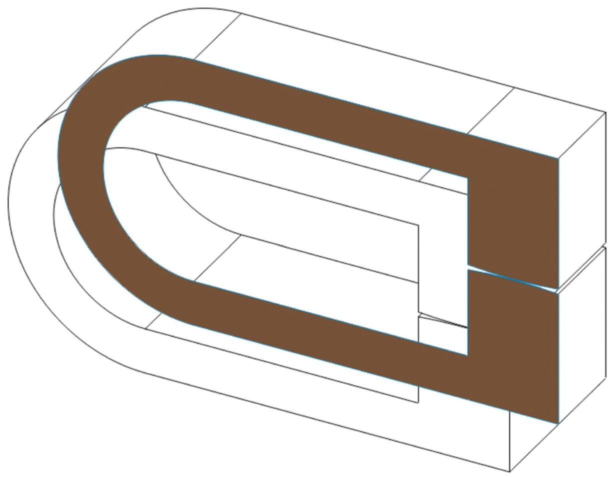


Figure 7.23 Analysis of contact stresses requires a very refined mesh in the area of contact. In this model, a highly refined mesh is easy to use when a 2D analysis is implemented. The illustration shows a 2D slice in the plane of symmetry of the model. This slice will be meshed with 2D plane strain elements.

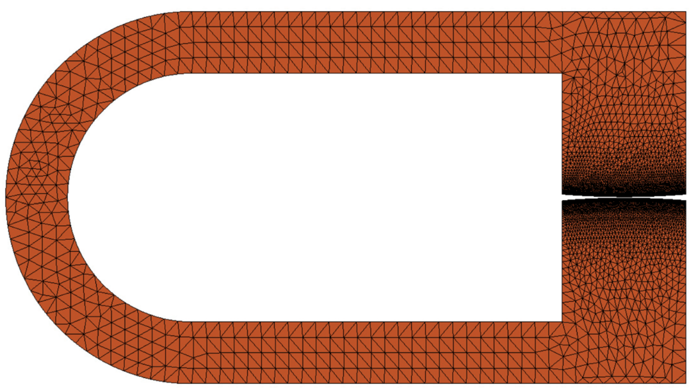


Figure 7.24 A 2D plane strain element mesh with highly refined elements in the contact zone. A 2D analysis is numerically efficient making it easy to solve large models.

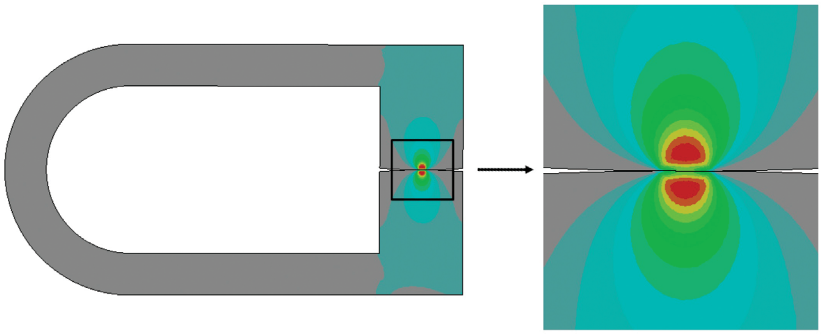


Figure 7.25 Von Mises stress results produced by the 2D plane strain model (left). Correct modeling of contact stresses in a very small region of the model (right) is possible using very small elements.

Model in Figure 7.21 has a very small gap, the upper arm does not have to move a lot to close the gap; therefore, a large displacement analysis is not required. Another clamp, shown in Figure 7.26, does require large displacement analysis or else the gap closes incorrectly. This problem combines two types of nonlinearities: large displacement and contact.

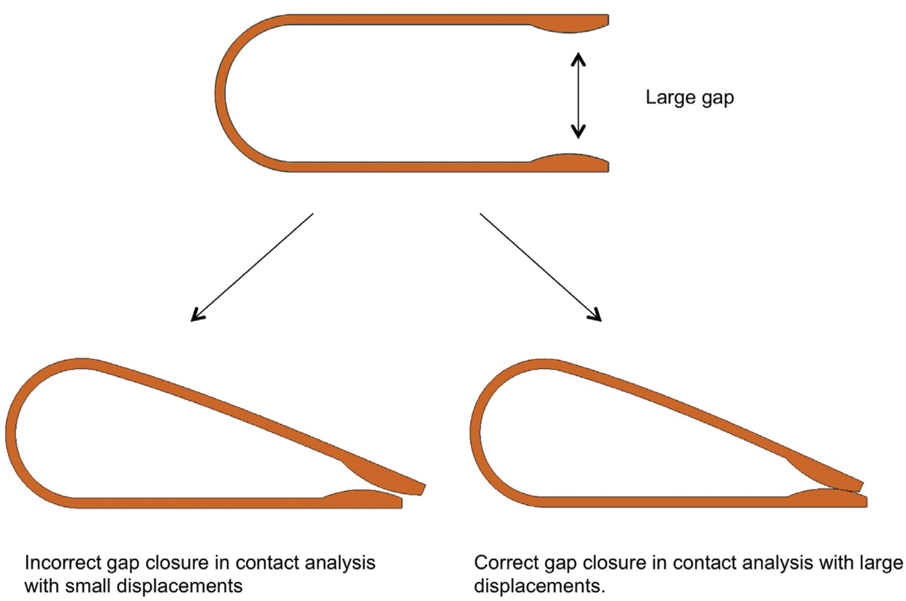


Figure 7.26 Closing a large gap requires nonlinear geometry analysis; this problem combines two types on nonlinearity: contact and large displacements.

Another category of contact problem is shown in Figure 7.27; an oversized ring is assembled into a cylinder. The red ring shrinks and the gray cylinder expands to make the assembly possible.

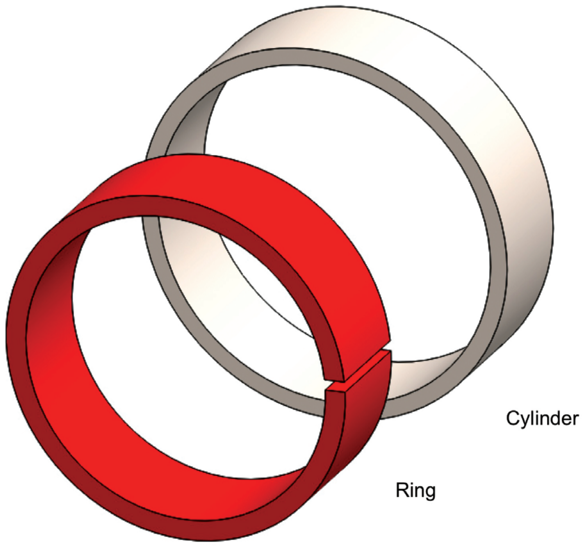


Figure 7.27 An oversized ring (red) is installed in a cylinder (gray); as a result of this, press fit contact stresses develop on touching faces of the ring and the cylinder.

Figure 7.28 shows model geometry before and after solution; it can be seen that the initial interference between the assembly components has been eliminated. The ring is held in its deformed shape by contact pressure that develops between two contacting faces (Figure 7.29).

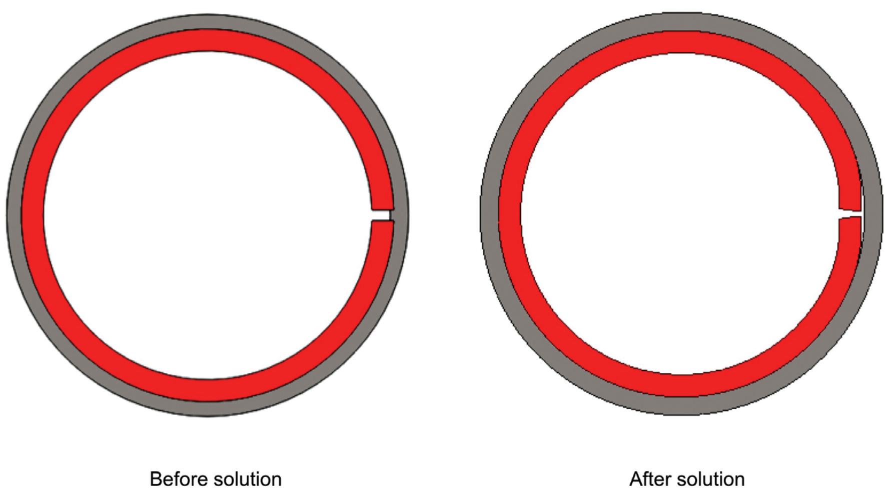


Figure 7.28 The model before and after solution; notice that the interference between the ring and the cylinder in "before solution" geometry has been eliminated in "after solution" geometry. Also notice gaps between contacting faces near the cut in ring in "after solution" geometry.

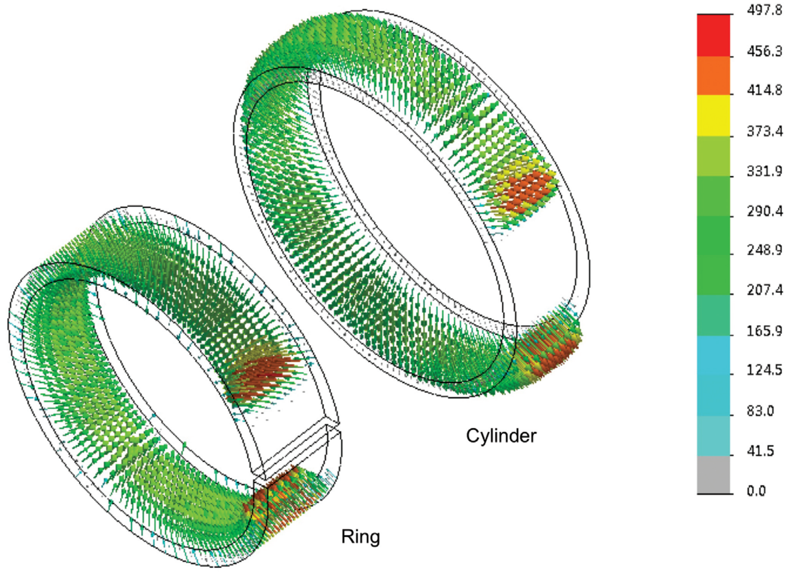


Figure 7.29 The plot of contact pressure on the faces of the ring and the cylinder. The contact pressure plot is shown as a vector plot.

Comment: Chapter 7 deals with nonlinear geometry, not with non linear material. In some exercises you will see stress well above yield but no yielding will be modeled because all these exercises use linear material model.

7.5 Hands-On Exercises

7.5.1 Cantilever Beam (Figure 7.1)

Model name

- 7.01.CANTILEVER_BEAM.x_t
- 7.01.CANTILEVER_BEAM.sldprt

7.5.1.1 Objective: Demonstrate the differences between incorrect linear solution and two nonlinear geometry solutions: with the following load and the nonfollowing load.

The following are the required steps:

1. Assign the material properties: Nylon 6/10.
2. Apply restraints: fixed to one end.
3. Apply load: 0.03 MPa to the top face.
4. Mesh with second-order solid elements, at least two elements across the beam height.
5. Find a linear solution.
6. Find a nonlinear geometry (often called large displacement) solution with the following load.

7. Find the nonlinear large displacement solution with the nonfollowing load.
8. Compare the displacements and stress results from the above three solutions.

7.5.2 Torsion Shaft (Figure 7.7)

Model name

- 7.02.TORSION_SHAFT.x_t
- 7.02.TORSION_SHAFT.sldprt

7.5.2.1 Objective: Demonstrate the differences between incorrect linear solution and nonlinear geometry (large displacements) solution with the following load.

7.5.2.2 Comment: Examine the CAD model and notice that the round shaft is much longer than how it appears in Figure 7.7. A long shaft has low stiffness in torsion and this produces large displacements. Notice that linear results show the lug stretching and shaft diameter "swelling."

The following are the required steps:

1. Assign the material properties: steel.
2. Apply restraints: fixed to the end of shaft.
3. Apply load: 3500 N to square holes as shown in Figure 7.7.
4. Mesh with second-order solid elements, at least four elements across the shaft diameter; use mesh control on the fillet to control the turn angle.
5. Find a linear solution.
6. Find a nonlinear large displacement solution with the following load.
7. Compare the displacements and stress results from the above solutions.

7.5.3 Round Plate (Figure 7.12)

Model name

- 7.03.ROUND_PLATE.x_t
- 7.03.ROUND_PLATE.sldprt

7.5.3.1 Objective: Demonstrate the nonlinear effect of membrane stress stiffening

7.5.3.2 Comments: This problem can be solved in a number of different ways. It can be solved "as is" by meshing it with solid elements. It can also be solved with shell elements by meshing either top or bottom round face. In either case the complete model may be analyzed or an arbitrary section with proper boundary conditions along the radial cuts. Finally, this problem may also be analyzed using 2D axi-symmetric model. The following assumes that model is analyzed "as is."

The following are the required steps:

1. Assign the material properties: steel.
2. Apply restraints: fixed to the cylindrical face along the circumference.

3. Apply load: 0.3 MPa to top face.
4. Mesh with second-order solid elements, at least one layer of correctly shaped elements across the plate thickness.
5. Find a linear solution.
6. Find a nonlinear geometry (large displacement) solution with the following load.
7. Compare the displacements and stress results from the above solutions.

7.5.4 LINK (Figure 7.17)

Model name

- 7.04.LINK.x_t
- 7.04.LINK.sldprt

7.5.4.1 Objective: Demonstrate that linear analysis misses the development of tensile stress in the link and incorrectly models stiffness.

7.5.4.2 Comment: Link with two hinges in fixed positions is analyzed (Figure 7.16 bottom). This problem could be analyzed using ½ of link with symmetry boundary conditions.

The following are the required steps:

1. Assign the material properties: ABS plastic.
2. Apply restraints: fixed hinge support to cylindrical holes on both sides.
3. Apply load: 300-N uniformly distributed to top face.
4. Mesh with second-order solid elements; four layers across the link height.
5. Find a linear solution.
6. Find a nonlinear geometry (large displacement) solution with the following load.
7. Compare the displacements and stress results from the above solutions.

7.5.5 Sliding Support (Figure 7.18)

Model name

- 7.05.SLIDING_SUPPORT.x_t
- 7.05.SLIDING_SUPPORT.sldprt

7.5.5.1 Objective: Demonstrate that linear analysis does not model displacement of the sliding end.

7.5.5.2 Comment: You may want to run linear analysis twice: first with sliding end, then with fixed end. You will see that results of these two linear analyses are the same. This problem could be analyzed using shell elements even though it would not be possible to model the wide end. 2D plane stress or plane strain representation could be also used to demonstrate that the sliding end doesn't actually slide in linear analysis.

The following are the required steps:

1. Assign the material properties: steel.
2. Apply restraints: fixed support to the wide end; sliding support to the narrow end.
3. Apply load: 0.1MPa uniformly distributed pressure to top face.
4. Mesh with second-order solid elements, one layer of correctly shaped elements across the beam thickness.
5. Find a linear solution; notice that the sliding does not move.
6. Find a nonlinear large displacement solution with the following load and observe displacements of the sliding end.

7.5.6 CLAMP01 (Figure 7.21)

Model name

- 7.06.CLAMP01.x_t
- 7.06.CLAMP01.sldprt

7.5.6.1 Objective: Analyze gap closure and contact stresses.

7.5.6.2 Comments: The 2D plane strain model is used here because it offers significant reduction of time required for solving. This problem could be used using solid elements as long as sufficiently small elements are used and long meshing and long solution times are acceptable.

The following are the required steps:

1. Assign the material properties: steel.
2. Use 2D plane strain simplification.
3. Apply restraints: fixed support to the bottom face.
4. Apply load: a 15000-N uniformly distributed pressure to the split face as shown in Figure 7.21.
5. Define no penetration contact between two cylindrical faces that are represented as lines in the 2D model.
6. Define mesh control along two edges that will come in contact; the element size is 0.05 mm.
7. Mesh with 2D plane strain elements.
8. Find a nonlinear small displacement solution and analyze contact stresses.

7.5.7 CLAMP02 (Figure 7.26)

Model name

- 7.07.CLAMP02.x_t
- 7.07.CLAMP02.sldprt

7.5.7.1 Objective: Demonstrate a problem that combines two types of nonlinearities: large displacements and contact. Contact stress analysis is not the objective.

7.5.7.2 Comment: Because contact stresses do not have to be analyzed, mesh may be less refined compared with CLAMP01 problem. This make is practical to use solid elements.

The following are the required steps:

1. Assign the material properties: steel.
2. Apply restraints: fixed support to the bottom face.
3. Apply load: a 200-N load to the top split face as shown in Figure 7.26.
4. Define no penetration contact between two cylindrical faces.
5. Mesh with solid elements default size; this exercise is not intended for analysis of contact stresses.
6. Find a small displacement solution and a large displacement solution and compare deformed shapes in 1:1 scale of deformation.

7.5.8 Shrink Fit (Figure 7.27)

Model name

- 7.08.SHRINK_FIT.x_t
- 7.08.SHRINK_FIT.sldasm

7.5.8.1 Objective: Demonstrate contact analysis with initial interference, which is eliminated during the nonlinear solution. Analyze contact stresses between assembly components.

7.5.8.2 Comment: This problem can be solved using solid elements or 2D elements. If you use 2D representation, compare plane strain and plane stress solutions to the solution obtained with solid elements. The following assumes the use of solid elements.

The following are the required steps:

1. Assign the material properties: steel.
2. Apply restraints: fixed support to the outside face of cylinder; restrain circumferential and axial displacements of the inside face of the ring using a local cylindrical coordinate system.
3. Define shrink fit contact between the faces of the ring and the cylinder that will be touching.
4. Mesh with solid elements, at least two layers of second-order elements across the ring and cylinder thicknesses.
5. Find solution; review the deformed shape in 1:1 scale of deformation.
6. Analyze contact stresses between the ring and the cylinder.

Chapter 8

Nonlinear Material Analysis

8.1 Review of Nonlinear Material Models

A linear material model is defined by only two parameters: the modulus of elasticity, which establishes relationship between strain and stress, and Poisson's ratio which is the negative ratio of transverse to axial strain. The linear relationship between strain and stress demonstrates that the modulus of elasticity does not change with strain (Figure 8.1).

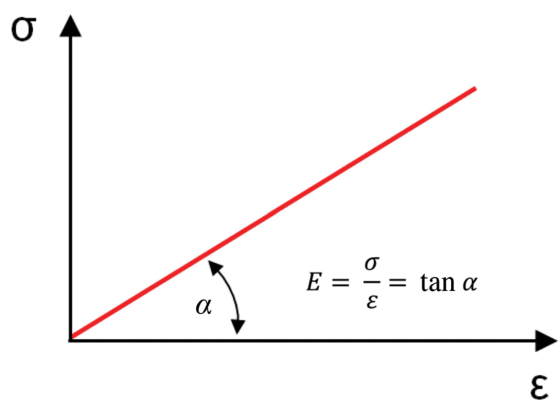
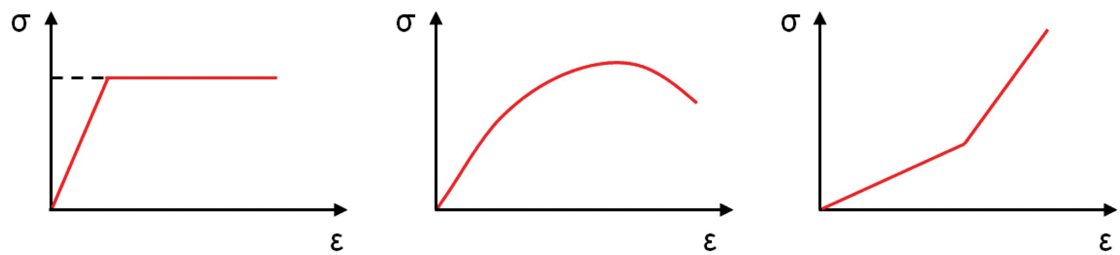


Figure 8.1 Strain–stress curve of a linear material model. Stress is a linear function of strain.

A nonlinear material model is one that does not follow a linear relationship between strain and stress and, therefore, does not have a constant modulus of elasticity. Using such a material, model stiffness changes during the loading process and stiffness matrix must be recalculated during the solution process. The nonlinear material requires a more complex material property definition that depends on what a strain–stress relationship is assumed in the material model.

A large number of nonlinear material models exist; nonlinear material models include elastic-perfectly plastic material (Figure 8.2(a)), nonlinear material with softening (Figure 8.2(b)), bilinear material with hardening (Figure 8.2(c)) and many more.



a. Elastic–perfectly plastic   b. Nonlinear with softening   c. Bilinear with hardening

Figure 8.2 Strain–stress curves of the selected nonlinear material models.

8.2 Elastic–Perfectly Plastic Material Model

The simplest nonlinear material model is the elastic–perfectly plastic material model. Because of its simplicity it is commonly used and we will limit our discussion to this type of nonlinear material only.

The elastic–perfectly plastic material model assumes a linear relation between strain and stress up to a certain threshold level of stress called yield strength. Above the yield point, stress remains constant regardless of the strain level (Figure 8.2(a)). Compared with the linear material model, the only additional piece of information required is the yield strength, usually defined in terms of von Mises stress.

A bracket made out of aluminum with a yield strength of 27.6 MPa is shown in Figure 8.3. A linear analysis was run first; a plot of von Mises stresses is shown in Figure 8.4. Large portions of the model show stresses above the yield strength.

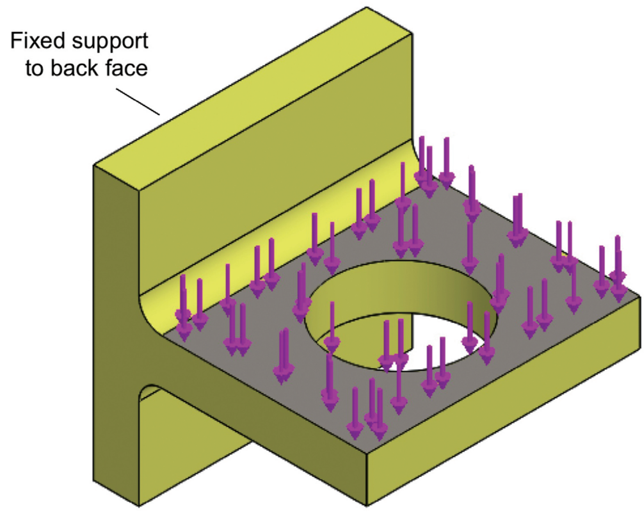


Figure 8.3 A bracket loaded with a uniformly distributed load to the top (gray) face and supported at the back face.

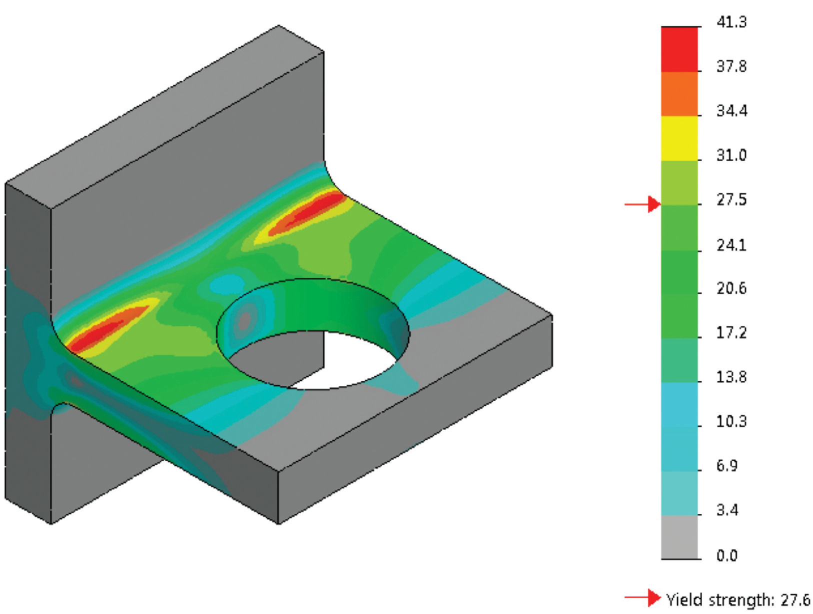


Figure 8.4 A von Mises stress plot produced with a linear material model. The red arrow on the color legend indicates the yield strength of material. Large portions of the model show stresses above the yield strength of 27.6 MPa.

The next analysis used an elastic–perfectly plastic material model and provided results shown in Figure 8.5. Notice that results based on the elastic–perfectly plastic material model show the maximum stress equal to the yield strength of the material. A plasticized material now occupies almost entire cross section of cantilever portion of the

bracket. Base of this finding we may conclude that the applied load is very close to the maximum load that model is able to take. A further increase in the load magnitude would cause complete plasticization of the cross section in bending and the model would become a mechanism (Figure 8.6).

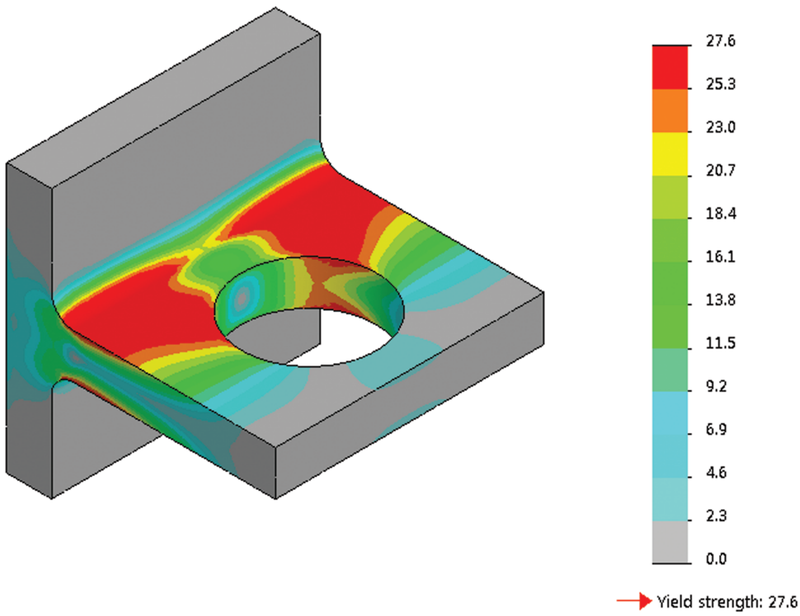


Figure 8.5 A von Mises stress plot produced using an elastic–perfectly plastic material model. The maximum von Mises stress is limited to 27.6 MPa, which corresponds to the flat portion of the strain–stress curve in the material model.

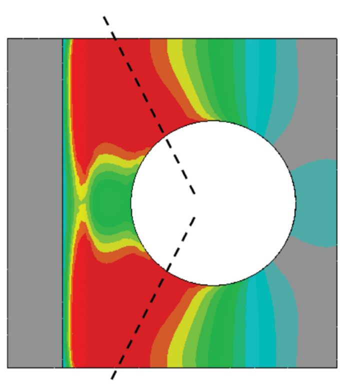


Figure 8.6 The dotted lines schematically show the location of plastic hinges that will develop when the maximum load bearing capacity of the bracket is exceeded. This applies only to analysis that uses an elastic–perfectly plastic material. When a linear material model is used, an increase in load magnitude will produce larger displacements, larger strain, and larger stress, all with no bound.

8.3 Use of Nonlinear Material to Control Stress Singularity

The use of an elastic–perfectly plastic material can eliminate stress singularities like those accompanying sharp reentrant edges. Using an elastic–perfectly plastic material places an upper bound on the stress level; the highest stress equals the yield strength as defined in the material model.

Consider an L bracket shown in Figure 8.7. If it is analyzed with a linear material model, the maximum stress diverges with mesh refinement because of stress singularity cause by the sharp re-entrant edge.

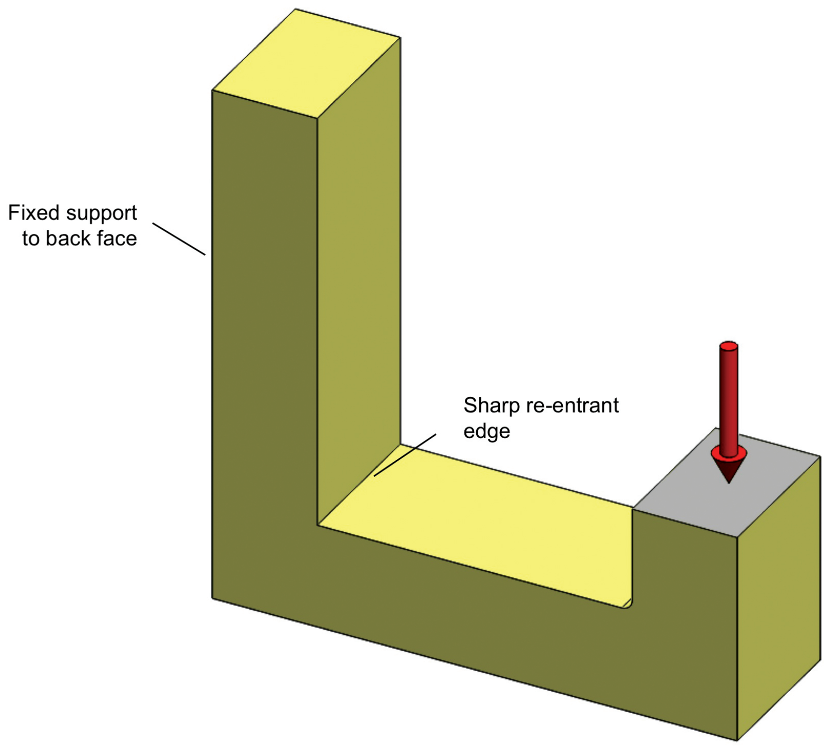


Figure 8.7 A bracket loaded with uniformly distributed load to the top (gray) face and supported at the back face. Note that a sharp re-entrant edge is part of the model geometry.

The maximum von Mises stress of 567.7 MPa is meaningless; this result is completely dependent on the element size.

An attempt to truncate stress magnitude found in linear analysis at the yield strength leads to erroneous results as shown in Figure 8.8.

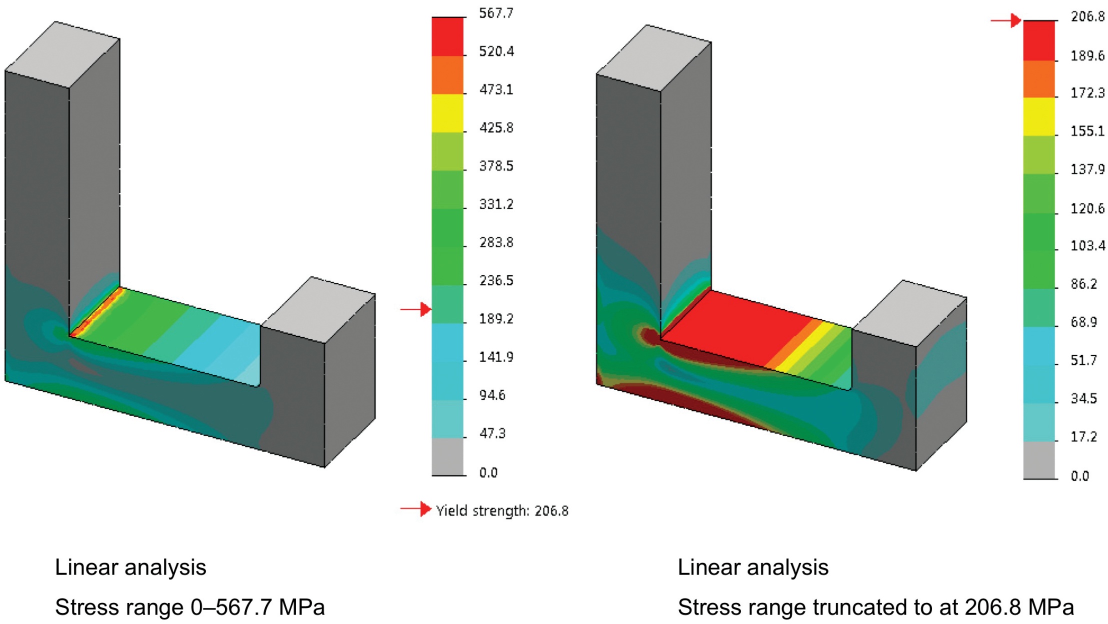
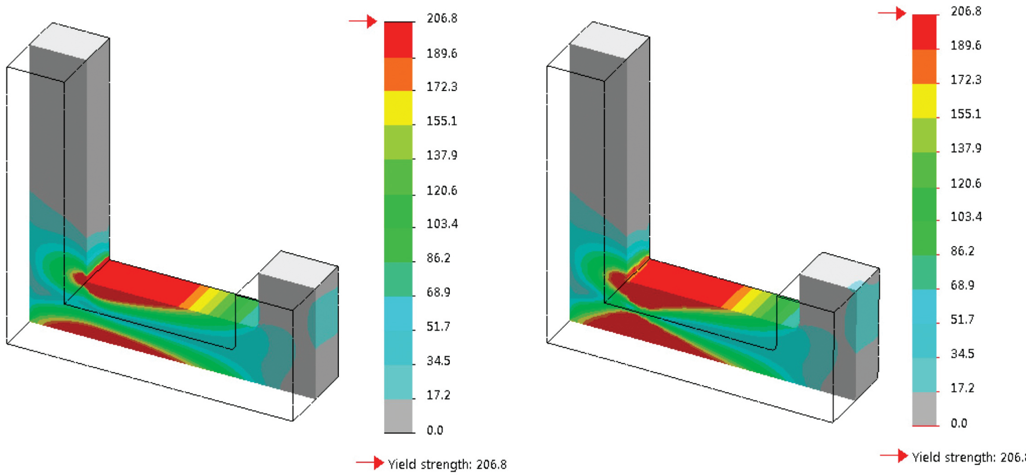


Figure 8.8 The von Mises stress in the model with a linear material. Left: the maximum stress of 568 MPa in the sharp re-entrant edge is meaningless because of stress singularity. Right: the range of stress magnitude truncated at 206.8 MPa, which is the yield strength of the model material.

The same L bracket analyzed with the elastic–perfectly plastic material does not show stress singularity. Mesh refinement would not produce divergent stresses because the maximum stress is capped at 206.8 MPa, which is the yield strength of the elastic–perfectly material (Figure 8.9). It is important to point out that while stress is capped at the magnitude of the yield strength of the model's material, strain does diverge with mesh refinement.



Linear analysis

Stress range truncated to at 206.8 MPa

Nonlinear material analysis

The maximum stress is 206.8 MPa which is the yield strength of elastic–perfectly plastic material

Figure 8.9 Stresses in the mid-plane of the model. Left: a repetition of plot in Figure 8.8. It is supposed to show the extend of yield, that is, where the von Mises stress exceeds 206.8 MPa this is why the stress magnitude is truncated at 206.8 MPa. However, the linear material model cannot be used to study the extend of plasticity zone.

Right: the results of nonlinear material analysis correctly display the extend of plasticity based on the elastic–perfectly plastic material. The maximum stress is 206.8 MPa and the stress range is not truncated; by definition of the elastic–perfectly plastic material used in this model, the highest stress cannot exceed 206.8 MPa. Notice that two zones of plasticized material almost meet; the bracket is very close to structural collapse.

8.4 Other Types of Nonlinearities

Geometric nonlinearities were discussed in chapter 7, and material nonlinearities were discussed in this chapter. There is one more important type of nonlinearity: nonlinear buckling, which is presented in chapter 10. For better understanding of nonlinear buckling analysis, first we have to review modal analysis presented in chapter 9.

8.5 Hands-On Exercises

8.5.1 BRACKET NL (Figure 8.3)

Model name

- 8.01.BRACKET_NL.x_t
- 8.01.BRACKET_NL.sldprt

8.5.1.1 Objective: Conduct analysis with the elastic–perfectly plastic material model; study the structural collapse caused by material yielding.

8.5.1.2 Comments: Once the analysis is completed, double the original load of 5000N and run it again. Load will be applied in increments as before but it would not reach the maximum load, which is now 10000 N. The last successfully performed step corresponds to the last load increment that does not yet cause plastification of the cantilever portion of the model (Figure 8.6). The next load increment causes complete plastification of the cross section. Once that happens, a plastic hinge develops causing a rigid body motion in the model. That in turn, causes solution to crash.

The following are the required steps:

1. Assign Material Properties: Elastic–perfectly plastic material; 1060 alloy (aluminum alloy) with a yield strength of 27.6 MPa.
2. Apply Restraints: Fixed to the back face.
3. Apply Load: 5000 N to the top face.
4. Mesh with second-order solid elements; adjust global element size to have four to five elements covering fillets (turn angle of ~20° or less)
5. Find a linear solution.
6. Find a nonlinear solution with an elastic–perfectly plastic material; a small displacement assumption may be used.
7. Compare the displacements and stress results from the above solutions.
8. Double the load magnitude and run nonlinear material analysis again. Observe a solution to crash.

8.5.2 L BRACKET (Figure 8.7)

Model name

- 8.02.L_BRACKET.x_t
- 8.02.L_BRACKET.sldprt

8.5.2.1 Objective: Demonstrate that the use of elastic–perfectly plastic material eliminates stress singularity.

8.5.2.2 Comments: Having completed this exercise, you may want to eliminate the stress singularity by adding a fillet. Then run nonlinear material analysis of the model with fillet and compare the results with those produced using the model with a sharp re-entrant edge.

The following are the required steps:

1. Assign Material Properties: Elastic–perfectly plastic material; AISI304 (steel) with a yield strength of 206.8 MPa.
2. Apply Restraints: Fixed to the top face (Figure 8.7).
3. Apply a 10000-N normal load to the gray face shown in Figure 8.7.
4. Mesh with second-order solid elements; element size: 2 mm or less.
5. Find a linear solution.
6. Find a nonlinear solution with an elastic–perfectly plastic material; a small displacement assumption may be used.
7. Compare the displacements and stress results from the above solutions.
8. Perform several mesh refinements in the nonlinear model; observe that the maximum stress does not change.

Chapter 9
Modal Analysis

9.1 Differences Between Modal and Static Analysis

Modal analysis, also called frequency analysis, finds natural frequencies and the shapes of vibration that are associated with those frequencies. Modes of vibration describe a body vibrating in the absence of damping and excitation forces. Even though any real-life structure has an infinite number of degrees of freedom, it still has discrete modes of vibration. Each mode, with its frequency and the associated modal shape, corresponds to the situation where stiffness forces cancel out with inertial forces.

We recall the fundamental finite-element method equation, which is applicable to static analysis:

$$[K] * [d] = [F] \tag{2.1}$$

To consider dynamic effects, it needs to be extended to account for inertial and damping effects and for the fact that load can be a function of time

$$[M]\ddot{d} + [C]\dot{d} + [K]d = [F(t)] \tag{9.1}$$

where

$[M]$—mass matrix: known

$[C]$—damping matrix: known

$[K]$—stiffness matrix: known

$[F]$—vector of nodal loads: known

$[d]$—vector of nodal displacements: unknown

Modal analysis deals with free and undamped vibrations where $[F(t)] = 0$ (no excitation force) and $[C] = 0$ (no damping). Therefore, (9.1) can be simplified to:

$$[M]\ddot{d} + [K]d = 0 \tag{9.2}$$

Finding nonzero solutions of (9.2) presents an eigenvalue problem; it provides with modal frequencies and associated modal shapes of vibration:

$$[K]\{\varphi\}_i = \omega_i^2[M]\{\varphi\}_i \tag{9.3}$$

Equation (9.3) has n solutions, where ω_i^2 is called the eigenvalue and the corresponding vector $\{\varphi\}_i$ is called the eigenvector. The relation between eigenvalue and frequency expressed in Hertz is

$$f_i = \frac{\omega_i}{2\pi} \tag{9.4}$$

9.2 Interpretation of Displacement and Stress Results in Modal Analysis

Referring to (9.2), we notice that if d is a solution satisfying (9.2), then an arbitrary number times d is also a solution. Therefore, modal analysis cannot provide any quantitative information on displacements or stresses.

FEA programs take different approaches to presenting displacements results in modal analysis. One approach is to normalize displacements in such a way as to make the generalized mass the identity matrix.

In another approach, displacements are normalized so that the maximum displacement reads 1 (of whatever unit of length is used). Some programs present those normalized displacement plots without any units in an attempt to avoid confusion with real displacements.

Not all programs provide users with stress results in modal analysis. If they do that, it is important to remember that stresses are calculated based on normalized displacements and have nothing to do with real stress.

Displacements calculated in modal analysis may be used only to compare relative displacements between different portions of the analyzed model; the same applies to stresses. Any comparisons can only be made in the qualitative sense and only within the same mode (Figure 9.1). Comparing the results from different modes makes no sense.

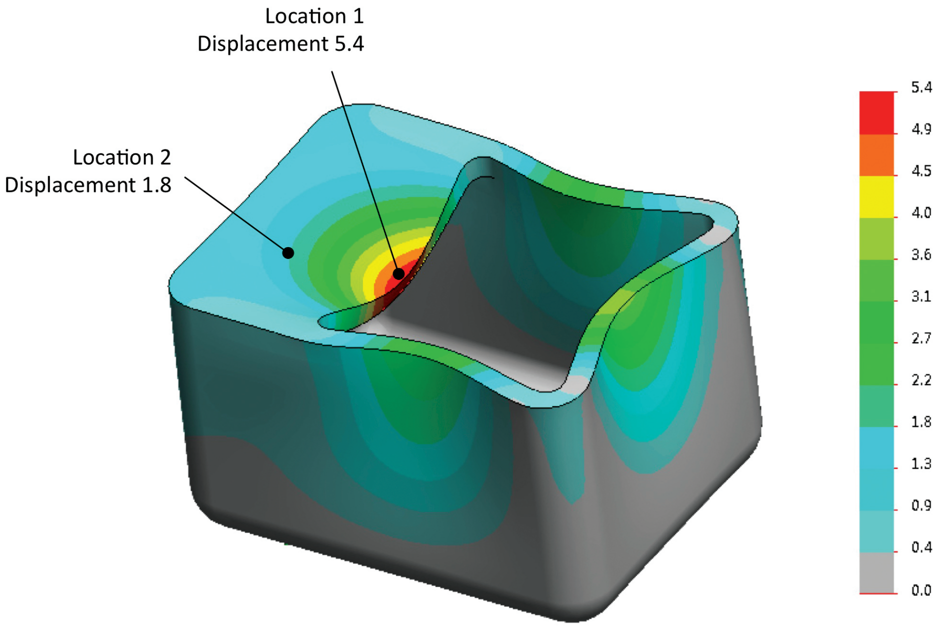


Figure 9.1 Normalized resultant displacement results in the first mode of vibration a box; the box has a fixed support applied to the bottom face; there are no RBMs. Notice that no units are given in the color legend. Different normalization schemes may be used depending on the program. The numerical values may be used only to compare relative displacements in different portions of the model. In this case, the ratio of resultant displacement in locations 1 and 2 is 5.4/1.8 = 3.

In real life, actual values of displacements, stresses, strain energy, etc. depend on the excitation force and damping, which are not considered in modal analysis.

9.3 Modal Analysis With Rigid Body Modes

A structural analysis requires a fully supported structure; modal analysis can be executed on a partially supported or unsupported model. As we remember from Chapter 1, such a model has Rigid Body Motions. Rigid Body Motions are represented in modal analysis as rigid body modes. Both Rigid Body Motions and Rigid Body Modes use the same acronym: RBM. An unsupported 3D model has six rigid body modes with zero frequency (or very close to zero because of discretization and solution errors) corresponding to six rigid body motions that are translations along three directions and rotations about three directions (Figure 9.2). The first mode of vibration that is associated with a deformation, we call it the first elastic mode, is mode number 7 (Figure 9.3).

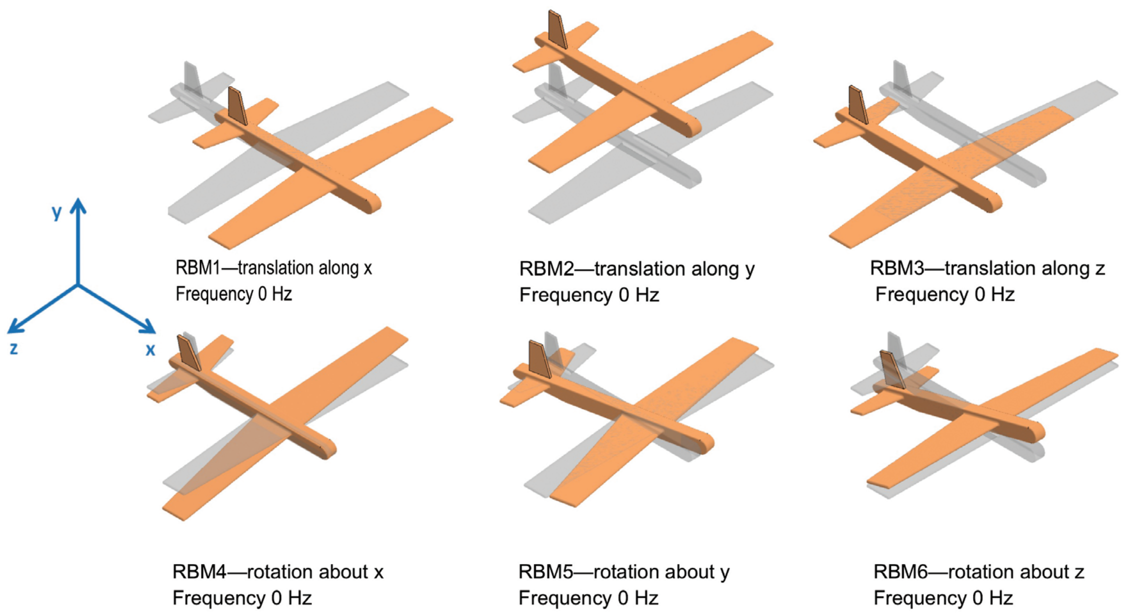


Figure 9.2 Model airplane with no supports has six rigid body motions; they are represented by six rigid body modes shown above. Three translations and three rotations. The model in the original position is shown in gray color.

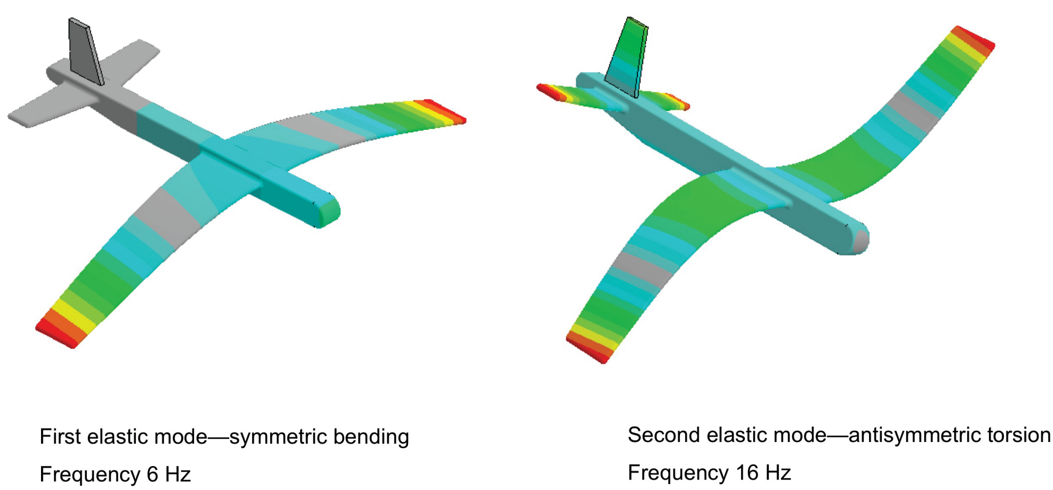


Figure 9.3 Elastic modes of an unsupported airplane model start with mode 7. Here, the first two elastic modes are shown.

An analysis of partially supported model will return as many zero frequency modes as many rigid body motions are found in the model. A ball joint assembly is shown in Figure 9.4. Assuming that the base is fully restrained, the ball has three rigid body

motions that are all rotations. Notice that the base is included in the assembly only for show. The analysis of rigid body modes of the ball may be conducted on part only by restraining the spherical surfaces of the ball in a spherical coordinate system. This will be done in one of the hands-on exercises in this chapter.

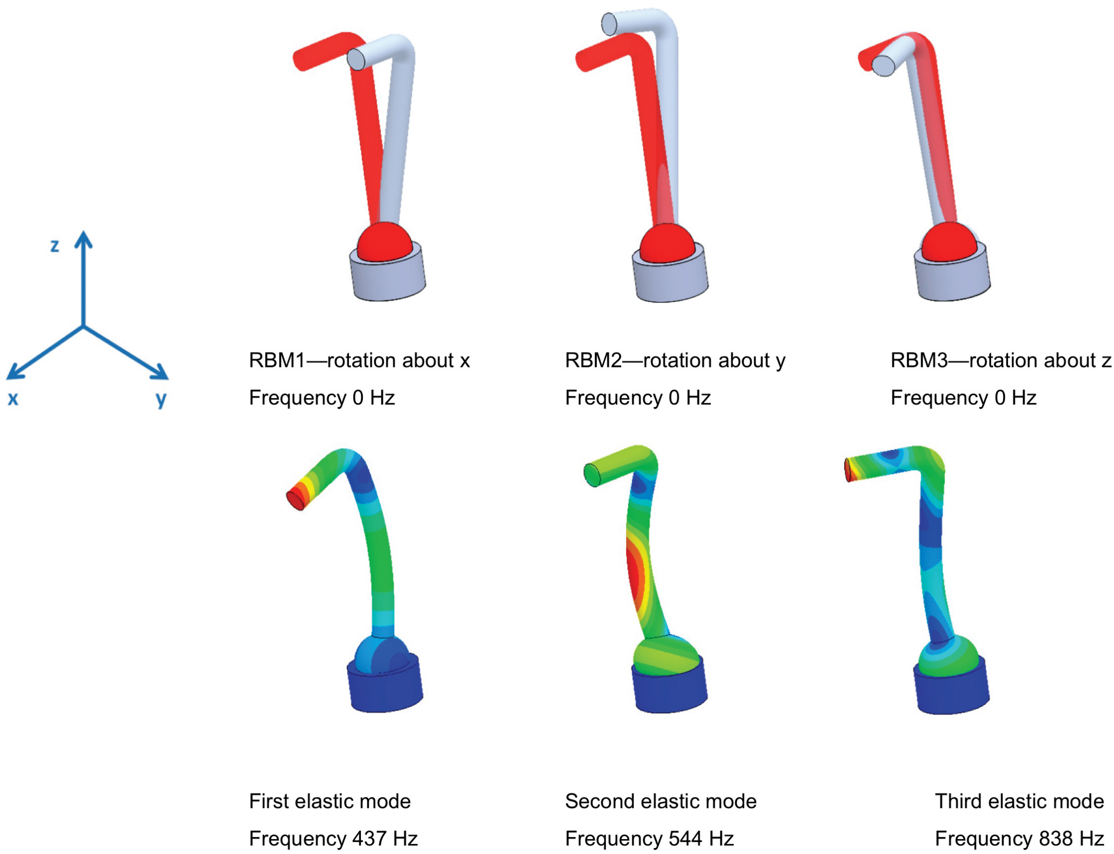


Figure 9.4 A ball joint has three RBMs, which are rotations around three axes; these RBMs are shown in the top row and undeformed shape (red) is superimposed onto the deformed shape. The first three elastic body modes are shown in the bottom row.

9.4 Importance of Supports in Modal Analysis

The natural frequencies strongly depend on supports. Most often, natural frequencies will increase with adding support because the added support will increase the structure stiffness. However, adding supports may also have the opposite effect: natural frequencies may decrease as illustrated in Figure 9.5. The unsupported model has six rigid body modes. The first elastic mode is mode 7; its shape demonstrates that the unsupported model vibrates about its center of mass. The hinge-supported model has no rigid body modes, the shape of the first elastic mode shows that the center of mass participates in vibration.

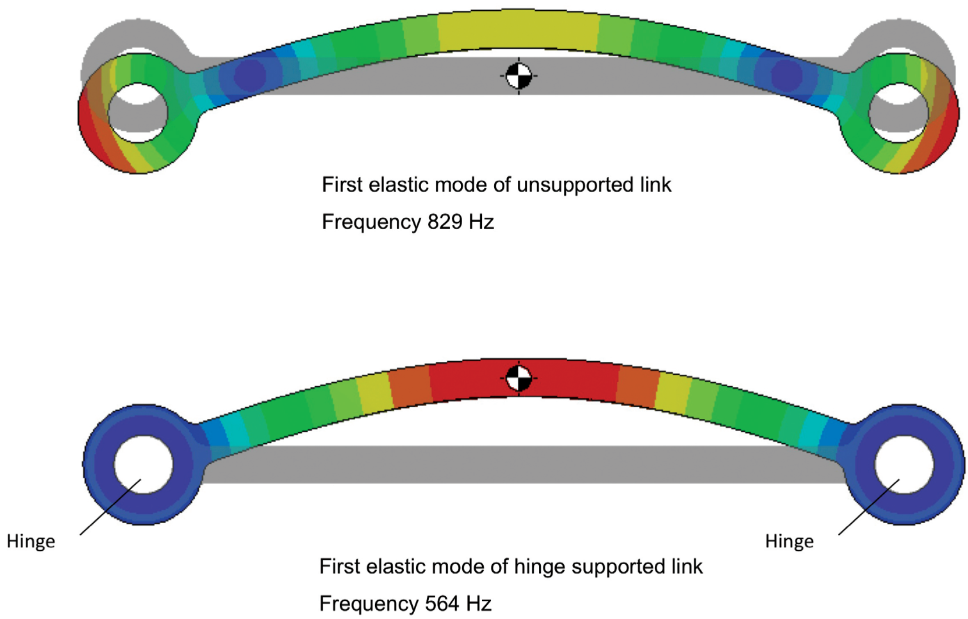


Figure 9.5 The first elastic mode of vibration of unsupported (top) and hinge supported link (bottom). The undeformed shape in overlaid on the deformed modal shape. In unsupported link, vibration takes place about the center of mass. In hinge supported link, the center of mass participates in vibration.

Even though the supported link has a higher stiffness, the effect of higher stiffness is offset by higher mass participating in vibration. The combined effect of higher stiffness and higher mass participation makes the natural frequency of the supported link lower than the frequency of unsupported link.

9.5 Applications of Modal Analysis

We will review important applications of modal analysis.

9.5.1 Finding Modal Frequencies and Associated Shapes of Vibration

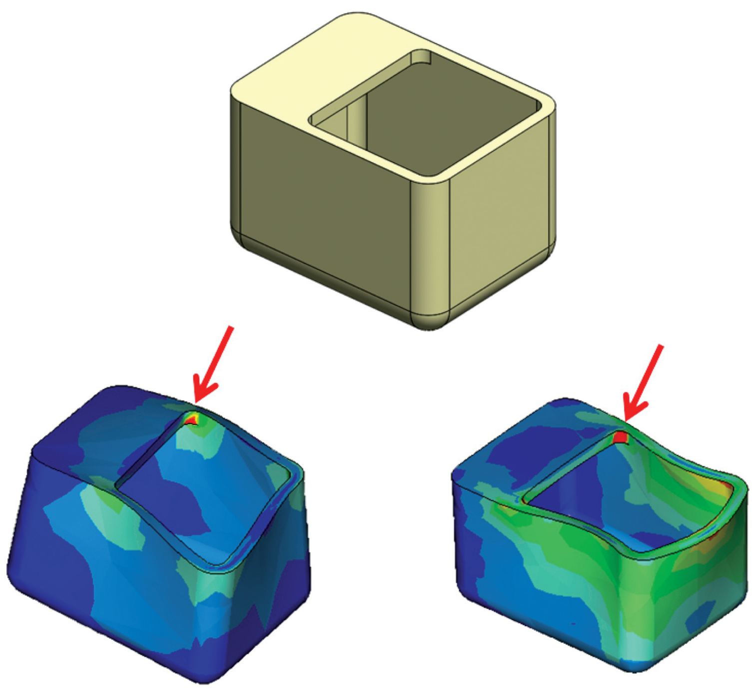
Natural frequencies are closely related to resonant frequencies. The natural frequencies and the associated modes are structural characteristics of an object. Resonance is what happens when that object is subjected to an excitation with frequency equal to the natural frequency and in the direction aligned with the modal shape. Most often, only the first few resonant frequencies are of interest.

Knowing natural frequencies and associated shapes is important in preventing unwanted vibration and in vibration analysis in general.

9.5.2 Locating "Weak Spots" in Structure

Displacement and stress results in modal analysis are meaningless in terms of numbers but provide important qualitative information. They may be used to review the distribution of stiffness and mass in the analyzed structure and to locate "elastic hinges," which are stiffer regions surrounded by relatively softer parts.

The distribution of stresses does provide valuable insight into the structural properties of the analyzed model. The location of highest stresses related to the first few modes highlight locations that are naturally predisposed to experience structural failure (Figure 9.6). This information can be used, for example, to decide where to place strain gages in experimental stress analysis.



Mode 1 Mode 2

Figure 9.6 The first two elastic modes of vibration of an unsupported box; these modes will be listed as modes 7 and 8 because the first six modes are rigid body modes. Von Mises stress is used to show stress concentrations that highlight potential "weak spots" as indicated by arrows. In this case, the first two modes highlight the same location. Other types of stress plots may also be used.

9.5.3 Modal Analysis Provides Input to Vibration Analysis

In addition to providing very important results on its own, modal analysis is a prerequisite to the modal superposition method used in linear vibration analysis. Vibration analysis based on the modal superposition method assumes that vibration response of a model can be represented as a superposition of responses of several single degree of freedom systems corresponding to the modes of vibrations considered in the modal superposition method.

9.6 Prestress Modal Analysis

Prestress modal analysis is also called preload modal analysis. These two terms are interchangeable.

The natural frequencies may significantly depend on the applied load if that load changes structure's stiffness. If this is the case, modal analysis needs to account for prestress. Predominantly tensile stresses will increase natural frequencies as is illustrated by tuning a guitar string or stress stiffening of rotating components like a turbine blade or a helicopter rotor. Rotating machinery requires considering the effect of prestress (Figure 9.7).

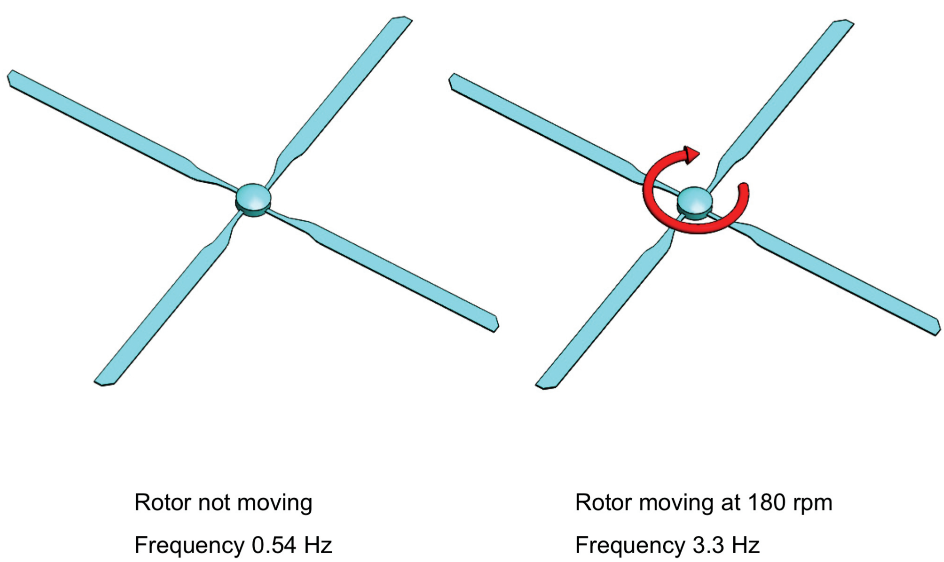
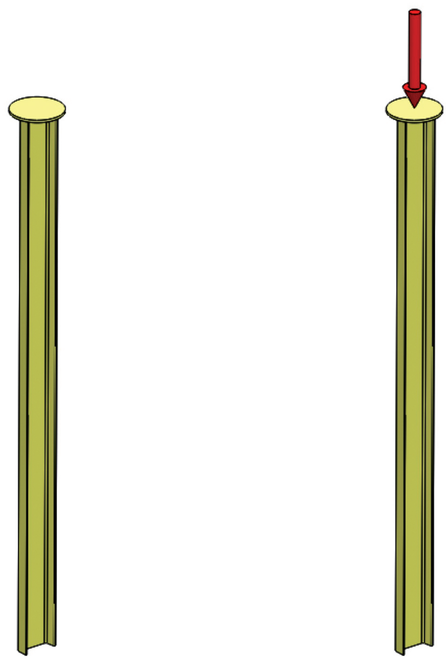


Figure 9.7 The modes of vibration of the helicopter blade are strongly influenced by tensile stresses produced by centrifugal force. Frequencies in the first mode and listed. The analysis may be done using only one blade.

Predominantly compressive stresses will decrease natural frequencies. For example, the analysis of natural frequencies of a compressed column demonstrates that the first natural frequency decreases with the load (Figure 9.8). It is interesting to notice that load magnitude corresponding to zero frequency equals the buckling load (Figure 9.9).



Unloaded column
Frequency 6.85 Hz

Column loaded with 87% of buckling load
Frequency 1.72 Hz

Figure 9.8 The frequency of column in the first mode is strongly influenced by compressive stresses produced by the applied compressive load.

A word of caution: it is a severe error to confuse the preload with excitation load. Remember that excitation load does not exist in modal analysis.

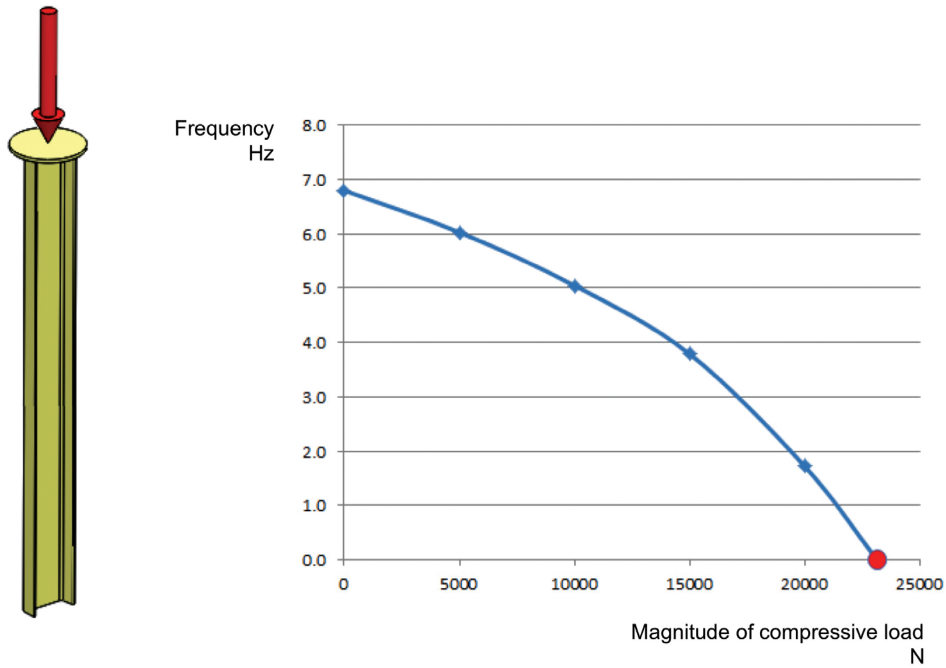


Figure 9.9 The first natural frequency of a column as a function of compressive load. Compressive load reduces the first natural frequency of a column. The magnitude of compressive load that brings the first natural frequency down to zero (red marker) is the buckling load.

9.7 Symmetry and Antisymmetry Boundary Conditions in Modal Analysis

Symmetry and antisymmetry boundary conditions are applicable to modal analysis but must be used with caution. Symmetry boundary conditions eliminate antisymmetric modes and antisymmetry boundary conditions eliminate symmetric modes. A one half of a symmetric model can be used to extract all modes of vibration if the results of modal analysis with symmetry boundary condition are combined with the results of modal analysis of the same model with antisymmetry boundary conditions; this is illustrated in Figure 9.10 and Table 9.1. The overhead required to manage two models and two sets of results is usually quite high and is worth the effort only for large models that could not be solved without cutting them into half.

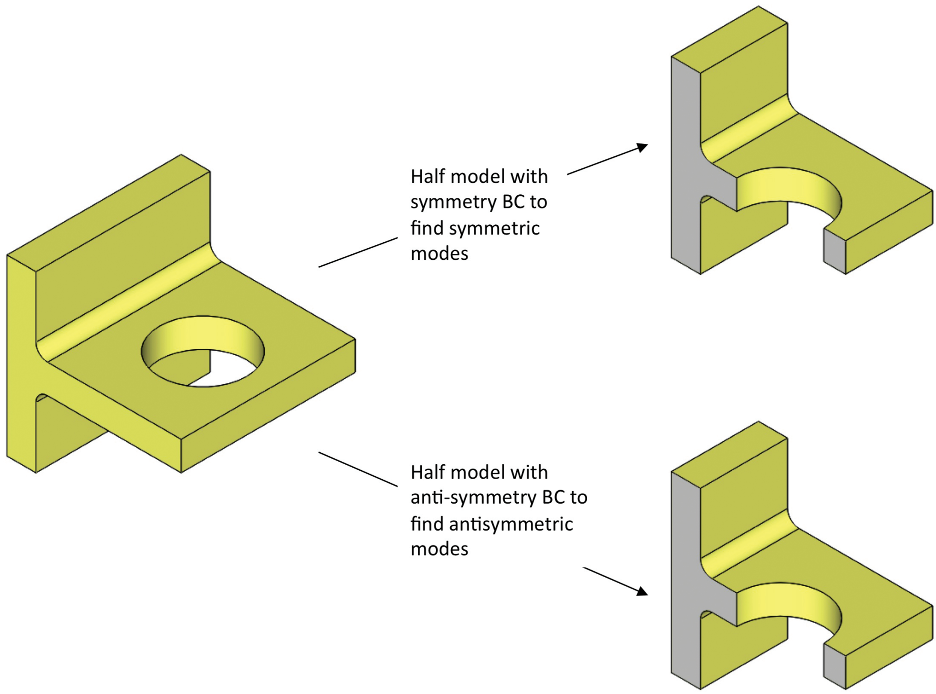


Figure 9.10 A half model can be used to extract all modes of vibration of a symmetric model, but this requires combining results of two modal analyses: with symmetry boundary conditions and antisymmetry boundary conditions.

Table 9.1 An example of modal results from the full model, half model with symmetry boundary conditions, and half model with antisymmetry boundary conditions. Combining results from both half models provides the complete set of modes, the same as reported by the full model

Mode number	Mode description	Full model (Hz)	Half model Symmetry boundary conditions(Hz)	Half model Anti symmetry boundary conditions (Hz)
1	Symmetric	412	412	
2	Antisymmetric	720		720
3	AntiSymmetric	1034		1034
4	Symmetric	1808	1814	
5	Symmetric	2359	2365	
6	Symmetric	2533	2538	

9.8 Convergence of Modal Frequencies

In chapter 4, we explained that during the convergence process, degrees of freedom are added to the model and, in effect, "artificial restraints" resulting from approximations imposed by meshing become less significant and this makes the finite element model softer. The same mechanism is at work when convergence process is conducted in modal analysis. Finite element model becomes slightly softer after each refinement making modal frequencies lower; consequently, modal frequencies converge from above to the asymptotic value. Therefore, the results of modal analysis overestimate modal frequencies (Figure 9.11).

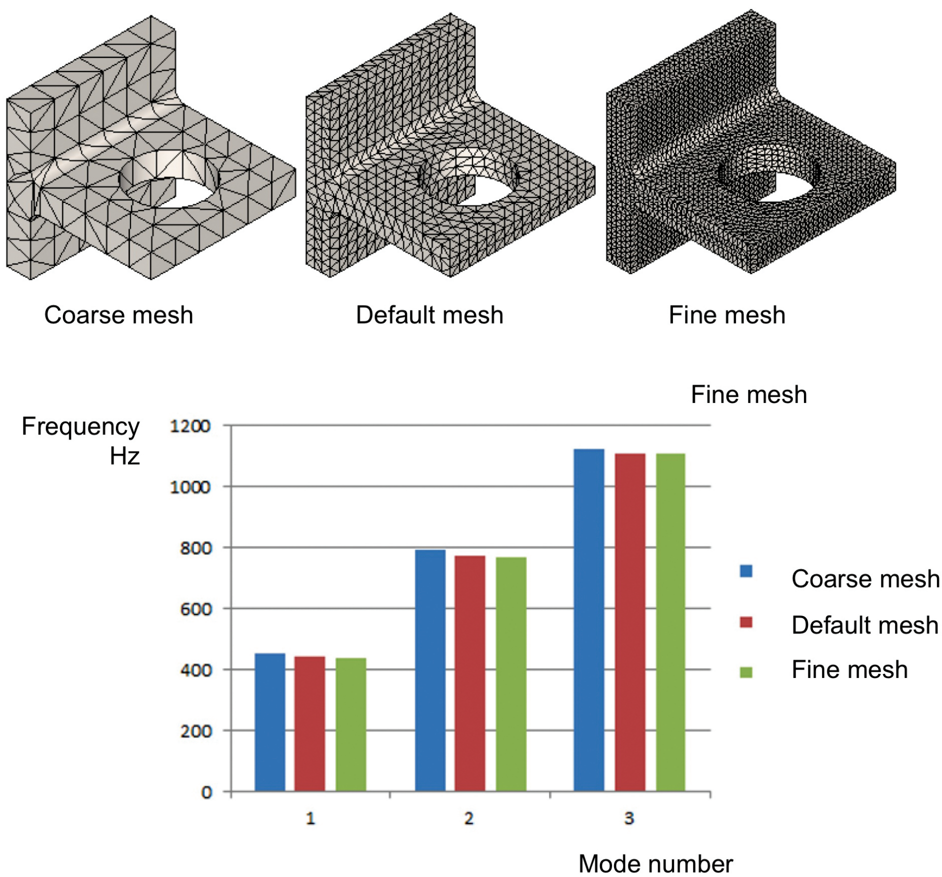


Figure 9.11 Convergence of frequency of the first three modes of vibration of a bracket. Natural frequencies converge from above with mesh refinement; a default mesh is usually sufficient for modal analysis.

9.9 Meshing Consideration for Modal Analysis

As shown in Figure 9.11, the results of modal analysis are not strongly dependent on the element size. In general, modal analysis requires less-refined mesh than stress analysis of the same model for the same reason why displacement analysis requires less refined mesh than stress analysis. Small details, important for stress analysis, can be removed from the model as long the analysis objective is to find modes of vibration and modal frequencies. However, if modal analysis is needed to provide data for the subsequent vibration analysis with modal superposition method, and the objective of vibration analysis is to find stresses, then small details must be included in the model just like in static stress analysis.

9.10 Hands-On Exercises

9.10.1 Tuning Fork (Figure 9.12)

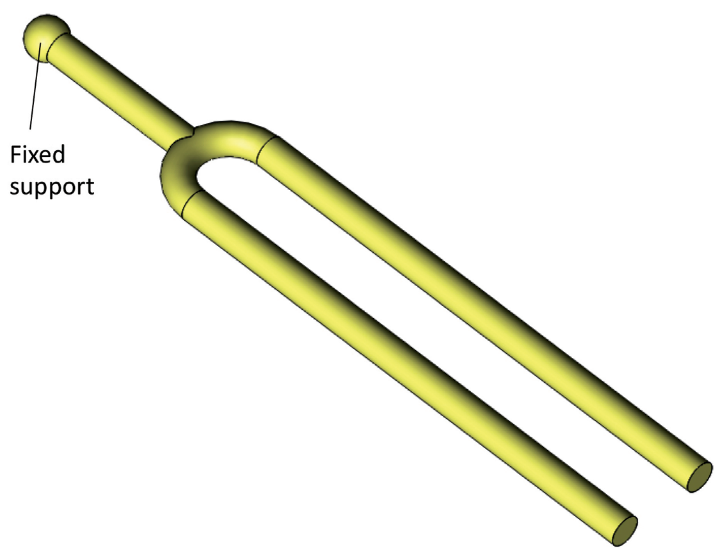


Figure 9.12 Model of a tuning fork.

Model name

- 9.01.TUNING_FORK.x_t
- 9.01.TUNING_FORK.sldprt

9.10.1.1 Objective: Modal analysis without supports and modal analysis with supports.

The following are the required steps:

1. Apply material properties (steel).
2. Specify at least ten modes to be calculated.

3. Mesh and obtain the modal solution without supports.
4. Apply support.
5. Mesh and obtain the modal solution with support.
6. Compare the results for both solutions.

9.10.1.2 Comments: Observe that displacements and stress results are meaningless in terms of providing quantitative information. Observe that solution with no supports returns six modes with zero frequency corresponding to six rigid body modes. The first non-zero frequency mode (or the first elastic mode) is the fundamental frequency of the tuning fork: 440 Hz (lower A in music terminology). The modal solution with supports returns three modes with frequency lower than 440 Hz. However, those modes require support in order to exist and, therefore, they are damped out allowing the supported tuning fork to vibrate to 440 Hz as it is designed to do.

9.10.2 Box (Figure 9.1)

Model name

- 9.02.BOX.x_t
- 9.02.BOX.sldprt

9.10.2.1 Objective: Demonstrate modal analysis used to locate "weak spots."

9.10.2.2 Comments: Modal analysis provides results of displacement and stress that are of comparative importance only. In this exercise we observe that stresses are calculated based on qualitative displacement results.

The following are the required steps:

1. Apply the material properties (ABS plastic).
2. Specify at least ten modes to be calculated.
3. Mesh and obtain a modal solution without supports.
4. Review the displacement results in the first elastic mode and probe displacements in selected locations to find the ratio between displacements.
5. If your program supports this option, analyze stress results for each mode; locate "weak spots" corresponding to each mode. Observe that the stress solution for each mode corresponds to normalized displacements.
6. Perform several mesh refinements and analyze the convergence of modal frequencies; note that they converge from above.

9.10.3 Airplane (Figure 9.2)

Model name

- 9.03.AIRPLANE.x_t
- 9.03.AIRPLANE.sldprt

9.10.3.1 Objective: Review modes of an unsupported model.

9.10.3.2 Comments: Model is symmetric, therefore all modes are either symmetric or antisymmetric. Model with no restraints is used in this example, but symmetric and antisymmetric modes may be also observed in restrained models if geometry and restraints are symmetric.

The following are the required steps:

1. Apply the material properties (ABS plastic).
2. Do not define any restraints.
3. Specify at least twelve modes to be found.
4. Mesh and solve.
5. Animate six rigid body modes and notice that translations and rotations are associated with rigid body modes.
6. Animate six elastic modes and observe that modal shapes are either symmetric or antisymmetric; this happens because the model is symmetric and there are no restraints.

9.10.4 Ball (Figure 9.4)

Model name

- 9.04.BALL.x_t
- 9.04.BALL.sldprt

9.10.4.1 Objective: Review modes of a partially supported model with rigid body modes associated with rotation.

9.10.4.2 Comments: This exercise is conducted on a part; the assembly shown in Figure 9.4 is not used.

The following are the required steps:

1. Apply the material properties (1060 alloy).
2. Define restraints on the spherical face in a spherical coordinate system associated with that face and restrain radial translations; this simulates ball joint support.
3. Specify at least six modes of vibration to be calculated.
4. Mesh and solve.
5. Notice that the first three are rigid body modes corresponding to rotations.
6. Notice that the remaining modes are elastic modes associated with deformation on the model.

9.10.5 Link (Figure 9.5)

Model name

- 9.05.LINK.x_t
- 9.05.LINK.sldprt

9.10.5.1 Objective: Demonstrate the effect of mass participation on modal frequencies.

9.10.5.2 Comments: Observe that the corresponding modal frequencies are lower for an unsupported shaft because less mass participates in vibrations of the unsupported shaft. This is because in the supported shaft the center of mass participates in vibration while the unsupported shaft vibrates about its center of mass.

The following are the required steps:

1. Apply the material properties (1060 alloy).
2. Specify at least eight modes to be found.
3. Do not define any supports.
4. Mesh and solve; expect six rigid body modes.
5. Define two hinge supports by restraining radial and axial translations on both cylindrical faces. Each hinge support has to be defined in its own local cylindrical coordinate system.
6. Obtain the modal solution with hinge supports.
7. Compare the frequency of mode 7 in solution without supports to the frequency of mode 1 in solution with hinge supports.

9.10.6 Helicopter Blade (Figure 9.7)

Model name

- 9.06.BLADE.x_t
- 9.06.BLADE.sldprt

9.10.6.1 Objective: Demonstrate the effect of tensile prestress on modal frequencies.

9.10.6.2 Comments: Observe that the applied angular velocity generates a centrifugal load that produces tensile stresses in the blades. Because of stress stiffening effect, tensile stresses increase blade stiffness. Consequently, the natural frequencies of the spinning blade are higher compared with those of the blade that is not moving.

Modal analysis with prestress is executed in three steps: first static stress solution is found and then that stress is used to modify models stiffness. Finally, modal analysis is run on the model with modified stiffness.

Use 02 one blade configuration in SOLIDWORKS model; delete three blades in Parasolid model. The following are the required steps:

1. Define fixed restraints to the hub.

2. Specify four modes to be found.
3. Mesh and solve.
4. Define the load as an angular velocity 180 rpm.
5. Find the first four modes of vibration of the blade with preload. Compare results without and with pre-load.

9.10.7 Column (Figure 9.8)

Model name

- 9.07.COLUMN.x_t
- 9.07.COLUMN.sldprt

9.10.7.1 Objective: Demonstrate the effect of compressive prestress on modal frequencies.

9.10.7.2 Comment: The last modal analysis with prestress will report a low frequency because the preload is close to the buckling load. Try running the analysis again with 100% of buckling load and observe that solution will crash because the model has no stiffness. Now run the analysis with the preload equal to 99.9% of the buckling load; the natural frequency will be very low, the model has almost no stiffness and that stiffness comes mostly from artificial stiffness. You now have a tool to demonstrate the presence of artificial stiffness. Run the modal analysis again with preload 99.9% but with a more refined mesh. The solution will crash because the artificial stiffness has decreased and model has lost stiffness.

9.10.7.3 Note: This exercise starts with linear buckling analysis; you may want to work on it after completing chapter 10.

The following are the required steps:

1. Define the material properties (steel).
2. Define a fixed support to the bottom end face of the channel.
3. Complete linear buckling analysis to find the buckling load.
4. Run six modal analyses using compressive load of increasing magnitude: 0% of buckling load, 20% of buckling load, 40% of buckling load, 60% of buckling load, 85% of buckling load, and 95% of buckling load. These six analyses are modal analyses with prestress.
5. Construct a graph similar to the one shown in Figure 9.9.

9.10.8 Bracket (Figure 9.10)

Model name

- 9.08.BRACKET.x_t
- 9.08.BRACKET.sldprt

9.10.8.1 Objective: Perform modal analysis using symmetry and antisymmetry boundary conditions.

9.10.8.2 Comments: Observe that combining solutions of a half model with symmetry and antisymmetry boundary conditions produces the full set of results. Assemble the symmetric and antisymmetric solutions and sort results by the ascending order of modal frequencies.

The following are the required steps:

1. Apply the material properties (ABS plastic).
2. Apply a fixed support to the back face.
3. Specify six modes to be calculated.
4. Mesh and solve.
5. If you use parasolid geometry, cut the model into half; if you use SOLIDWORKS geometry, switch to configuration *02 half*.
6. Obtain the modal solution (six modes) for the half model with symmetry boundary conditions.
7. Obtain the modal solution (six modes) for the half model with antisymmetry boundary conditions.
8. Combine the solutions obtained in 5 and 6, as shown in Table 9.1.

Chapter 10
Buckling Analysis

The phenomenon of buckling is best presented in the context of two other failure modes: excessive displacements and yielding as summarized in Table 10.1. Buckling is often overlooked as a mode of failure; this is why understanding of how buckling analysis is implemented in FEA is particularly important.

Table 10.1 Different modes of failure encountered in structural design

Mode of failure	Level of recognition	Characteristics
Excessive displacement	At times, we forget to check for displacements	This is a global failure mode. Excessive displacements, by themselves, may render design unusable but do not have to lead to a structural collapse. Displacements are easy to model using FEA
Yielding	A well-recognized mode of failure	Typically a local failure mode; small yield zones can often be tolerated. Does not necessarily lead to structural collapse. Yield is relatively easy to model using the FEA.
Buckling	Often forgotten mode of structural failure	This is a global failure mode. Often happens without prior warning. Almost always leads to structural collapse. Buckling is difficult to model with FEA.

Buckling analysis is available in commercial FEA programs under the name buckling analysis, eigenvalue buckling analysis or linear buckling analysis. It is also available as a part of nonlinear analysis. This fragmentized approach may lead to misunderstanding of what buckling analysis really is. It is the author's hope that the following discussion will help understand buckling analysis conducted with FEA as well as the buckling phenomenon itself.

10.1 Linear Buckling Analysis

We start with linear buckling analysis, also referred to as eigenvalue buckling analysis. The linear buckling analysis is in many ways similar to modal analysis, as explained in Table 10.2. Linear buckling analysis is easy to execute, but, it is limited in the results it can provide.

Table 10.2 Analogies between buckling and modal analysis

	Buckling analysis	Modal analysis
Eigenvalue	Load causing buckling	Natural frequency (or more precisely a square of the natural frequency in rad/s)
Eigenvector	Buckled mode (shape) assumed by the structure when loaded with a load causing buckling	Mode (shape) assumed by the structure when it vibrates with the natural frequency
How many modes have practical importance?	The first mode with a positive buckling load factor (BLF)	Usually the first few modes

Buckling takes place when, as a result of adding negative stress stiffness induced by compressive load to geometric stiffness, the resultant structure stiffness drops to zero. This remains in close analogy to modal analysis with compressive prestress where negative inertial stiffness is added to geometric stiffness also producing zero resultant stiffness.

Typically, results of linear buckling analysis do not report the buckling load. Instead, they present buckling load factor (BLF). The BLF is a number by which the applied load must be multiplied by (or divided, depending on the FEA program in use) in order to obtain the magnitude of buckling load. The buckling mode presents the shape that the structure assumes when it buckles in a particular mode, but says nothing about the numerical values of the displacements or stresses. Most FEA programs offer the option to display numerical values of displacements but these are meaningless. This is in close analogy to modal analysis that calculates the natural frequency and provides qualitative information on the modes of vibration (modal shapes), but not on the actual magnitude of displacements.

The linear buckling analysis calculates buckling loads and associated buckling shapes. FEA programs offer a choice of how many buckling modes should be found. Theoretically, as many buckling modes as the number of degrees of freedom in the FEA model could be calculated.

Usually, only the first positive buckling mode and its associated BLF need to be found. This is because higher buckling modes have no chance of taking place. Buckling most often causes catastrophic failure or renders the structure unusable even if the structure is still able to hold the load in the postbuckling state. Note that we said, "the first positive buckling mode," because buckling modes are reported in ascending order according to their numerical values. A buckling mode with a negative BLF means that the load

direction has to be reversed (in addition to multiplying by the BLF) in order for the buckling to happen.

To illustrate the linear buckling analysis, we use a model of a column in compression; the column material is 1060 alloy. We study this model in two configurations: with a free end and a sliding end. Before proceeding, notice the small notch and try predicting which way the column will buckle in each configuration. The importance of the notch will soon become clear.

Figure 10.1 shows the results of the linear buckling analysis of a column with the free end.

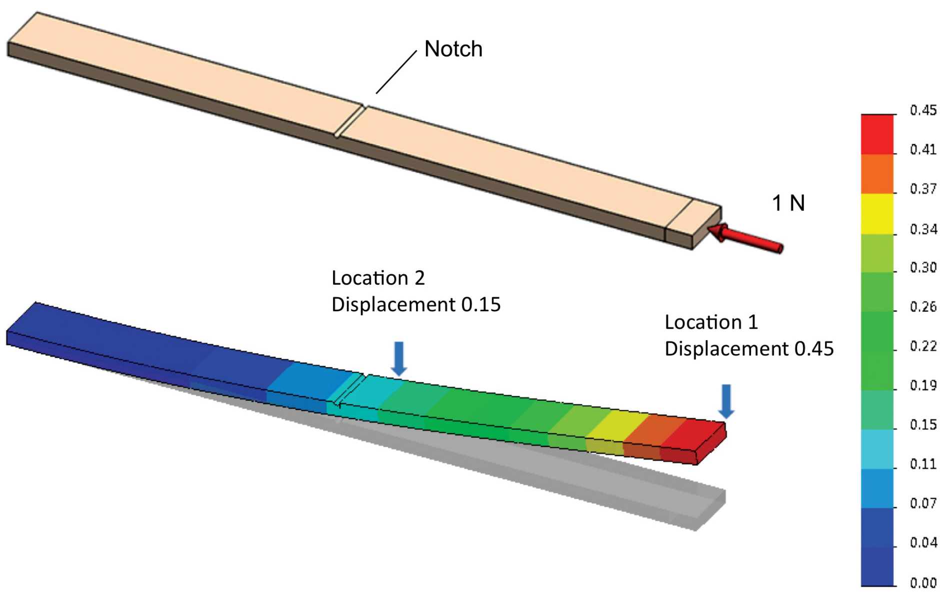


Figure 10.1 A notched column is subjected to a compressive load of 1 N as shown in the top illustration. A fixed restraint is applied to the end face opposite to the loaded side. The loaded end is not restrained in any way. The BLF found in this analysis is 566, meaning that the load of 566 N will cause buckling according to the linear buckling analysis. The bottom illustration shows the first buckling mode. An undeformed shape is overlaid on the deformed shape. Numbers may only be used to find the ratio between displacements in different locations and only within the same buckling mode. Here, the ratio of displacements between locations 1 and 2 is 0.45/0.15 = 3.

The BLF is 566, therefore considering that a 1-N load has been applied, the buckling is predicted to happen at 566 N. Notice that the column has buckled toward the notched side, but this result is purely coincidental; linear buckling analysis can predict the buckled shape, but not the direction of buckling. Figure 10.2 shows the result of the analysis of the column with a sliding end. The BLF is 9622; this is much higher than before because the sliding support makes the column stiffer. The direction of buckling shown in Figure 10.2 is again purely coincidental. The color legends with displacement

magnitudes in both figures are shown only to point out their meaninglessness in terms of absolute numbers.

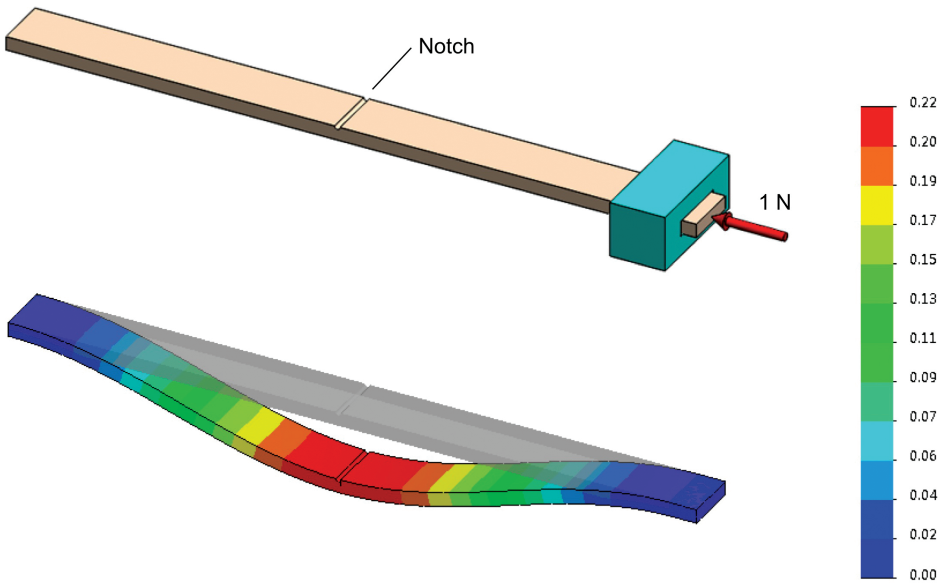


Figure 10.2 A notched column is subjected to compressive load as shown in the top illustration. A fixed restraint is applied to the end face opposite to the loaded side. The loaded end may only slide along the length on the column. The BLF found in this analysis is 9564, meaning that the load of 9564 N will cause buckling.

Linear buckling analysis cannot provide any quantitative results for displacements or stresses; it does not provide any answer to what happens to the structure after buckling. Does the structure collapse or will it retain the load bearing ability in the buckled shape? How much will it deform when it buckles? These questions remain unanswered. All what linear buckling does is finding combinations of compressive load and deformed shape that lead to the cancellation of stiffness.

Four buckling modes of the column with free end are shown in Figure 10.3.

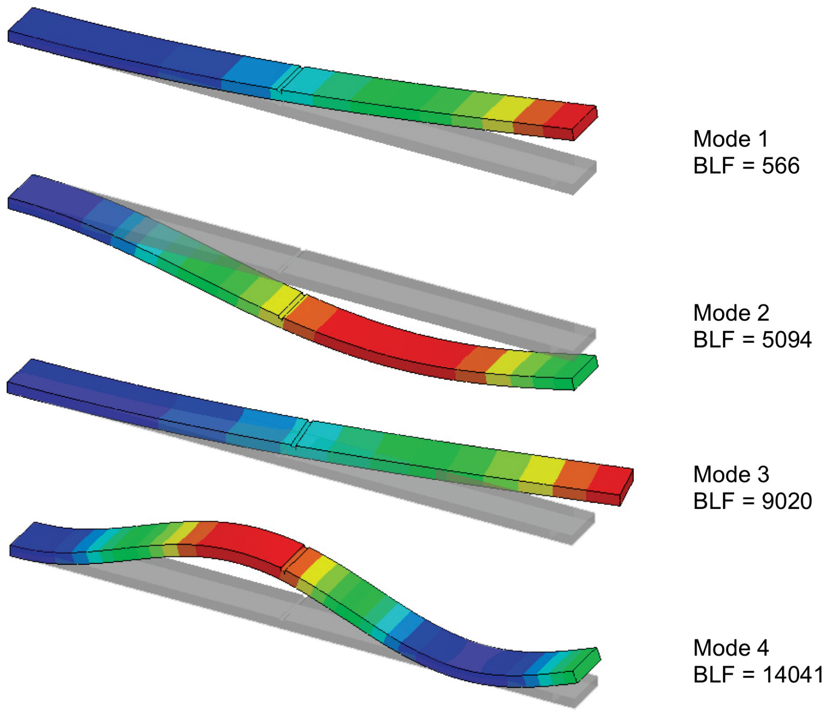


Figure 10.3 The first four buckling modes of the notched column. The BLF relates to a compressive load of 1 N. Only mode 1 is of practical importance; the other three modes have no chance to appear.

If we are looking for answers beyond the BLF and the buckled shape, we need to use the nonlinear buckling analysis.

10.2 Convergence of Results in Linear Buckling Analysis

Just like modal frequencies in modal analysis, BLFs converge from above. Therefore, buckling analysis overestimates buckling load and as a consequence of discretization error provides nonconservative results. However, the results of buckling analysis are also overestimated because of modeling errors. Finite-element models represent geometry with no imperfections, and loads and supports are applied with perfect accuracy with no offsets. In reality, the load is always applied with an offset, walls are never perfectly flat, and supports are never perfectly rigid. Considering the combined effect of both discretization error (minor effect) and modeling error (major effect), the results of buckling analyses must be interpreted with caution.

10.3 Nonlinear Buckling Analysis

Buckling is inherently nonlinear phenomenon. Compressive loads produce compressive stresses, which in turn generate negative stress stiffness. If the magnitude of compressive loads is high enough, then the negative stress stiffness cancels with geometric

stiffness making the resultant stiffness zero. The structure with zero stiffness cannot support load and collapses; this is what we call buckling.

Linear buckling analysis (or eigenvalue based buckling analysis), as discussed above, can only identify combinations of load and deformed shape that produce cancellation of stiffness. These results, while useful, do not describe the buckling fully. An in-depth analysis of buckling requires a nonlinear approach.

As with any other nonlinear analysis, a nonlinear buckling analysis requires that the load be applied in multiple steps rather than in one step as is done in a linear analysis. Each load increment changes the structure's shape and stiffness. Therefore, the structure stiffness has to be updated during the process of step-by-step load increments. The gradual load application can be conducted in many ways. The process illustrated in Figure 10.4 is called the load control method and is used in most types of nonlinear analyses, but not in a buckling analysis. When buckling happens, the structure experiences a momentary loss of stiffness and the load control method would result in numerical instabilities. For this reason, nonlinear buckling analysis requires another way of controlling the load application process, which is called the arc length control method shown in Figure 10.5. Using this method, points corresponding to consecutive load increments are evenly spaced out along the load–displacement curve, which itself is constructed during the process of the load application.

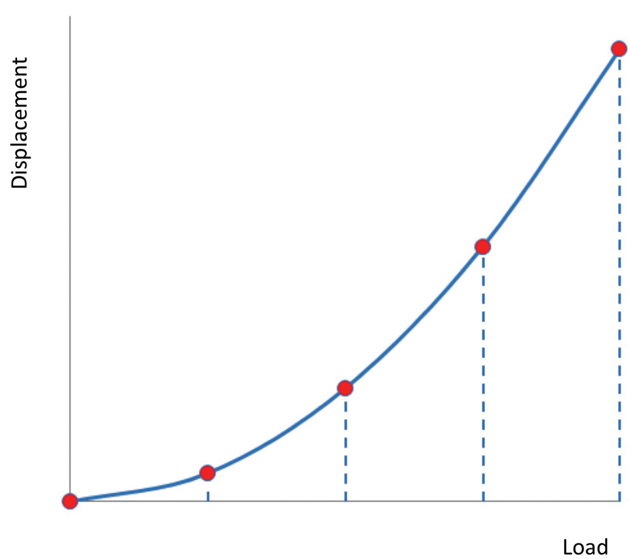


Figure 10.4 Force control method. Load steps are defined either by user or automatically so that the difference in displacement between two consecutive steps is not too large.

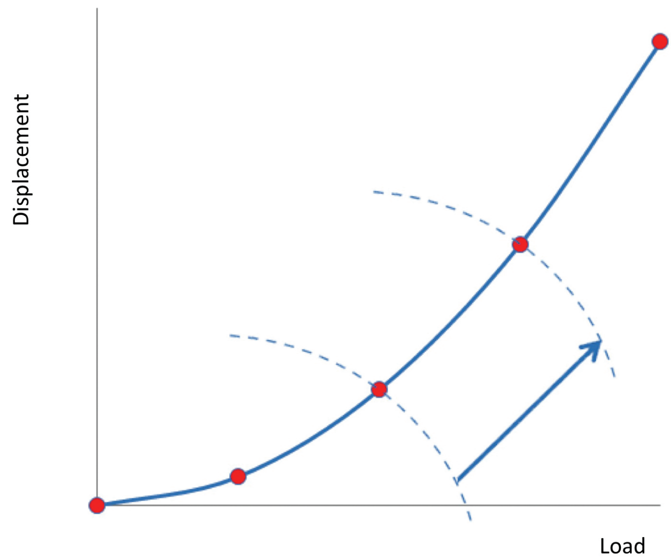


Figure 10.5 The arc-length control method. Load is increased in such a way that consecutive steps are equally spaced out along the load–displacement curve.

As opposed to how linear buckling analysis calculates only the potential buckling shape with no quantitative values of importance, the nonlinear buckling analysis continues past buckling and calculates actual displacements and stresses. To understand the inner workings of nonlinear buckling analysis, we must first consider what happens if we run a nonlinear buckling analysis on an idealized structure. Imagine a perfectly round and straight column under a perfectly aligned compressive load. Theoretically, buckling will never happen, but in real life, it will happen because of imperfections always present in the geometry, loads, and supports. If these imperfections are absent in the FEA model, the buckling analysis will still report buckling, but it will be initiated by imperfections introduced by discretization errors coming from meshing. Therefore, nonlinear buckling requires a model with some initial imperfection. If no such imperfections exist, they must be added to control the onset of buckling. In our case, the imperfection is the notch shown in Figures 10.1 and 10.2. Another approach is to run the analysis on a model in a deformed shape, which is based on a scaled result of linear buckling analysis.

Figure 10.6 shows the buckled shape and numerical values of displacement in Y direction of the loaded end. These numbers and direction of buckling are meaningful. Notice that buckling takes place at approximately 560 N. This load magnitude is close to what the linear buckling analysis predicted. The column buckles toward the notched side as it should.

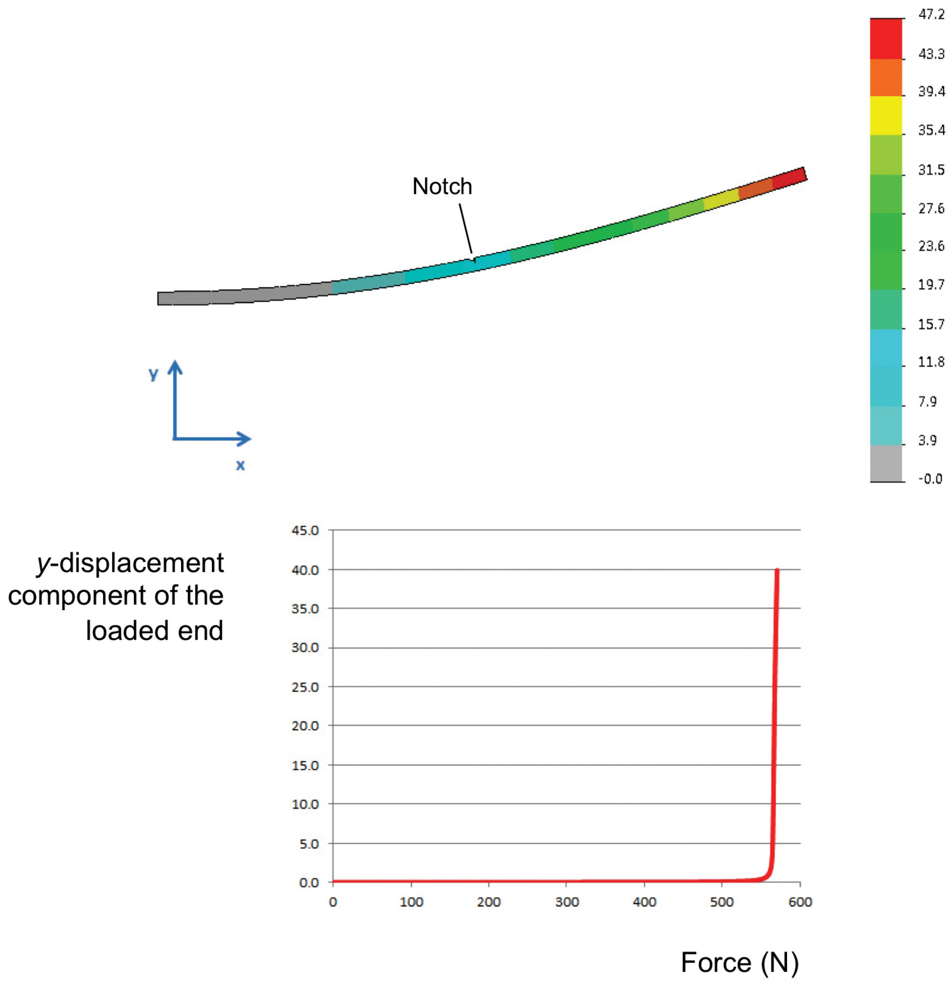


Figure 10.6 Displacement results of nonlinear buckling analysis of the notched column with free end using the linear material model.

Once buckling takes place, the displacement grows at an almost constant force. The graph stops at 40 mm, but the analysis could have been run further; this implies that the column is capable of balancing the 550 N acting at an increasing offset, which would mean it is capable of withstanding an increasing bending moment with no bounds. Clearly, that cannot happen because of material yielding.

Figure 10.6 shows the results of a nonlinear analysis where the only source of nonlinear behavior is buckling. The material is linear elastic and yielding is ignored. This is clearly

unrealistic; experiments demonstrate that buckling initiates a chain of events shown in Figure 10.7. To model these events, we must account for another source of nonlinearity: yielding. We do so using an elastic–perfectly plastic material characterized discussed in chapter 8. The deformed shape and the load–displacement curve produced by this new model is shown in Figure 10.8. The buckling still happens at 560 N indicating that the onset of buckling takes place while the material is still in the elastic range. After a period of elastic buckling (the vertical portion of the blue curve), the load rapidly drops. This means that to maintain the equilibrium, the applied force must be reduced. If the load stays the same, as is most often the case, the end of the vertical portion of the curve marks the column's structural collapse. At this point, the column is totally plasticized and can no longer support the offset load. Examine the deformed shape and note that most of the deformation takes place at the support where the plastic hinge develops.

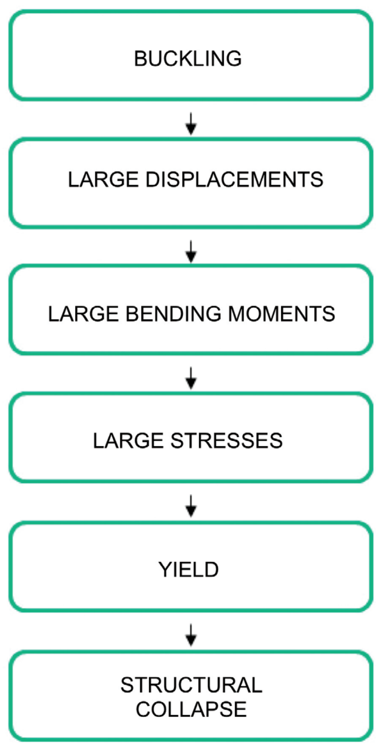


Figure 10.7 Chain of events initiated by buckling leads to a structural collapse.

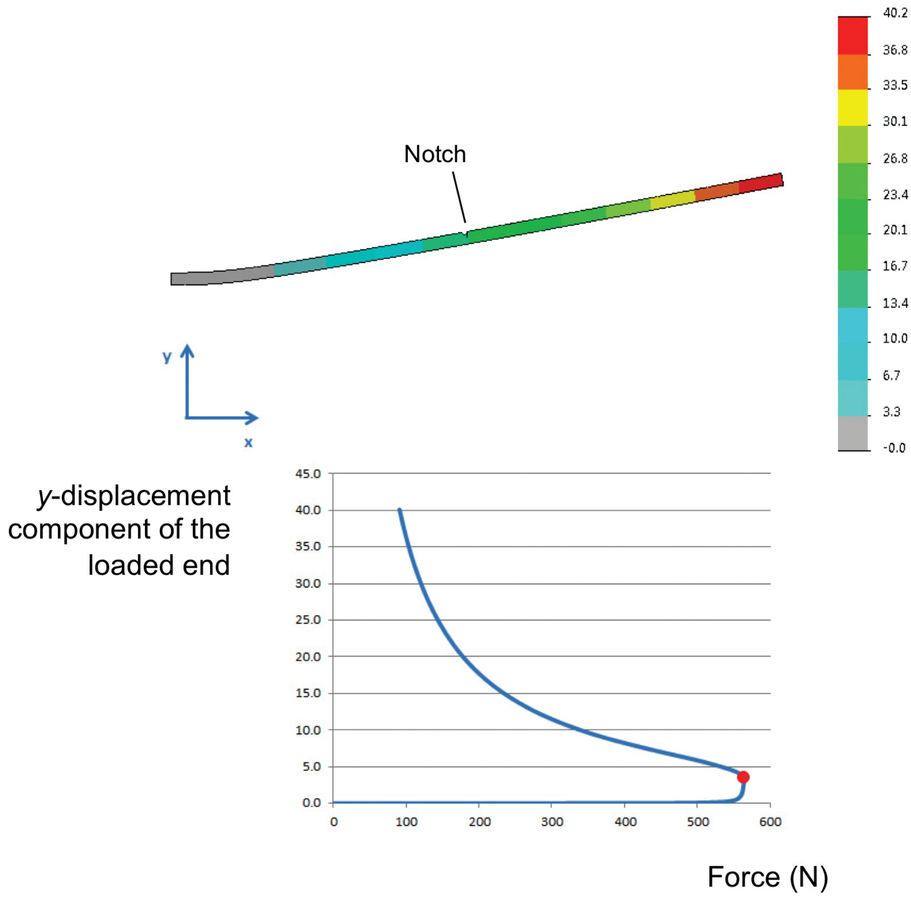


Figure 10.8 Displacement results of nonlinear buckling analysis of the notched column with free end using an elastic–perfectly plastic material model. The red point indicates a structural collapse of the compressed column.

We now conduct the analysis for the column with the sliding restraint to examine the results of a nonlinear buckling analysis with a linear material and an elastic–perfectly plastic material. The linear material model (Figure 10.9) predicts buckling at approximately 9500 N, which is close to the results of linear analysis. However, this load causes yielding of the entire cross section in the notched area and that cannot be modeled with a linear elastic material model. Using the elastic–perfectly plastic model (Figure 10.10), we see that buckling takes place at 2500 N when the notched cross section is already completely plastic.

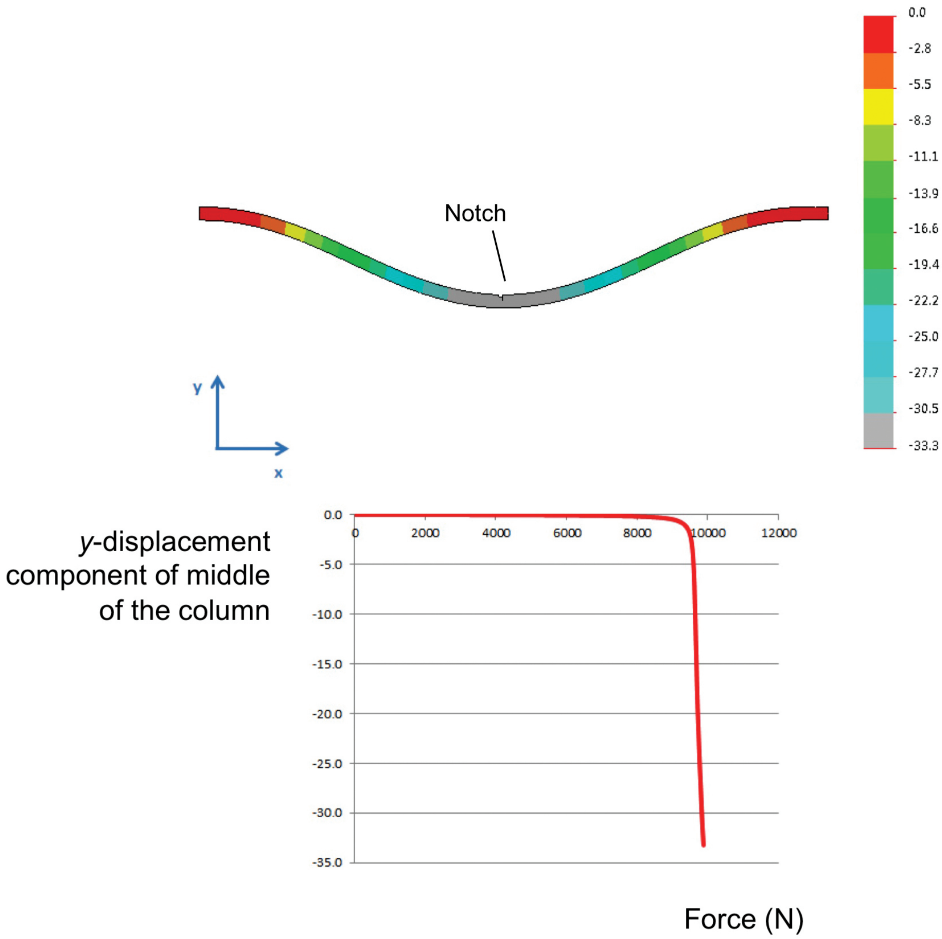


Figure 10.9 Displacement results of nonlinear buckling analysis of the notched column with sliding end using the linear material model.

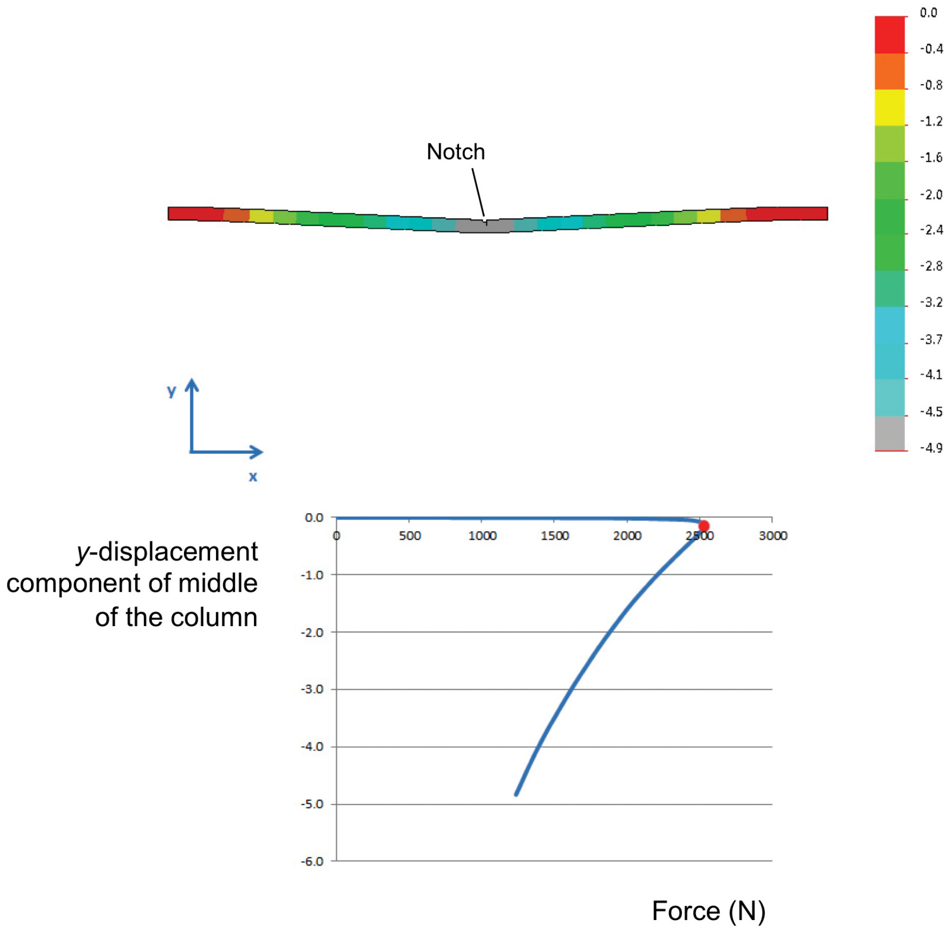


Figure 10.10 Displacement results of nonlinear buckling analysis of the notched column with sliding end using the elastic–perfectly material model. The red point indicates a structural collapse of the compressed column.

Sometimes, buckling does not lead to a structural collapse. To illustrate this, we review a model of a thin spherical steel plate loaded and supported as shown in Figure 10.11. Linear buckling analysis predicts that buckling will take place when pressure euqals 0.69 MPa and the direction of buckling is opposite to the direction of load, an obviously wrong result (Figure 10.12).

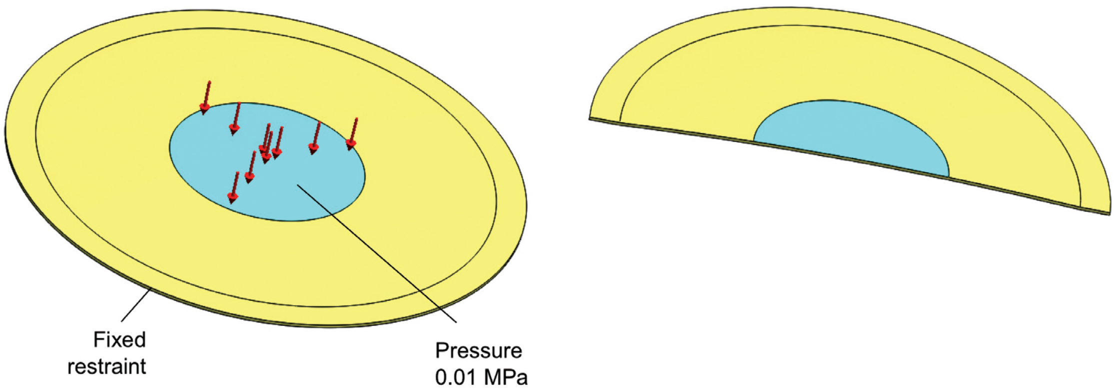


Figure 10.11 A round spherical sheet under a pressure load. The left illustration shows a complete view with restraint and load; fixed restraint is applied to the cylindrical circumference face and pressure is applied to the blue face. The right illustration shows a radial cross section that makes it easier to observe a slight spherical curvature.

Figure 10.12 shows results of linear buckling analysis conducted on CAD model without any simplifications.

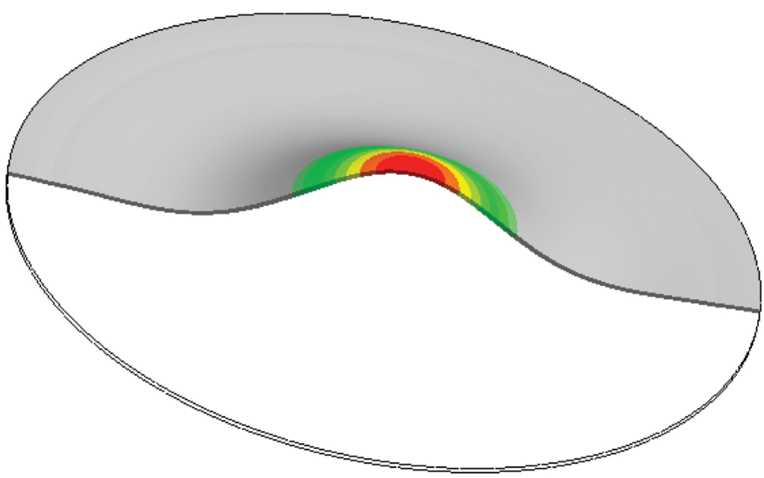


Figure 10.12 Results of linear buckling analysis showing the first buckling mode. The pressure causing buckling is 0.69 MPa. Notice an incorrect direction of buckling. Cross section view is used to show the buckling shape. The BLF is 69 meaning that linear buckling analysis predicts buckling at pressure 0.69MPa.

Nonlinear buckling analysis is conducted using a 2D axisymmetric model. A 2D model is used to reduce solution times; nonlinear buckling analysis is run twice: with linear material and with an elastic–perfectly plastic material. The buckled shape is the same for both materials as shown in Figure 10.13; notice that the nonlinear buckling analysis correctly predicts that buckling happens in the direction of load. The load–displacement

curves for both material models are shown in Figure 10.14. Both curves are identical well past buckling meaning that after buckling, the plate is still in the elastic state and may return to the original shape if load is removed.

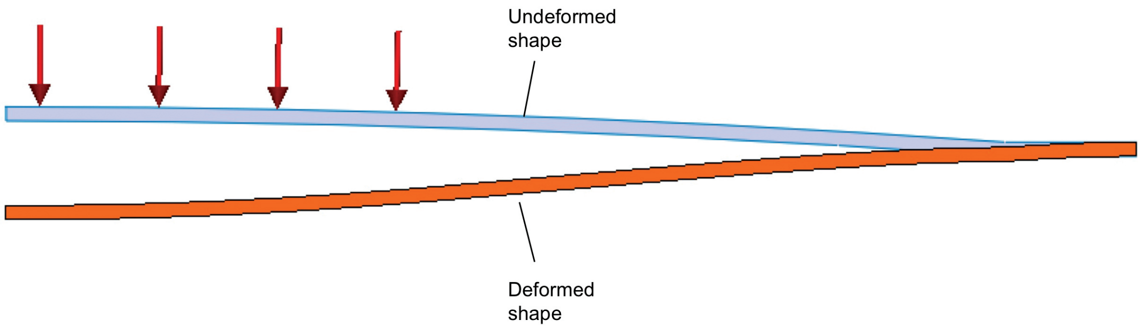


Figure 10.13 The 2D axisymmetric nonlinear model before and after deformation. The deformed shape corresponds to the end of the load–displacement curve.

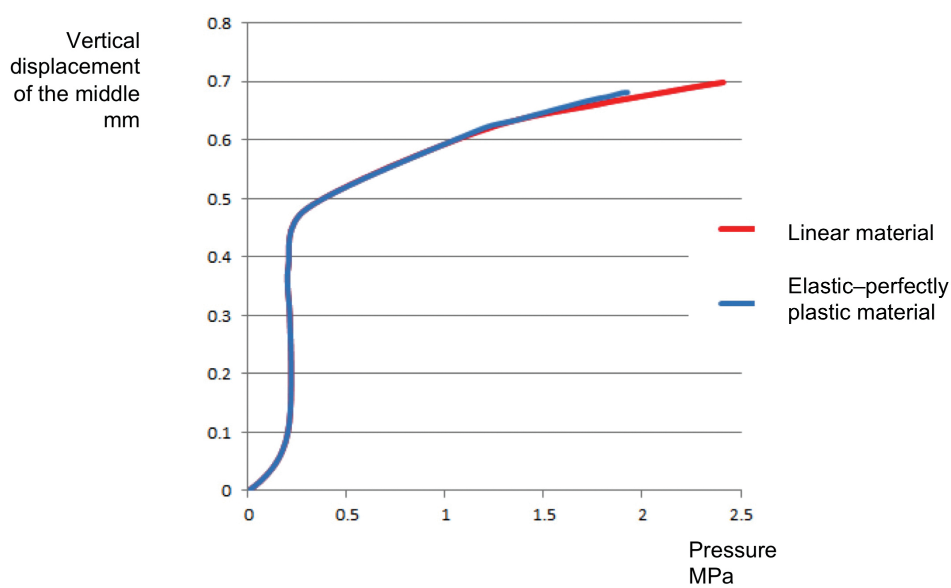


Figure 10.14 Load–displacement curves of nonlinear buckling models with the linear material and the elastic–perfectly plastic material. Both curves are identical until pressure reaches 1.5 MPa. The vertical portion of curve corresponds to buckling when model loses stiffness. Deformation progresses with no increase of load until it reaches the magnitude of ~0.45 mm and then model regains stiffness in the postbuckled shape and regains the load bearing capability.

A graph in Figure 10.14 shows a stiffening effect in the postbuckling stage meaning that the plate regains stiffness after buckling has taken place. The phenomenon shown in Figure 10.13 where a curved plate under compression "snaps through" is sometimes called oil-canning effect.

In nonlinear buckling analysis, the onset of buckling must be initiated in a controlled way. If it is not, the model will still buckle, but buckling will happen because of imperfections introduced by mesh.

In the notched column exercise, we controlled the onset of buckling with a notch. Another approach might be using a slightly curved column shaped after a scaled buckling shape found in linear buckling analysis (Figure 10.15). Nonlinear buckling analysis of a stand (Figure 10.16) uses load with an offset. Buckling analysis of a curved sheet (Figure 10.17) does not require any purposely introduced imperfection. The onset of buckling is controlled by sheet curvature.

Figure 10.15 A column similar to the one shown in Figure 10.1 but without a notch. An imperfection is introduced here as a slight curvature.

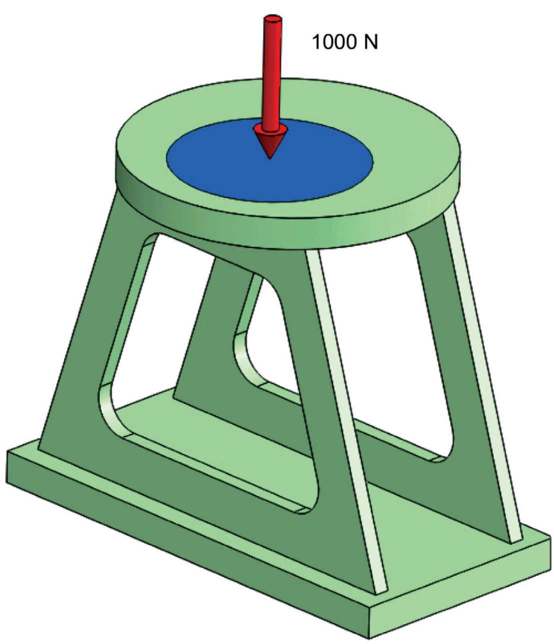


Figure 10.16 A stand subjected to an offset load. Load is uniformly distributed over the blue face.

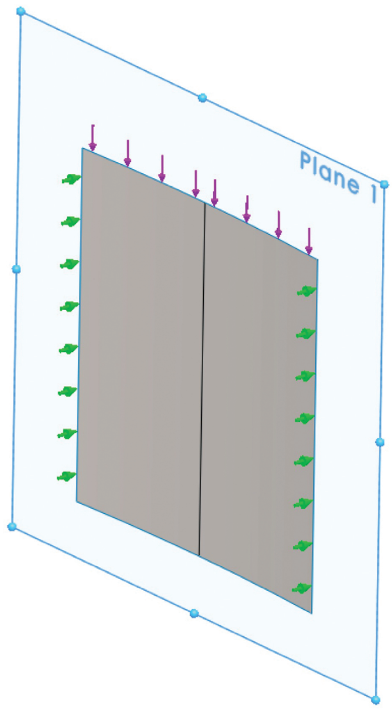


Figure 10.17 A curved sheet under a compressive load of 1000 N uniformly distributed along the top edge. Translations of vertical edges in the direction normal to Plane 1 are restrained. The bottom edge is fully supported (restraint symbols not shown).

10.4 Summary

The phenomenon of buckling requires an in-depth consideration during the design process. Linear buckling analysis can only provide limited information about buckling. Nonlinear buckling analysis is often required; We should always consider using a nonlinear material model in nonlinear buckling analysis.

What makes the nonlinear buckling analysis more difficult than other types of nonlinear analysis is that the onset of buckling is associated with a loss of stiffness. If analysis is continued past buckling, the structure can in some cases regain stiffness in the post-buckling state. That momentary loss of stiffness requires a special way of controlling load increments called arc control as shown in Figure 10.5. The shape of load–displacement curve can be used to categorize types of nonlinearities into "easy" and "difficult" ones (Figure 10.18).

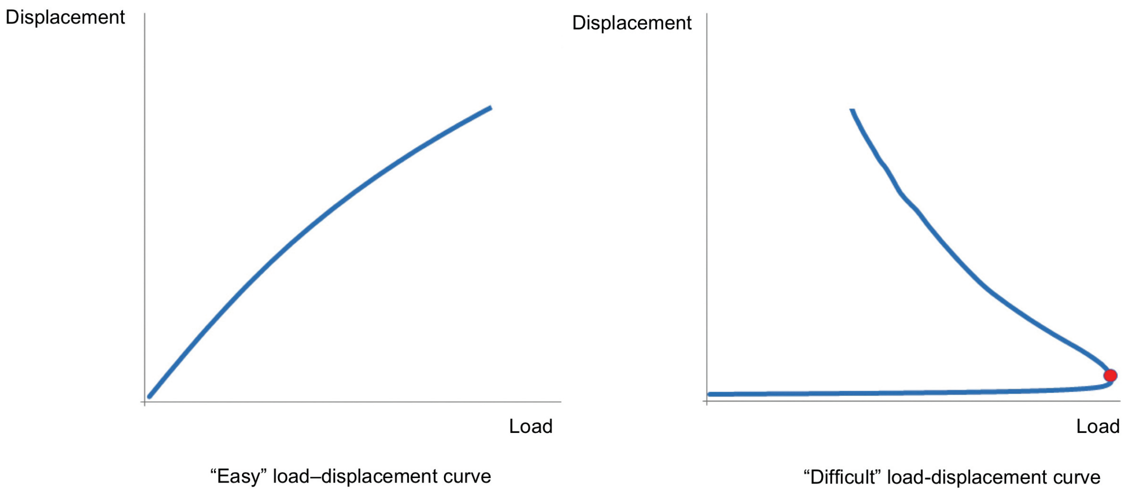


Figure 10.18 Membrane stress stiffening solution produces a smooth load–displacement curve (left). The stiffness changes gradually and at no point stiffness becomes zero. A buckling problem combined with a nonlinear material produces a more complicated load–displacement curve (right). The red dot indicates a vertical portion of the curve when stiffness momentarily disappears.

10.5 Hands-On Exercises

All exercises require arc-length control of load; all exercises require definitions of sensors, which are locations where displacements are monitored to construct load–displacement curves. Load–displacement curves should show displacement in the sensor location and in the direction of buckling.

Not all exercises specify the load to be applied. This is to demonstrate that load magnitude is required only when interpreting results of both linear and nonlinear analysis; different load magnitudes produce different BLFs.

10.5.1 Notched Column—Free End (Figure 10.1)

Model name

- 10.01.NOTCHED_COLUMN. x_t
- 10.01.NOTCHED_COLUMN. sldprt

10.5.1.1 Objective: Perform linear and nonlinear buckling analysis on a model where the onset of buckling is controlled by imperfection (notch).

Perform nonlinear buckling analysis on a model with linear and elastic–perfectly plastic materials.

10.5.1.2 Comments: Try using different meshes and different solvers (if available) in linear buckling analysis; you will see that the direction of buckling changes. Review

results of linear analysis showing higher buckling modes and compare their shapes with shapes of modes of vibration.

The following are the required steps:

1. Define the linear material model (1060 alloy).
2. Apply a fixed support to the end face opposite to the loaded face (Figure 10.1).
3. Specify four buckling modes to be found.
4. Mesh and solve the linear analysis.
5. Define the displacement sensor at the loaded end in the direction of buckling.
6. Perform a nonlinear buckling analysis; use the arc-length load control method.
7. Construct the load–displacement curve.
8. Compare the results of linear buckling analysis and nonlinear buckling analysis.
9. Repeat steps 1–8 using the elastic–perfectly plastic material model.

10.5.2 Notched Column—Sliding End (Figure 10.2)

Model name

- 10.01.NOTCHED_COLUMN.x_t
- 10.01.NOTCHED_COLUMN.sldprt

Follow the same steps as in the exercise in Section 10.5.1. Model sliding end with appropriate restraints applied to the sliding end. Notice that buckling in nonlinear analysis happens when the cross section is plasticized.

10.5.3 Button (Figure 10.11)

Model name

- 10.02.BUTTON.x_t
- 10.02.BUTTON.sldprt

10.5.3.1 Objective: Perform nonlinear buckling analysis using 2D axisymmetric representation.

10.5.3.2 Comment: You may repeat the analysis using an elastic–perfectly plastic material model and notice that postbuckling state where the model “snaps through” does not cause yielding. This means that upon the removal of load, the model returns to the original shape.

The following are the required steps:

1. Define linear material properties (steel).
2. Use 2D axisymmetric representation to define load and restraint, as shown in Figure 10.11.
3. Mesh and solve the linear buckling analysis.

4. Define the displacement sensor in the center of the top face where pressure load is applied.
5. Perform a nonlinear buckling analysis.
6. Construct the load–displacement curve.
7. Compare the results of linear buckling analysis and nonlinear buckling analysis.

10.5.4 Curved Column (Figure 10.15)

Model name

- 10.03.CURVED_COLUMN.x_t
- 10.03.CURVED_COLUMN.sldprt

10.5.4.1 Objective: Repeat exercise 10.5.1 with the difference that the onset of buckling is now controlled by an initial curvature, not a notch.

10.5.5 Stand (Figure 10.16)

Model name

- 10.04.STAND. x_t
- 10.04.STAND. sldasm

10.5.5.1 Objective: Perform a nonlinear buckling analysis with an elastic–perfectly plastic material on a model where an onset of buckling is controlled by an offset load.

10.5.5.2 Comments: Stress singularities along sharp re-entrant edges are eliminated by the elastic–perfectly plastic material. The maximum stress is limited to the yield strength of material.

The following are the required steps:

1. Define the elastic–perfectly plastic material properties (1060 alloy).
2. Apply a fixed support to the bottom face of the base plate (Figure 10.16).
3. Apply a load uniformly distributed over the blue face (Figure 10.16).
4. Mesh and solve the linear buckling analysis.
5. Define the displacement sensor in the center of the top face of the top plate.
6. Solve the nonlinear buckling analysis.
7. Construct the load–displacement curve with the sensor in the center of the top plate in the direction of buckling.
8. Compare the results of linear buckling analysis and nonlinear buckling analysis.

10.5.6 CURVED_SHEET (Figure 10.17)

Model name

- 10.05.CURVED_SHEET.x_t
- 10.05.CURVED_SHEET.sldprt

10.5.6.1 Objective: Perform a nonlinear buckling analysis on a model where the onset of buckling is controlled by a curvature of model geometry.

10.5.6.2 Comments: You may take advantage of model symmetry and work with one half of geometry. If you use Parasolid geometry, cut the model into half. If you use SOLIDWORKS geometry, switch to configuration *02 half*.

The following are the required steps:

1. Define the elastic material properties (alloy steel).
2. Surface geometry requires shell element meshing, define shell thickness 2mm.
3. Define restraints as shown in Figure 10.17.
4. Apply a normal load uniformly distributed load over the top edge (Figure 10.17).
5. Mesh and run the linear buckling analysis.
6. Define the displacement sensor in the middle of the loaded edge.
7. Perform the nonlinear buckling analysis.
8. Construct the load–displacement curve.
9. Compare the results of linear buckling analysis and nonlinear buckling analysis.

Chapter 11
Vibration Analysis

All types of analyses that we have discussed so far assumed that loads are static; they do not change with time and inertial effects are ignored.

We will now lift the restriction of static load to discuss common types of vibration analysis, but first we need to clarify an important terminology issue. Many commercial FEA programs use the term "dynamic analysis" to describe what we call "vibration analysis" in this book.

The term "dynamic analysis" is more general than "vibration analysis." It applies to analysis of any motion be it motion of a rigid body, an deformable body, or an assembly composed of rigid and deformable bodies. "Dynamic analysis" within the scope of FEA deals with vibrations of deformable bodies about the position of equilibrium; hence, the term "vibration analysis" is more precise.

Vibration analysis is a very broad topic and cannot be given an in-depth treatment in this introductory textbook. For this reason, we will discuss only two commonly performed types of linear vibration analysis: time response analysis and frequency response analysis and then we will introduce basic concepts of nonlinear vibration analysis.

11.1 Modal Superposition Method

The review of linear vibration analysis needs to be preceded by discussion of the modal superposition method on which both time response and frequency response analyses are based.

Let us summarize what we already know about modes of vibration from chapter 9. Each mode is characterized by frequency and shape; Each mode is also characterized by mass participating in vibration, this is called modal mass participation. Furthermore, the modal shape is associated with certain stiffness. Collecting these facts, we may represent an object vibrating in a given mode of vibration by a linear single degree of freedom

(SDOF) oscillator with mass, stiffness, and direction that may be derived from the properties of the mode of vibration under consideration.

Having realized that any particular mode of vibration may be represented by a purposely designed single degree of freedom oscillator (SDOF), we now decide how many modes are important in the vibration response of the system under investigation. If, for example, only three modes are important, we can represent the system with three SDOFs, each one corresponding to one mode. This way, the number of degrees of freedom (DOF) of the vibrating system is reduced to just three, no matter how complex the model is. Instead of studying an original model with tens or hundreds thousands of DOF, we may now study a simple 3-DOF system where each mode is represented by an SDOF and a response of the system is found as a superposition of individual responses of SDOFs. The concept of the modal superposition method is schematically illustrated in Figure 11.1.

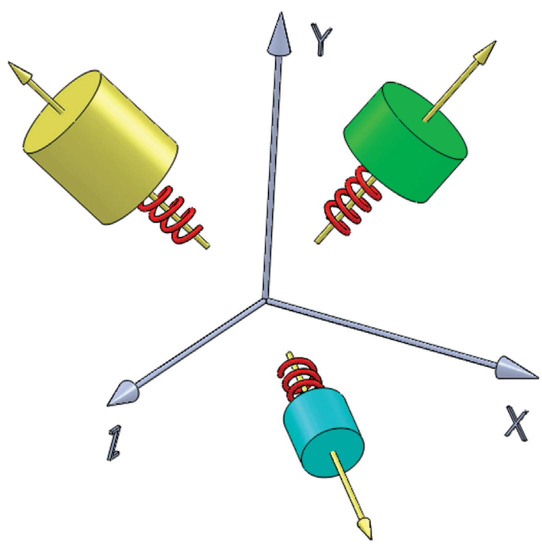


Figure 11.1 A vibration system represented by three linear SDOF oscillators. It is assumed here that three modes of vibration are sufficient to model the vibration response. Each mode is represented by an SDOF oscillator. Each oscillator is characterized by its mass, stiffness, and direction of oscillations.

When vibration analysis is based on the modal superposition method, the number of SDOFs contributing to vibration response is equal to the number of modes calculated by prerequisite modal analysis. How many modes should be used to represent vibration response using the modal superposition method? The first few modes are most important, but the exact number of required modes is not known prior to the analysis. One should demonstrate in a convergence process that increasing the number of modes past certain number no longer significantly affects results.

The modal superposition method offers a very significant simplification of vibration analysis, but in order for the modal superposition method to work, the vibration problem must be linear. The modal superposition method cannot be used for nonlinear vibration problems. Other methods, like direct integration method, are used in nonlinear vibration problems.

11.2 Time Response Analysis

In time response analysis, the applied load is an explicit function of time. Mass and damping properties are both taken into consideration and the equation of motion appears in its full form:

$$[M]\ddot{d}+[C]\dot{d}+[K]d=[F(t)] \tag{11.1}$$

where

$[M]$—mass matrix—known

$[C]$—damping matrix—known

$[K]$—stiffness matrix—known

$[F(t)]$—time dependent vector of nodal loads—known

$[d]$—vector of nodal displacements—unknown

Time response analysis, therefore, requires definition of damping, most often expressed as a percentage of critical damping. Damping must be defined for all modes used in the modal superposition method. Readers are referred to (1) in chapter 15 for values of damping coefficients.

A first example of time response analysis is a response to a load of short duration called impact load. We illustrate this with a hammer model shown in Figure 11.2. The hammer head is subjected to an impact force for which time history is shown in Figure 11.3. The vibration time response in the form of head displacements is shown in Figure 11.4.

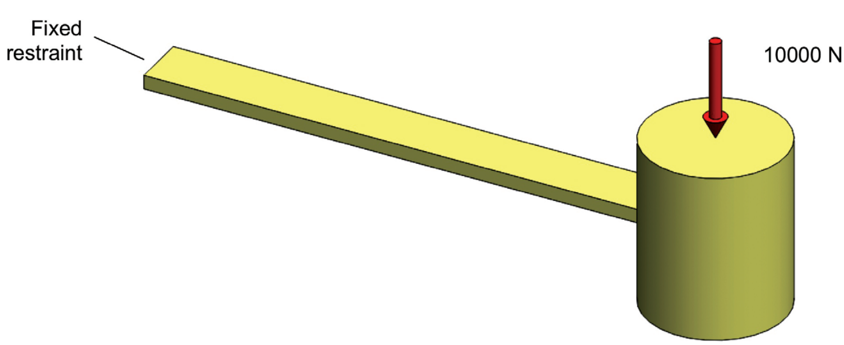


Figure 11.2 A hammer subjected to time-dependent load; the load changes with time; the maximum load magnitude is 10000 N.

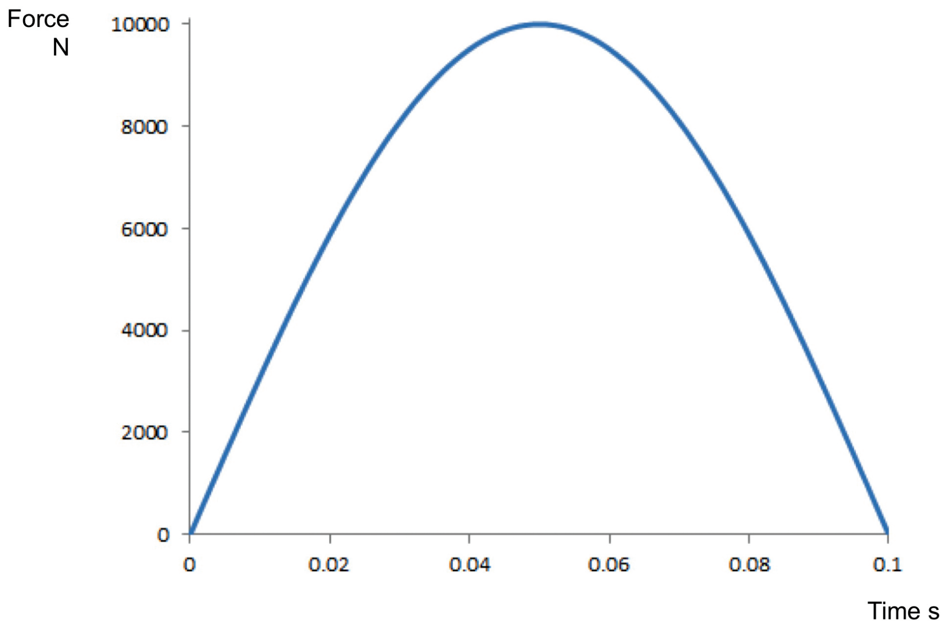


Figure 11.3 Load time history; the load follows a sine curve, after 0.05 s, it reaches the maximum magnitude of 10000 N and disappears after 0.1 s.

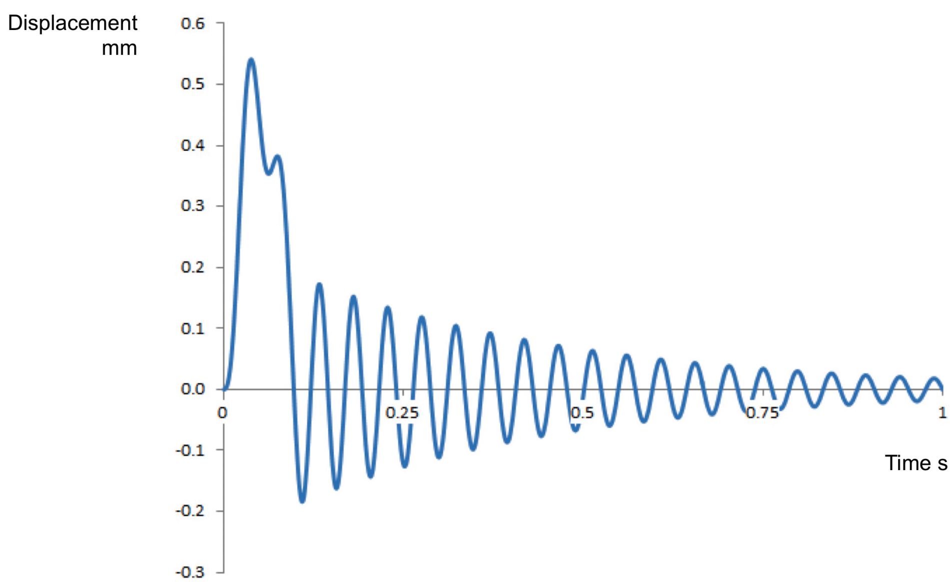


Figure 11.4 Displacement time history of the hammer head subjected to an impact load. Notice that after 0.1 s when the load has disappeared, the hammer enters into free damped vibration.

The second example presents time response analysis where an excitation load has a longer duration. The same model of hammer (Figure 11.2) is subjected to a harmonic load as shown in Figure 11.5; there is no damping in the system. The frequency of the excitation load is 19 Hz, which is close to the first natural frequency 21 Hz of the hammer. The displacement response of the hammer head shown in Figure 11.6 demonstrates the well-known effect of beating, which is a periodic amplification and cancellation of displacement amplitude.

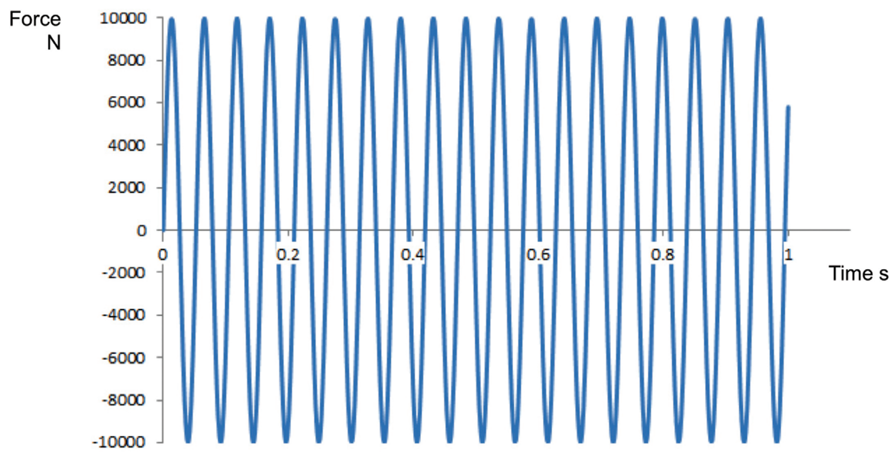


Figure 11.5 Time history of the excitation load is a harmonic function with the frequency 19 Hz and the maximum load magnitude is 10000 N.

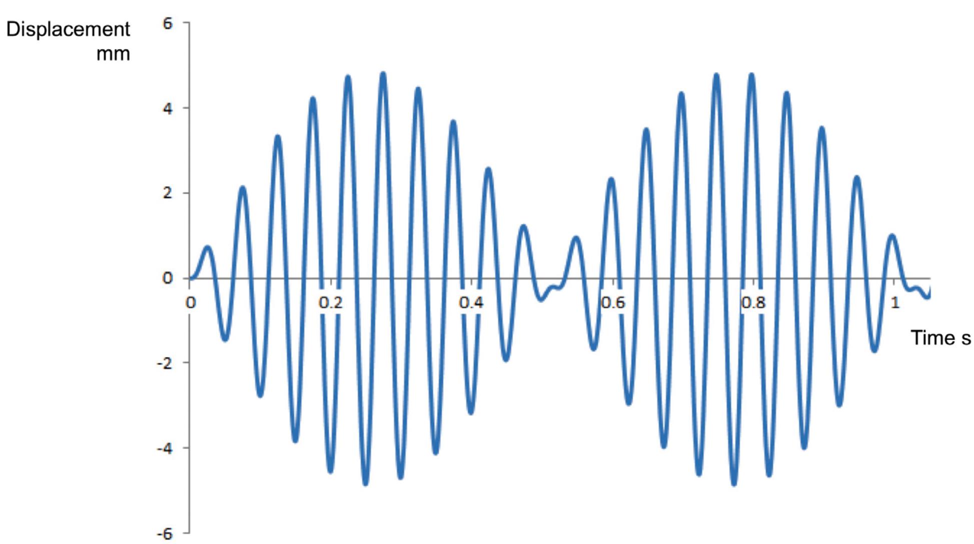


Figure 11.6 Displacement time history of the hammer head subjected to harmonic excitation load; there is no damping in the system. Notice periodic amplification and cancellation of displacement amplitude measured in the direction of load.

11.3 Frequency Response Analysis

Frequency response analysis assumes that the excitation is harmonic and the magnitude of excitation is a function of frequency rather than being directly dependent on time as was the case in time response analysis:

$$[M]\ddot{d} + [C]\dot{d} + [K]d = F(\omega) \tag{11.2}$$

Frequency response models structure response to load excitation or base excitation; base excitation is applied to supports as a displacement velocity or acceleration. Even though base excitation may be used in the previously discussed time response analysis, it is much more common in frequency response analysis. Both load excitation and base excitation must be harmonic functions of time. Frequency response analysis also uses the modal superposition method and requires damping be defined for each mode considered in the modal superposition.

We will review two typical applications of frequency response. The first one is a shaker table test simulation. An elbow pipe is installed on a shaker table and subjected to a harmonic base excitation with frequency slowly changing from 0 to 800 Hz in the direction shown in Figure 11.7. There are four modes of vibration within this range of frequencies; modal shapes and frequencies are shown in Figure 11.8.

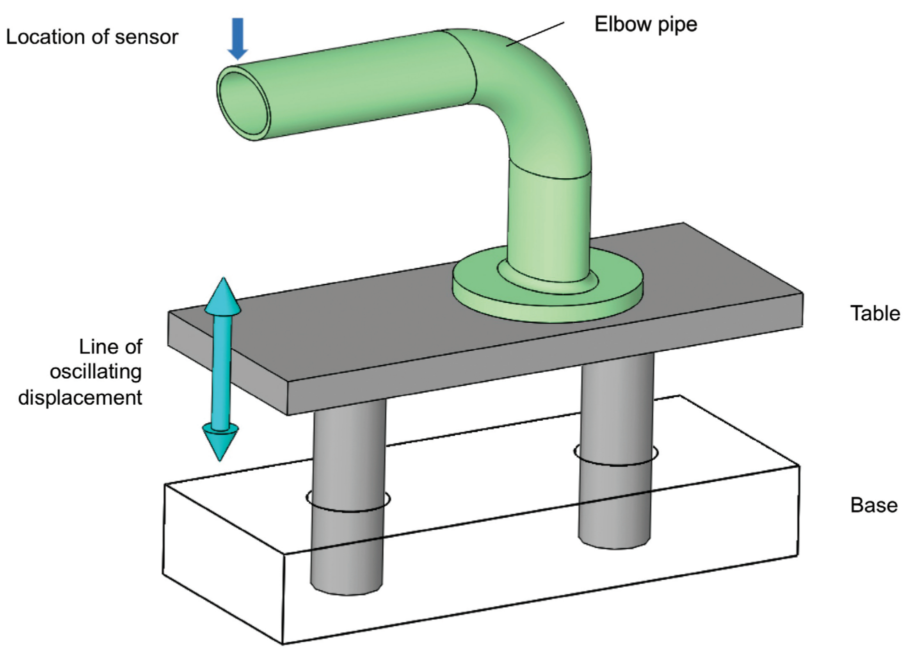


Figure 11.7 An elbow pipe is installed on a shaker table, which performs oscillating motion relative to base. The frequency of oscillation changes within the range 0–800 Hz.

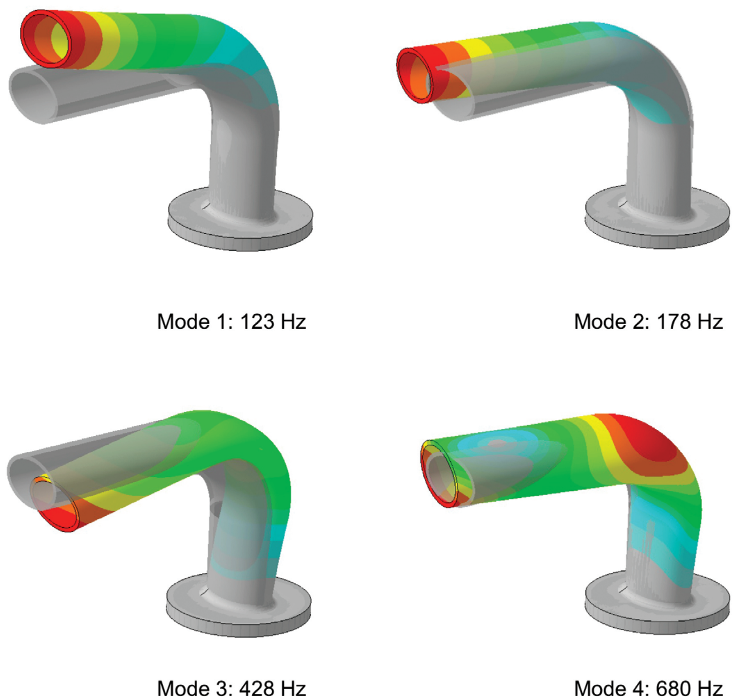


Figure 11.8 Four modes of vibration found in the frequency range of interest 0–800 Hz.

The base excitation is defined as an oscillating displacement of constant amplitude 1 mm (±1 mm about the position of equilibrium) slowly changing the frequency of oscillation from 0 to 800 Hz. Damping is defined as 2% of modal damping for all modes. The summary of results is shown in Figure 11.9 where two resonant frequencies can be seen clearly.

As shown in Figure 11.9, only two modes are excited by the vertical oscillations of the table: these are modes 1 and 3. Modes 2 and 4 have shapes orthogonal to the direction of excitation and can not be excited by vertical motion of the shaker table.

Notice that when the excitation frequency passes through the resonant frequency, the amplitude of vibrations reaches its local maximum. Furthermore, vibration at the frequency of excitation equal to the natural frequency is controlled only by damping.

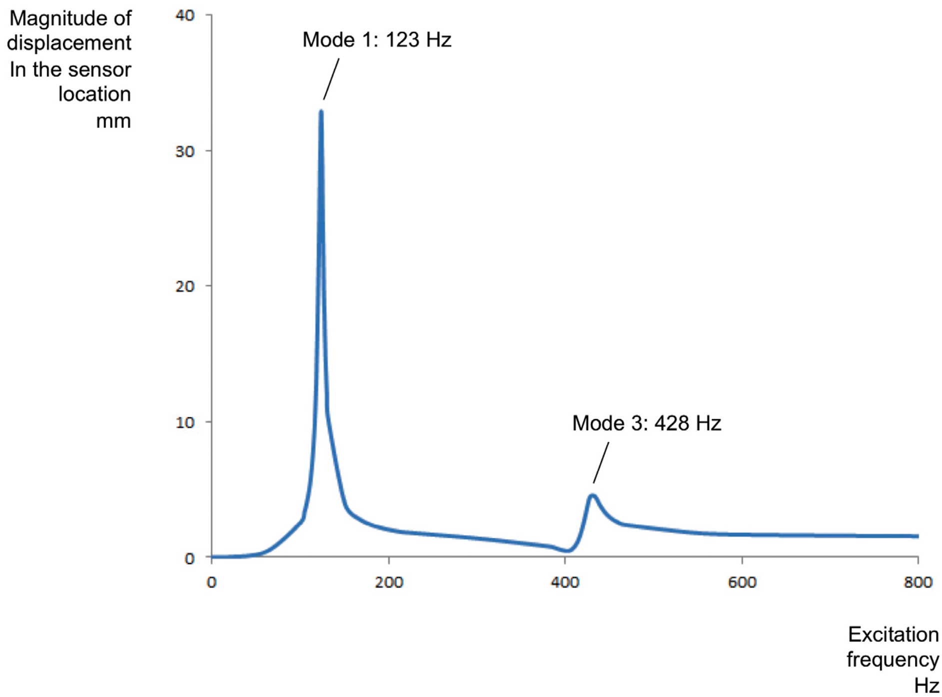


Figure 11.9 The result of frequency response analysis; modes 1 and 3 are excited by base excitation in the direction shown in Figure 11.7.

Frequency response analysis is not limited to base excitation with displacement of constant amplitude. Any load or base excitation may be used as long as it is harmonic and its magnitude is a function of the excitation frequency. The second example illustrates frequency response analysis with variable magnitude of excitation. This is vibration analysis of a centrifuge where the magnitude of excitation load is a square function of the excitation frequency. Such an excitation is called omega square excitation; it is a common type of excitation in rotating machinery.

Figure 11.10 shows a schematic model of an industrial centrifuge. The magnitude of an out of balance centrifugal load is proportional to ω^2, where ω is the angular velocity of drum in radians per second. During the operation, the angular velocity changes in the range 0–200 rad/s and the load magnitude in the range 0–25000 N (Figure 11.11). The frequency response analysis is used to find the amplitude of displacement of drum as a function of the angular velocity of drum.

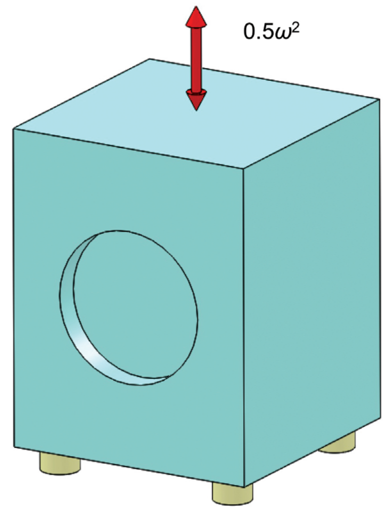


Figure 11.10 A centrifuge is subjected to a vertical oscillating load, where the load magnitude is not constant; it changes proportionally to ω^2, where ω is the excitation frequency in radians per second. Guide rails, not shown is this illustration, eliminate any horizontal load components. The body is supported by four rubber legs. The body is very stiff compared with the legs; therefore, the displacement sensor may be placed anywhere on the body.

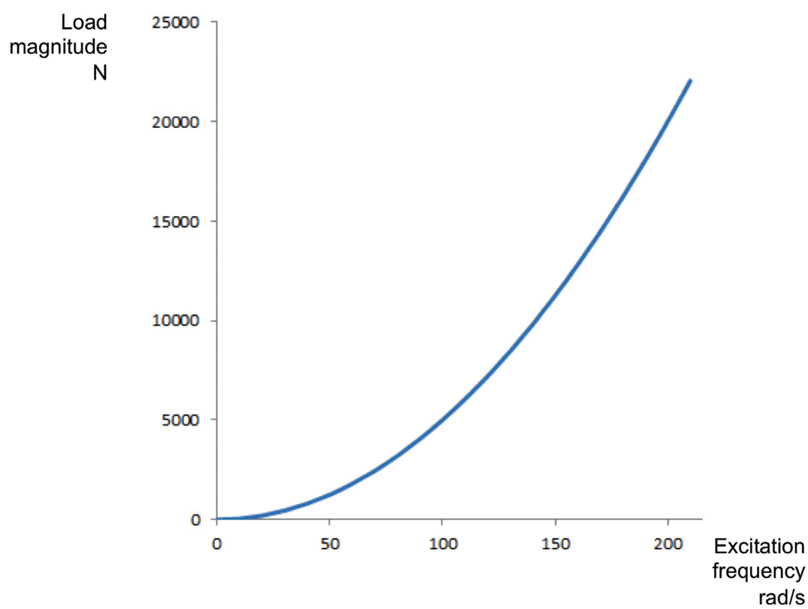


Figure 11.11 The magnitude of excitation load as a function of the excitation frequency. In the case of the centrifuge, the excitation frequency is the angular velocity of the drum. The curve equation is $F = me\omega^2$, where m = 5 kg is out of balance mass, e = 0.1 m is eccentricity, and ω is angular velocity.

The displacement response graph reveals one resonant frequency 9.6 Hz corresponding to the only mode of vibration in the excitation range of 0–35 Hz (Figure 11.12).

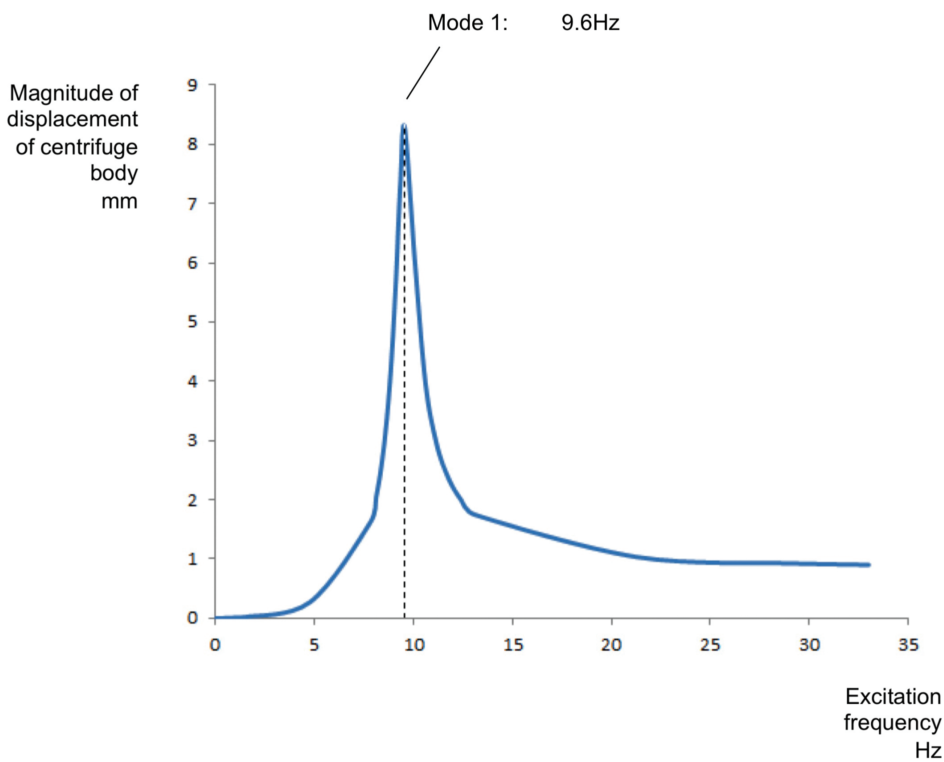


Figure 11.12 The magnitude of displacement of the centrifuge in the range of the operating frequencies 0–35 Hz; only one resonant frequency can be seen.

11.4 Nonlinear Vibration Analysis

Time response analysis and frequency response analysis both belong to the class of linear vibration analysis where the structure stiffness does not change with time (time response) or with frequency (frequency response). Being linear, these types of analyses are based on the modal superposition method leading to a very significant reduction in numerical effort required to obtain solution. A vast majority of vibration problems are linear; this is because displacements are small and material remains in the linear range.

The analysis of nonlinear vibration problems where stiffness does change with time faces two major challenges; both originate in the inability to use the modal superposition method for nonlinear vibration. First and most importantly, the number of DOF in the vibration response cannot be reduced to the number of modes considered in analysis.

The equations of motion must be based on the full set of DOF in the model. This means that in place of solving some three, four of five (for example) equations of motions in a linear vibration problem, we must solve as many equations of motion as the number of DOF in the analyzed problem.

Another consequence of the inability to use the modal superposition method is that we cannot use modal damping. Damping in nonlinear problems must be defined as Rayleigh damping. Rayleigh damping makes an arbitrary assumption that the damping matrix is a linear combination of the mass and stiffness matrices. This assumption is a mathematical convenience for the purpose of simplification as there is no physical justification for this. Rayleigh damping is specified by two damping constants: α and β, which are used as multipliers of the mass matrix M and the stiffness matrix K, respectively, when calculating damping matrix C

$$[C] = \alpha[M] + \beta[K] \tag{11.3}$$

$$\frac{\alpha}{2\omega} + \frac{\beta\omega}{2} = \zeta \tag{11.4}$$

where ω is the frequency and ζ is the damping ratio.

Alpha damping α is a viscous damping component also known as mass damping. It characterizes damping at lower frequencies. Beta damping β is a hysteresis damping component also known as solid or stiffness damping. It characterizes damping at higher frequencies.

Nonlinear vibration analysis is conducted in time domain; it is always time response analysis; equations of motions are explicit functions of time.

Examples of nonlinear vibration analysis include dynamic contact between two bodies or oscillations causing significant changes to stiffness of vibration structure.

An example of a nonlinear vibration problem is shown in Figure 11.13. A beam is held between two immovable hinges and subjected to a uniformly distributed pressure of short duration and time-dependent magnitude (Figure 11.14). The source of nonlinearity in this problem is membrane stresses that cause beam stiffness to increase during the process of deformation. A comparison between correct nonlinear solution and incorrect linear solution is shown in Figure 11.15.

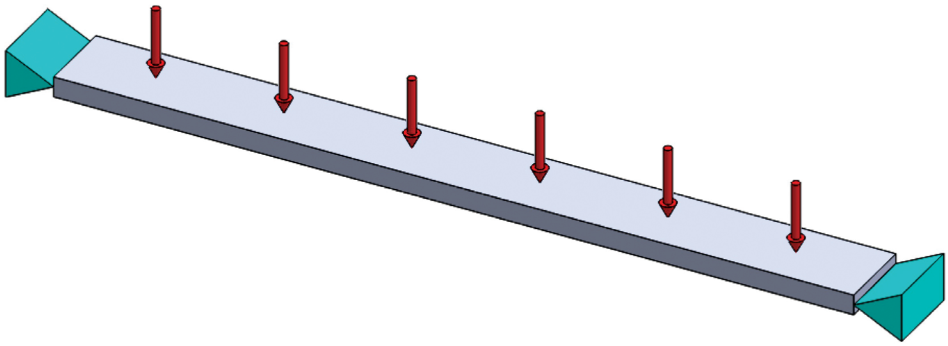


Figure 11.13 The beam held between two hinges and subjected to pressure is a classic example of nonlinear problem because of stress stiffening effect.

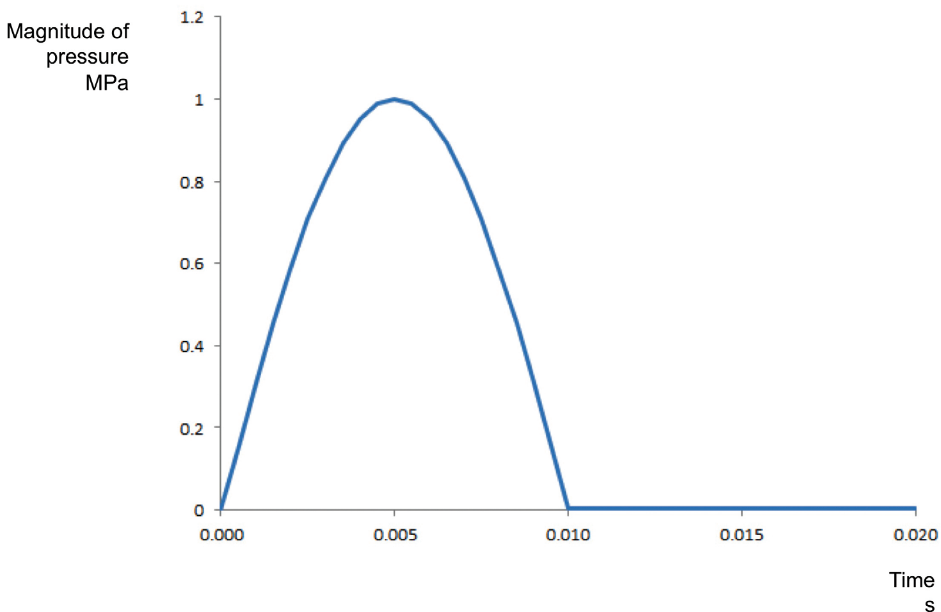


Figure 11.14 Time history of pressure magnitude applied to beam. The pressure follows a sine curve; it reaches 1 MPa and then drops down to zero within 0.01 s.

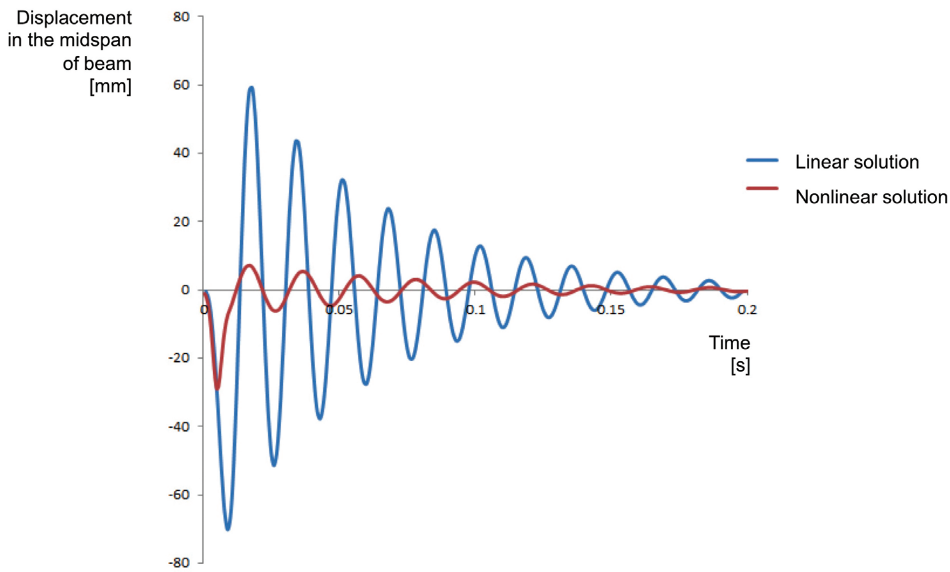


Figure 11.15 The displacement response in the midspan of the beam in linear and nonlinear solutions. Neglecting the nonlinear effect causing beam stiffening leads to over 200% error in the maximum displacement amplitude.

11.5 Hands-On Exercises

11.5.1 Hammer Impulse Load (Figure 11.2)

Model name

- 11.01.HAMMER. x_t
- 11.01.HAMMER. sldprt

11.5.1.1 Objective: Perform a time response analysis of a model subjected to an impact load.

11.5.1.2 Comments: The definition of time response analysis includes specifying a time step; make sure it is short enough to model response based on all modes in the 0–800-Hz range.

The following are the required steps:

1. Define the linear elastic material properties (steel).
2. Apply a fixed support as shown in Figure 11.2.
3. Apply a normal uniformly distributed load over the top face of hammer head (Figure 11.2) with time history shown in Figure 11.3.
4. Define the displacement sensor in the center of the loaded face.
5. Specify all modes in the range of 0–800 Hz to be used for the modal superposition method.

6. Apply 2% damping to all modes.
7. Mesh and run time response analysis for 1 s.
8. Construct displacement time history in the sensor location for the duration of analysis (Figure 11.4).

11.5.2 Hammer Beating (Figure 11.2)

Model name

- 11.01.HAMMER.x_t
- 11.01.HAMMER.sldprt

11.5.2.1 Objective: Perform a time response analysis of a model subjected to harmonic load with the frequency of excitation close to the first natural frequency. This will demonstrate the phenomenon of beating.

11.5.2.2 Comments: When the first analysis is complete, run analysis again using excitation with slightly different frequencies. Observe that when the excitation frequency approaches the natural frequency, the excitation period increases.

1. Define the material properties and restraints, the same as in the exercise in Section 11.5.1.
2. Mesh and run to find the first mode of vibration; verify that its frequency is 21 Hz.
3. Apply a normal uniformly distributed load over the top face of hammer head (Figure 11.2) with time history shown in Figure 11.5; the frequency of excitation is 19 Hz.
4. Specify only one mode (the first mode of vibration) to be used in the modal superposition method.
5. Apply 2% damping.
6. Specify small time step; 1/50 of the vibration period in the first mode.
7. Run time response analysis for 1 s.
8. Construct displacement time history in the sensor location for the duration of analysis (Figure 11.6).

11.5.3 ELBOW_PIPE (Figure 11.7)

Model name

- 11.02.ELBOW_PIPE.x_t
- 11.02.ELBOW_PIPE.sldprt

11.5.3.1 Objective: Perform a frequency response analysis of a model subjected to a harmonic base excitation with frequency range 0–800 Hz. This simulates a shaker table test.

11.5.3.2 Comments: This exercise is done on part, not on the assembly model shown in Figure 11.7:

1. Define the material properties of gray cast iron.
2. Define restraint to the flange to simulate support provided by the shaker table.
3. Define the sensor in the location shown in Figure 11.7.
4. Define displacement base excitation in the direction shown in Figure 11.7 and a displacement amplitude of 1 mm.
5. Specify four modes to be used in the modal superposition method.
6. Apply 2% damping to all modes
7. Mesh and run frequency response analysis in the range 0–800 Hz.
8. Construct a displacement amplitude graph in the sensor location for the excitation range (Figure 11.9).

11.5.4 Centrifuge (Figure 11.10)

Model name

- 11.03.CENTRIFUGE.x_t
- 11.03.CENTRIFUGE.sldasm

11.5.4.1 Objective: Perform frequency response analysis of a model subjected to omega square force excitation. This simulates excitation with an out-of-balance load. The load magnitude is proportional to ω^2, where ω is the angular velocity in radians per second.

11.5.4.2 Comments: Rubber legs are small in comparison with the centrifuge body. Define mesh controls on legs to make sure that elements are correctly shaped. Highly distorted elements will model rubber leg stiffness incorrectly; stiffness will be too high.

1. Apply the material properties of plain carbon steel to the centrifuge body.
2. Apply the material properties of rubber (modulus of elasticity 6.1MPa) to four legs.
3. Define the sensor anywhere on the centrifuge body.
4. Define fixed restraints to the bottom faces of the four rubber legs.
5. Define slider restraints to side faces of the centrifuge body; this will limit the centrifuge movement to oscillations in the vertical direction.
6. Specify one mode to be used by the modal superposition method.
7. Apply load to the centrifuge body in the vertical direction; the load is a function of angular velocity, as shown in Figure 11.11.
8. Define damping as 5% of critical damping.
9. Mesh and run frequency response in the range 0–35 Hz.
10. Construct the displacement response graph as shown in Figure 11.12.

11.5.5 PLANK (Figure 11.13)

Model name

- 11.04.PLANK.x_t
- 11.04.PLANK.sldprt

11.5.5.1 Objective: Perform linear and nonlinear vibration analyses of the PLANK model subjected to an impact load. The exercise can be completed using solid elements or shell elements. If you use shell element, construct a surface 1000 × 100 mm; define the shell element thickness of 25.4 mm.

11.5.5.2 Comments: This exercise is done on part, not on the assembly model shown in Figure 11.13.

Time step has to be defined in linear and nonlinear analyses; use a time step of 0.001 s.

The following are the required steps:

1. Apply the material properties of steel.
2. Define a sensor in the midspan of beam.
3. Define hinge supports to two short ends.
4. Define time-dependent pressure as shown in Figure 11.14.
5. Specify two modes to be used in the modal superposition method.
6. Define damping 5% for the first mode and 10% for the second mode.
7. Run linear time response analysis for the duration of 0.2 s.
8. Define Rayleigh damping with parameters: α = 20.17 and β = 0.00012.
9. Run nonlinear time response analysis for the duration of 0.2 s.
10. Construct the displacement response graph comparing the results of linear and nonlinear analyses (Figure 11.15).

Chapter 12
Thermal Analysis

Thermal analysis deals with heat transfer in solid bodies. The primary unknown in thermal analysis is temperature, which is a scalar entity. Therefore, only 1 DOF needs to be assigned to nodes of a thermal finite-element model regardless of the dimensionality of analysis, be it 3D, 2D, or 1D. This makes thermal analysis simpler than structural analysis in terms of the computational effort required to obtain solution.

An important conceptual difference between structural and thermal analysis is that while structural static analysis deals with the state of equilibrium under applied load, the analogous thermal analysis does not describe the state of equilibrium. Instead, it models the steady-state condition where heat flow continues but does not change in time. Therefore, thermal analogy of static analysis is steady-state thermal analysis and the analogy of dynamic (vibration) structural analysis is transient thermal analysis. The temperature is analogous to displacement in structural analysis and there are other analogies summarized in Table 12.1. Because of those close analogies, experience in structural analysis is directly transferable to thermal analysis.

Table 12.1 Analogies between structural analysis and thermal analysis; SI units are used

Structural	Thermal
Displacement [m]	Temperature [K]
Strain [1]	Temperature gradient [K/m]
Stress [N/m^2]	Heat flux [W/m^2]
Load [N] [N/m^2] [N/m^3]	Heat source or heat sink [W] [W/m^2] [W/m^3]
Prescribed displacement [m]	Prescribed temperature [K]
Elastic support [N/m]	Convection coefficient [W/(m^{2}K)]

12.1 Heat Transfer Induced by Prescribed Temperatures

Just like stress may be induced by load or by prescribed displacements, heat flow may be induced by a heat load or by prescribed temperatures. Consider the model with different

temperatures defined on each of two faces, as shown in Figure 12.1. Notice that temperature field establishes itself in the model, but heat flow continues because of temperature gradients. No heat is exchanged through faces other than the two faces with prescribed temperatures. This implies that the model is perfectly insulated, except for two faces where prescribed temperatures are defined. The only mechanism of heat transfer in problem illustrated in Figure 12.1 is heat conduction in the solid body.

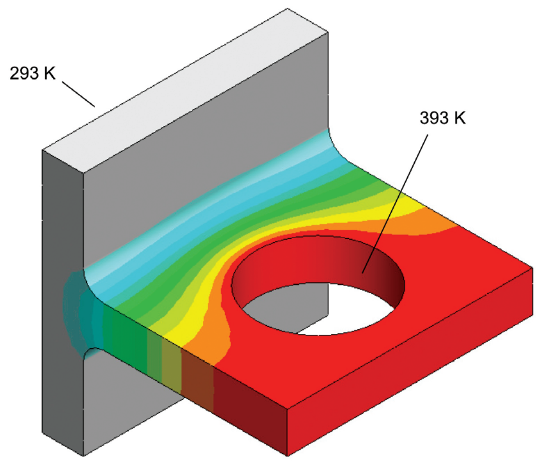


Figure 12.1 Temperature distribution in a model subjected to prescribed temperatures: 293 K (20°C) to the back face and 33 K (120°C) to the cylindrical face of the hole. The two faces with prescribed temperatures defined are the only faces participating in heat exchange between the model and the environment. No heat is exchanged through other faces. This may correspond to a situation where the bracket holding a hot pipe is mounted on a cold surface. There is only one mechanism of heat transfer, which is conduction in the solid body.

12.2 Heat Transfer Induced by Heat Power and Convection

Heat flow can be also induced by the heat load also called heat power.

Figure 12.2 presents a problem where heat is generated in a volume; this is called heat power and is in a close analogy to volume load such as gravity or centrifugal load. Heat power may be defined as total heat power [W] or heat power per volume [W/m^3].

In every heat transfer problem, a mechanism of heat flow must be fully defined. In the case of microchip assembly in Figure 12.2, heat travels to the radiator and is dissipated to environment through faces of radiator by convection. Convection is modeled in finite element analysis as boundary conditions. Convection boundary conditions require the definitions of convection coefficients [W/m^2/K], also called film coefficients and temperature of the surrounding fluid called ambient temperature [K]. Convection requires fluid; it cannot happen in vacuum.

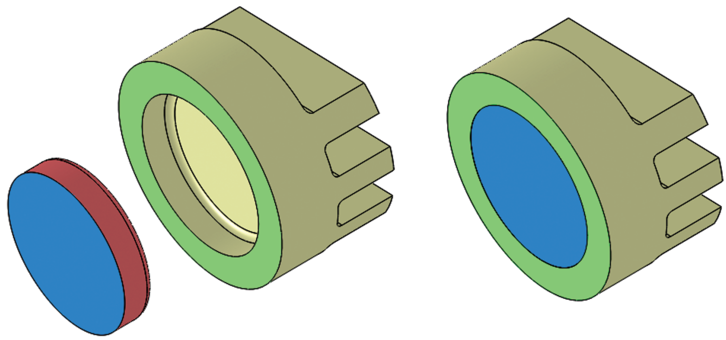
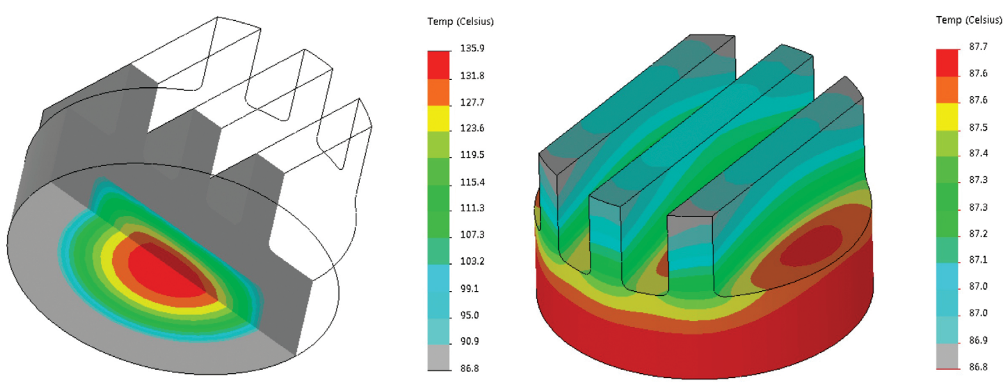


Figure 12.2 Microchip and radiator assembly are shown in exploded and collapsed views. The bottom face of the microchip (blue) and the bottom face of the radiator (green) are insulated. Heat generated in the microchip travels by conduction to the radiator; it is then dissipated by convection to the surrounding fluid. It is assumed that heat crosses the border between the microchip and the radiator without encountering any resistance. There are two mechanisms of heat transfer present: conduction in solid bodies and convection between the faces of solid bodies and the surrounding fluid, which in this case is air.

Convection coefficients are defined on all faces participating in removal of heat from the model. The temperature field that establishes itself in the microchip assembly model is shown in Figure 12.3. As this is a steady-state thermal analysis, we do not know how long it took for the temperature field to reach this steady state.



Temperature distribution in assembly

Temperature distribution in radiator

Figure 12.3 Temperature distribution in the assembly shows a hot spot in the microchip and the variation of temperatures within assembly is 49°C. The temperature in the radiator shows a very low temperature variation of 0.9°C.

Figure 12.4 shows a problem that can be solved using a 2D representation if there is no temperature gradient along the length on the channel; no temperature gradient requires end faces to be insulated.

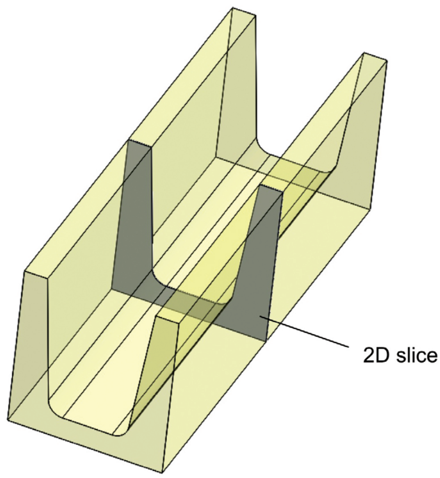


Figure 12.4 A long channel is heated at the bottom face where surface heat power is defined. Heat enters the model through the bottom face, travels through the model by conduction, and escapes to environment through faces where convective boundary conditions are defined. Assuming that end faces are insulated, there is no temperature gradient along the length of the channel and, therefore, the problem may be solved using a 2D representation.

Heat enters the model through the base where surface heat power is defined. Heat escapes the model through faces where convection coefficients are defined. The established heat flow produces temperature distribution and heat flux, as shown in Figure 12.5.

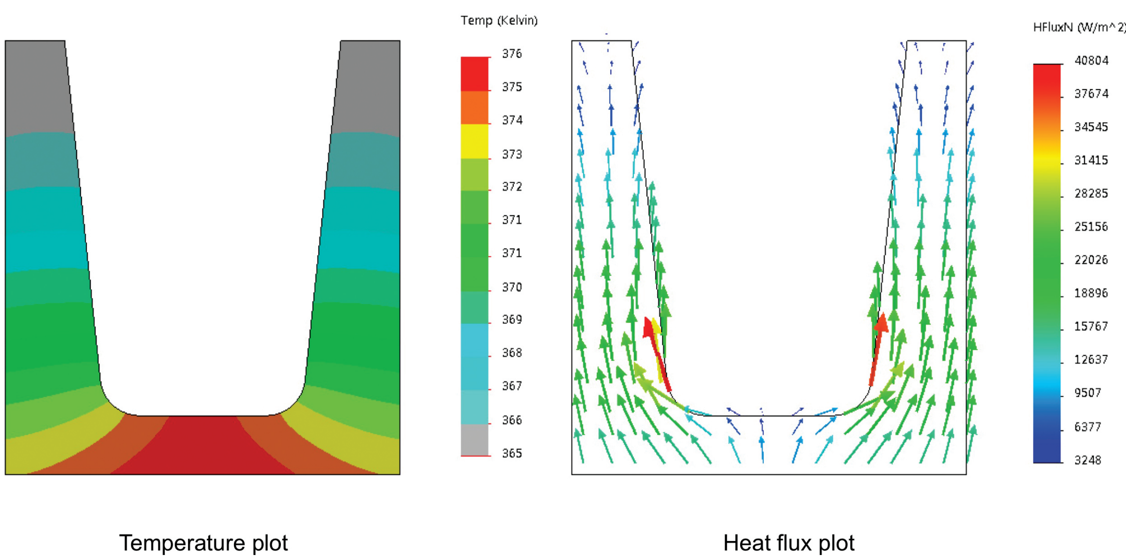


Figure 12.5 Temperature and heat flux results. The heat flux plot uses vector display. Notice arrows “coming out” of the walls; this visualizes heat escaping through the walls. Heat dissipation is enabled by convective boundary conditions.

Convective boundary conditions are analogous to elastic support offered by distributed springs. Just like supports and/or prescribed displacements are necessary to establish model equilibrium in structural analysis problem, convective boundary conditions and/or prescribed temperatures are necessary to establish heat flow in thermal analysis problem. An attempt to run thermal analysis with heat loads but without means of removing heat from the model such as convection or prescribed temperature, results is an error similar to the one caused by the absence of supports in structural analysis.

12.3 Heat Transfer by Radiation

Figure 12.6 shows a space heater where all three mechanism of heat transfer must be modeled: conduction, convection, and radiation. The bulb is heated from inside; a small portion of heat enters the reflector through the face touching reflector, but most of heat travels across the bulb thickness to the outside faces where it is dissipated by radiation. Some heat directly escapes to environment, some hits the reflector. Now, considering heat that reaches the reflector by radiation, a portion of that heat is reflected back and radiated to environment (it may also hit the bulb again), while some heat enters the reflector, travels across the solid body, and is dissipated by convection on the convex side of the reflector.

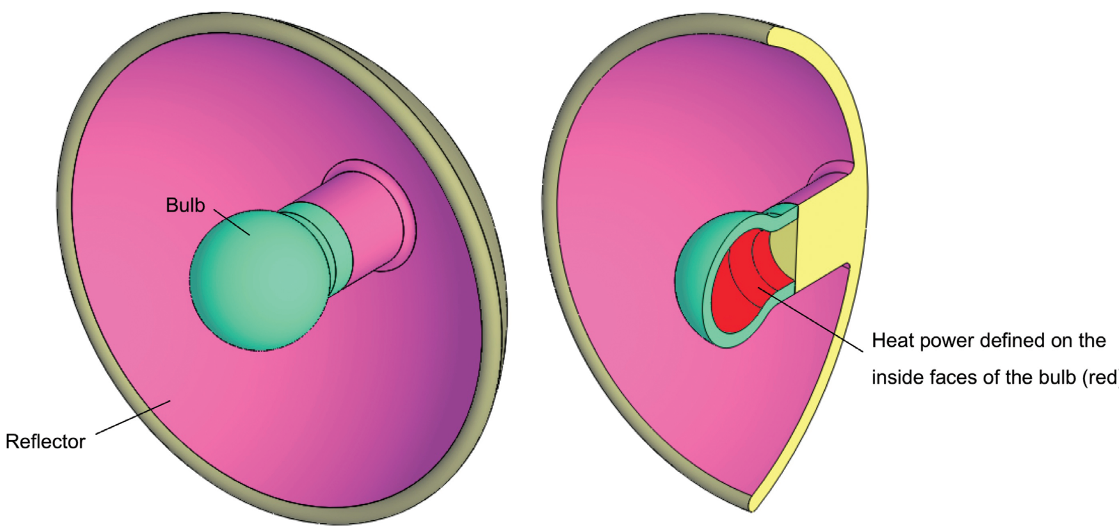


Figure 12.6 The outside faces of the bulb (green) and the reflector (magenta) are in vacuum; they exchange heat by radiation between themselves and between themselves and the environment. The back face of the reflector (gray) is exposed to air and dissipates heat by convection. Heat power is defined on the inside faces of the bulb (red). There are three mechanisms of heat transfer here: conduction responsible for heat transfer inside solid bodies, radiation responsible for heat exchange between faces of solid body and environment and between faces of solid themselves, and convection responsible for heat exchange between the face of the solid and the surrounding fluid (back side).

In this problem, only heat transfer by conduction is modeled directly; radiation and convection are modeled as boundary conditions. The steady-state temperature plot in the space heater is shown in Figure 12.7.

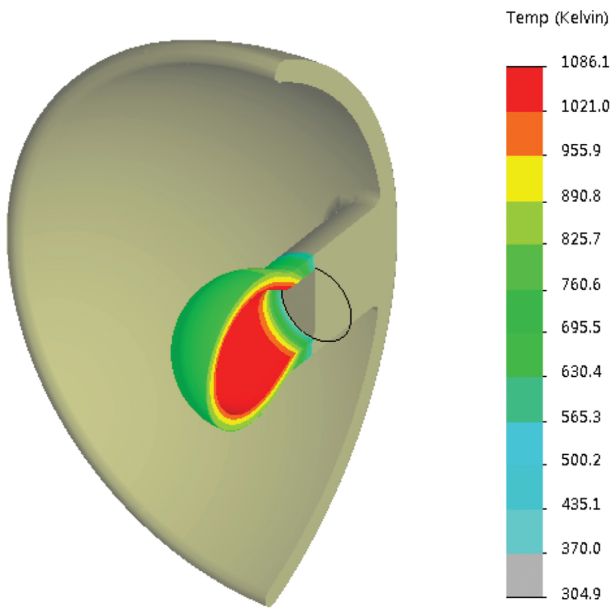


Figure 12.7 The steady-state temperature plot in the space heater assembly.

12.4 Modeling Considerations in Thermal Analysis

Symmetry boundary conditions can be used in thermal analysis based on the observation that if symmetry exists in both geometry and boundary conditions, then there is no heat flowing through a plane of symmetry. Therefore, after simplifying the model to half or quarter in the case of double symmetry (Figure 12.8), nothing needs to be done to faces created by cuts. No convection coefficients defined on faces means created by symmetrical cut automatically enforces symmetry of heat flow. Symmetry boundary condition are not applicable to radiation problems because of their 3D nature.

Examine the model shown in Figure 12.9 (left) and notice sharp re-entrant edges. This model is suitable for analysis of temperature distribution, but because of sharp re-entrant edges, it is not suitable for analysis of heat flux in the vicinity of sharp re-entrant corners because heat flux is singular in sharp re-entrant corners. This is in direct analogy to sharp re-entrant edges causing stress singularities in structural models.

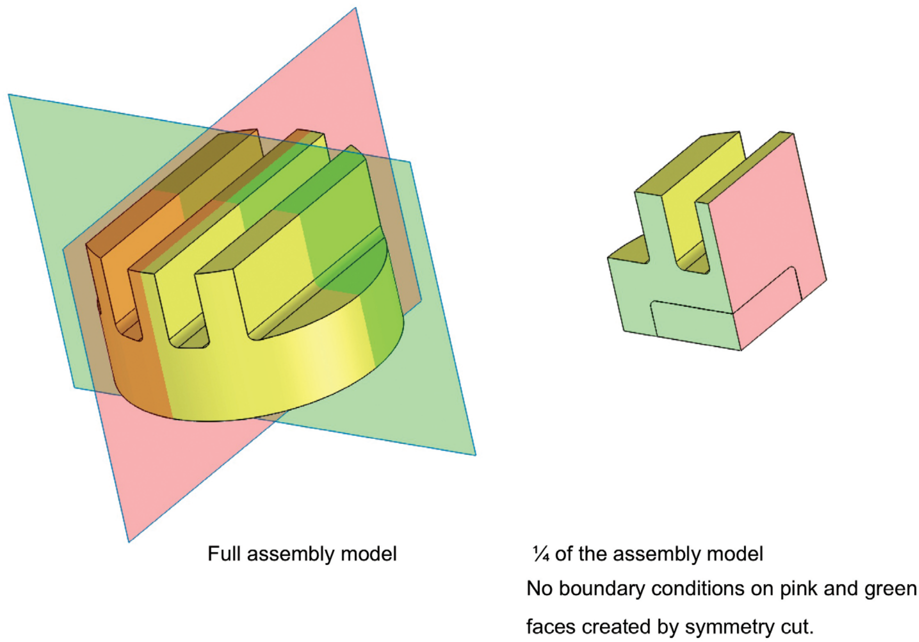


Figure 12.8 The microchip and radiator assembly have double symmetry in terms of geometry and convective boundary conditions. There is no heat flow across the pink and green planes. This double symmetry may be exploited and problem be represented by quarter of the model. To model symmetry boundary condition, do not define any thermal conditions on the faces created by symmetrical cuts. In the absence of any thermal boundary conditions, the faces in the plane of symmetrical cut will be insulated and there will be no heat flow across those faces.

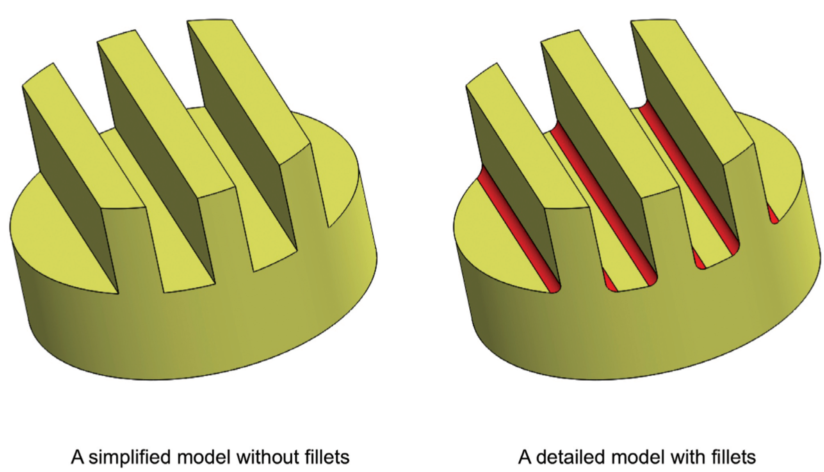


Figure 12.9 A model without fillets (left) may be used for analysis of temperature but is not suitable for the analysis of heat flux because sharp re-entrant edges produce heat flux singularity. The analysis of heat flux requires a detailed modeling of geometry to include fillets (right) even if they are small.

12.5 Challenges in Thermal Analysis

Thermal analysis may be seen as easy to setup and execute. Direct analogies between structural and thermal analyzes make thermal analysis easy to execute; 1 DOF/node makes thermal problems fast to solve. However, thermal analysis may be also seen as very challenging; we will now explain why it may be difficult to produce meaningful results.

Thermal analysis with FEA is capable of modeling heat transfer in solid bodies only. In reality, however, heat transfer in solid bodies is accompanied by heat exchange between faces of solid bodies and a surrounding fluid; in radiation problems, heat is radiated out. Modeling convective heat transfer is a major problem because FEA cannot model fluid flow around a solid model and convection must be modeled as a boundary condition. The only difference between natural and forced convection or between air and water surrounding a solid body is a different value of convection coefficient and ambient temperature.

Meaningful results of thermal analysis with FEA require reliable data on the convective boundary conditions and that is often difficult to find. An alternative to defining convective boundary conditions using test data or numbers found in reference materials is solving fluid motion problem. This is called conjugate heat transfer problem and includes fluid flow around the solid body; conjugate heat transfer requires computational fluid dynamic (CFD) program to find convective boundary conditions, and to transfer them to FEA. Many computer-aided engineering (CAE) programs offer both FEA and CFD modules. CFD and FEA may be interfaced with CAD and between themselves (Figure 12.10).

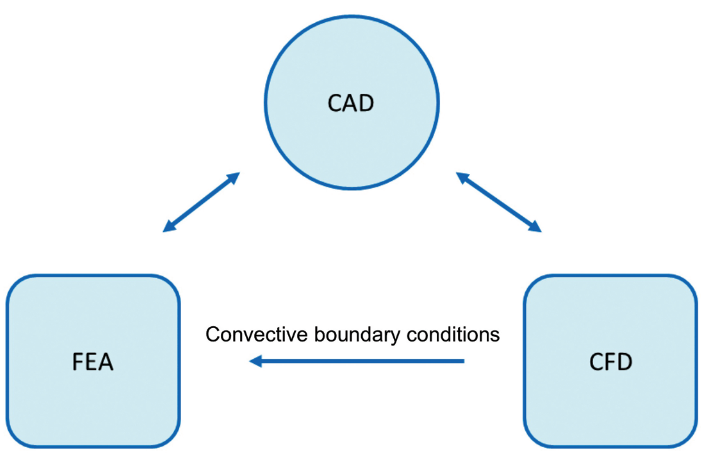


Figure 12.10 If convective boundary conditions are not known prior to thermal analysis with FEA, then a conjugate heat transfer problem must be solved. In conjugate heat transfer problem, heat transfer is modeled in both fluids and solids. Convective boundary conditions found with CFD are transferred to FEA. In modern CAE programs, CFD programs are integrated with CAD just like FEA programs.

12.6 Hand-On Exercises

12.6.1 Bracket (Figure 12.1)

Model name

- 12.01.BRACKET.x_t
- 12.01.BRACKET.sldprt

12.6.1.1 Objective: Demonstrate thermal analysis with prescribed temperatures where conduction is the only mechanism of heat transfer.

12.6.1.2 Comments: Notice that no heat escapes through the walls because no convection coefficients are defined; heat enters and exits the model only through faces where prescribed temperatures are defined.

The following are the required steps:

1. Define any material properties.
2. Define prescribed temperatures to the back face and to the cylindrical face of hole, as shown in Figure 12.1.
3. Mesh and solve.
4. Analyze the plots of temperature and heat flux.

12.6.2 Heat Sink (Figure 12.2)

Model name

- 12.02.HEAT_SINK.sldasm
- 12.02.HEAT_SINK.x_t
- 12.02.HEAT_SINK_sharp_edges.x_t

12.6.2.1 Objective: Demonstrate thermal analysis with heat source and convection.

12.6.2.2 Comments: Start this exercise using geometry with filets. If you use parasolid geometry, use model 12.02.HEAT_SINK.x_t; if you use SOLIDWORKS geometry, switch to configuration *01 filets*. Repeat the exercise using geometry with sharp re-entrant edges using parasolid file 12.02.HEAT_SINK_shapr_edges.x_t or SOLIDWORKS model in configuration *02 sharp edges*.

Observe very high (meaningless) values of heat flux near sharp re-entrant edges; model with sharp re-entrant edges has heat flux singularities.

Repeat the exercise after deleting all convective boundary conditions. Observe that solution fails as no mechanism for heat transfer now exists.

The following are the required steps:

1. Define the material properties of aluminum to the radiator and of ceramic porcelain to the microchip.
2. Define a heat power of 25 W to the volume of the microchip.
3. Define convective boundary conditions on the outside faces of radiator except for the bottom face; the definition of convective boundary condition includes a convection coefficient of 25W/m^2/K and an ambient (bulk) temperature of 300 K.
4. Mesh and solve.
5. Analyze the plots of temperature distribution and heat flux; use section fringe plots for temperature and vector plots for heat flux.

12.6.3 Channel (Figure 12.4)

Model name

- 12.03.CHANNEL.x_t
- 12.03.CHANNEL.sldprt

12.6.3.1 Objective: Demonstrate 2D thermal analysis with heat power and convection.

12.6.3.2 Comments: This exercise may also be completed using a 3D model without 2D representation. Working with 3D model you will use section plots that enable the analysis of heat flow inside the solid body. Section plots are frequently used in thermal analysis.

Vector plots are useful in reviewing and visualizing heat flux results; this is because a heat flux is a vector. Vector plots are also frequently used in thermal analysis.

The following are the required steps:

1. Specify 2D thermal analysis.
2. Define the material properties of aluminum (1060 alloy).
3. Define a heat power of 500 W to the bottom face.
4. Define convective boundary conditions on all other faces: a convection coefficient of 50 W/m^2/K and an ambient temperature of 300 K.
5. Mesh and solve.
6. Analyze the plots of temperature distribution and heat flux; use section plots for temperature and vector plots for heat flux.

12.6.4 Space Heater (Figure 12.6)

Model name

- 12.04.SPACE_HEATER.x_t
- 12.04.SPACE_HEATER.sldasm

12.6.4.1 Objective: Demonstrate heat transfer problem with conduction, radiation, and convection; this is the complete list of heat transfer mechanisms.

12.6.4.2 Comments: Emissivity compares the radiating surface to a black body whose emissivity is 1 and to a perfectly reflective body whose emissivity is 0.

The following are the required steps:

1. Define the material properties of aluminum (1060 alloy) to a reflector and glass to a bulb.
2. Define a heat power of 500 W to the inside faces of the bulb (Figure 12.6).
3. Define convective boundary conditions on the back face of reflector: a convection coefficient of 50 W/m^2/K and an ambient temperature of 293 K.
4. Define radiation boundary conditions on the outside faces of bulb: an emissivity of 0.7 and the temperature of distant enclosure of 293 K.
5. Define radiation boundary condition on the front (concave) face of reflector: an emissivity of 0.1 and the temperature of distant enclosure of 293 K.
6. Mesh and solve.
7. Analyze the plots of temperature distribution and heat flux; use section plots for temperature and vector plots for heat flux.

Chapter 13
Implementation of Finite Element Analysis in the Design Process

In this chapter, we focus on implementation issues: how to realize full benefits of finite element analysis (FEA) used as a design tool during product development process?

13.1 Differences Between CAD and FEA Geometry

The differences between CAD and FEA geometries are important issues encountered by design engineers using FEA as a design tool. CAD geometry is detailed geometry of a part or an assembly; it contains information necessary for manufacturing. Why FEA geometry has to be different and why cannot we just use CAD geometry "as is" for FEA? The reason is that geometry must be meshed prior to analysis. Further, the mesh must be able to model the data of interest properly and must be of reasonable complexity suitable for a solver. Those requirements are summarized as follows:

- *CAD Geometry:* Must contain information necessary for manufacturing.
- *FEA Geometry:* Must be meshable, must allow for creation of a mesh that will correctly model the data of interest, and must allow for creation of a mesh solvable within a reasonable time.

Often, CAD geometry will not satisfy requirements of FEA geometry. CAD geometry serves as a starting point in the process of FE model preparation but is seldom usable for FEA without modifications.

We will describe several actions performed on CAD geometry in order to convert it into FEA-specific geometry.

13.1.1 Defeaturing

CAD geometry contains features necessary to make a part. Many of those features are unimportant for analysis and should be removed prior to meshing (Figure 13.1). Leaving such features results in unnecessarily complicated mesh and long solution times, or worse, it may prevent the mesher from completing its task. Decision on which features can be removed and which should be included in finite-element model requires careful engineering judgment. The small size of the feature compared with the overall size of the model does not always justifies removing. For example, even very small internal fillets should be retained if analysis objective is to find stresses in the area of those fillets; defeaturing the round will create stress singularity (Figure 13.2). However, the round can be removed if analysis objectives are, for example, displacements or stresses in another part of the structure far from edges. Remember, we always have to make sure that defeaturing doesn't change stiffness in any significant way.

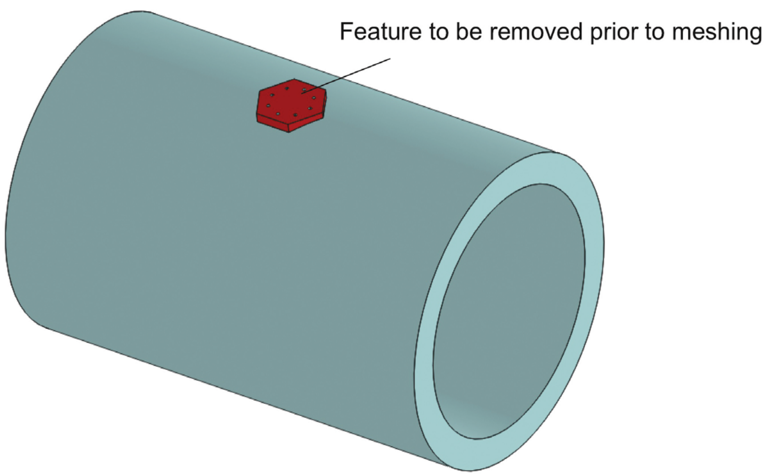


Figure 13.1 An example of a small, not important structural feature (a datum plane) that should be removed prior to meshing.

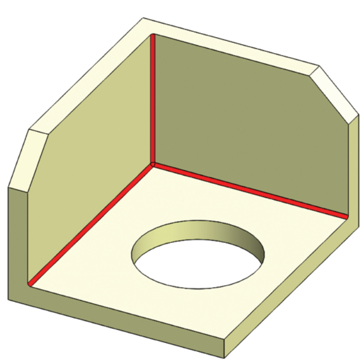


Figure 13.2 Internal rounds (highlighted in red) no matter how small must not be defeatured if the analysis objective is to find stress along the rounded edges.

Note that defeaturing is done on solid CAD geometry. After defeaturing, the geometry becomes the FEA-specific geometry, but it still remains solid geometry eventually leading to 3D solid element FEA model. Thus, defeaturing does not change the type of geometry.

13.1.2 Idealization

Idealization modifies CAD geometry more substantially than defeaturing. Idealization may, for example, involve converting 3D solid CAD geometry into surface geometry suitable for subsequent meshing with shell elements (Figure 13.3) or into wireframe geometry suitable for meshing with beam elements (Figure 13.4). At times, idealization leads to the analysis of reduced dimensionality as in the case of a pressure vessel shown in Figure 13.5. In all those cases, geometry is not just simplified to facilitate meshing. It is converted into abstract geometry suitable exclusively for analysis. Often, solid CAD geometry cannot be converted into idealized FEA geometry and the idealized geometry must be constructed solely for FEA.

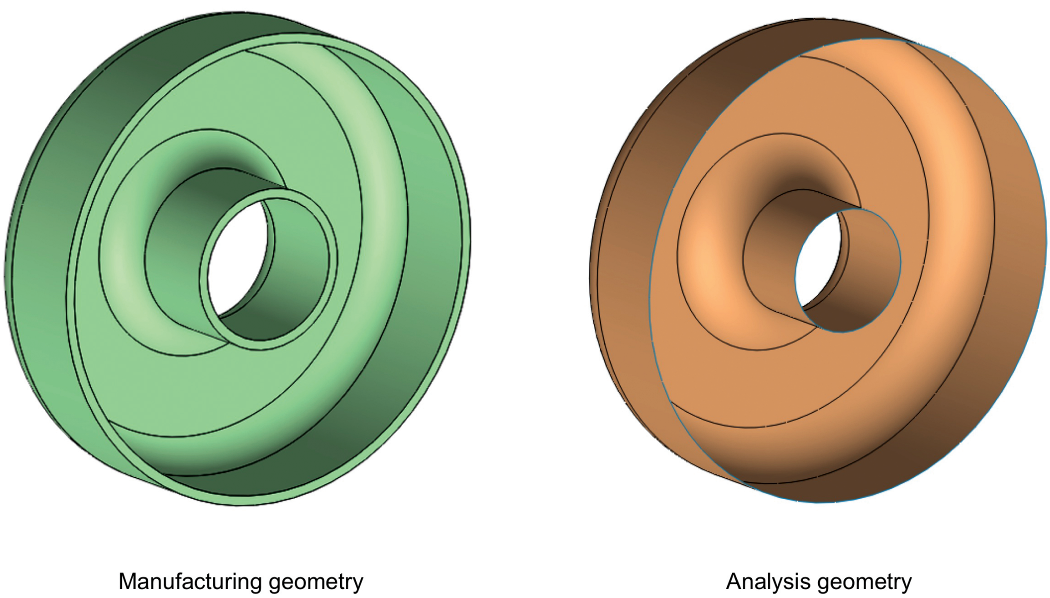


Figure 13.3 A solid model representing a stamped steel pulley (left) is converted into analysis-specific surface geometry for subsequent meshing with shell elements (right). However, remember that shell elements would not model the asymmetric placement of the neutral bending layer, which happens because of high curvature of the pulley geometry. Depending on application, this problem may require the use of several layers of solid elements across thin wall, making it into a problem with many DOF.

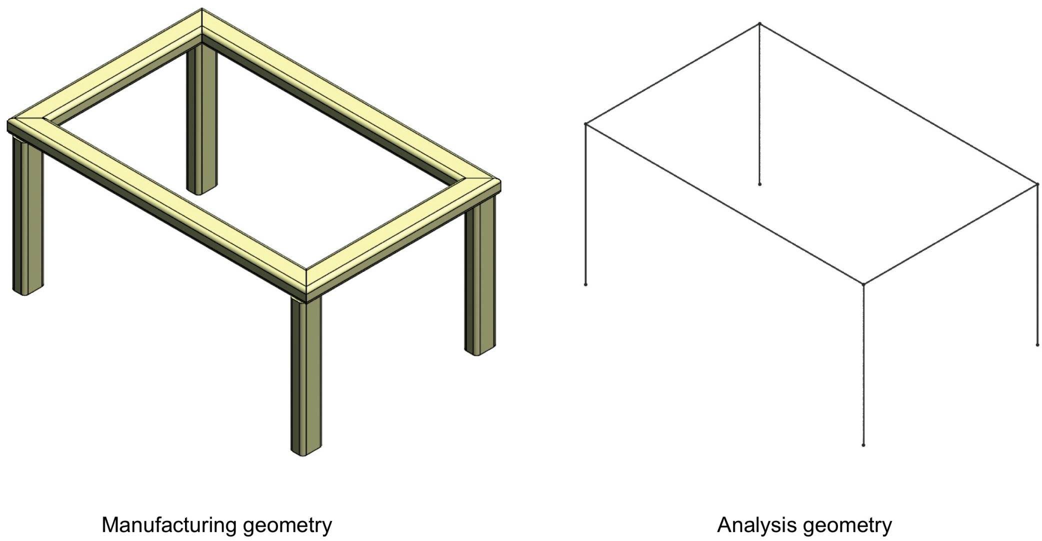


Figure 13.4 A manufacturing-specific solid CAD model of a rollover protective structure (left) is converted into analysis-specific wireframe geometry (right) consisting only of curves (here straight lines) for subsequent meshing with beam elements.

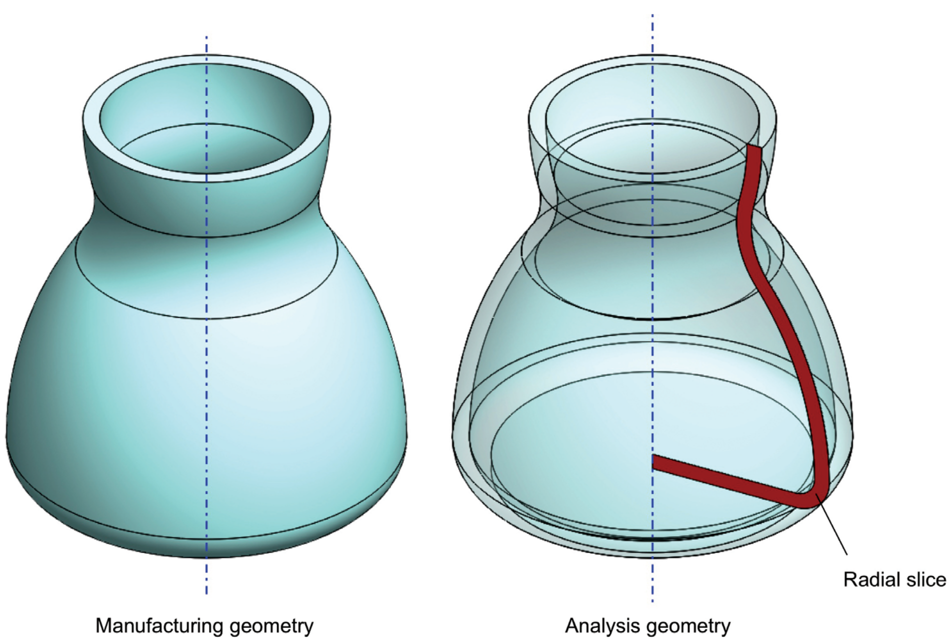


Figure 13.5 A CAD geometry of a vase (left) is converted into a 2D geometry (a flat surface) (right) for subsequent meshing with 2D axisymmetric elements.

13.1.3 Cleanup

An ambivalent term “geometry cleanup” refers to geometry quality issues that must be dealt with to enable meshing. Quite often, geometry that is satisfactory for manufacturing purposes contains features that either will not mesh or will force the mesher to create a large number of elements. Examples are very short edges and/or faces as illustrated in Figures 13.6–13.8. Those miniscule features must be removed, or else, the automesher will try to mesh them. Mesh creation may also fail because of quality issues like multiple entities, floating solids, and other problems that can usually be detected by CAD quality control tools.

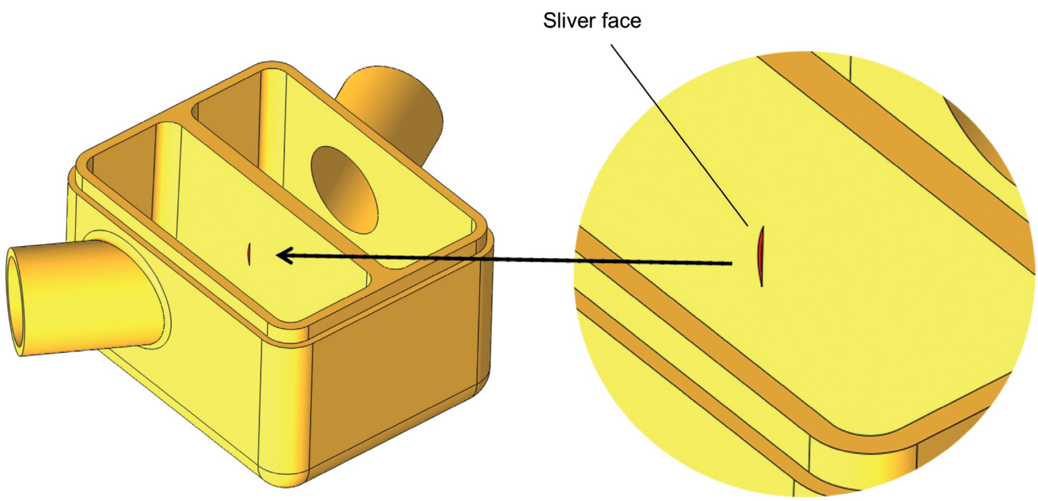


Figure 13.6 A “sliver” face created by geometry modeling error must be removed to enable meshing.

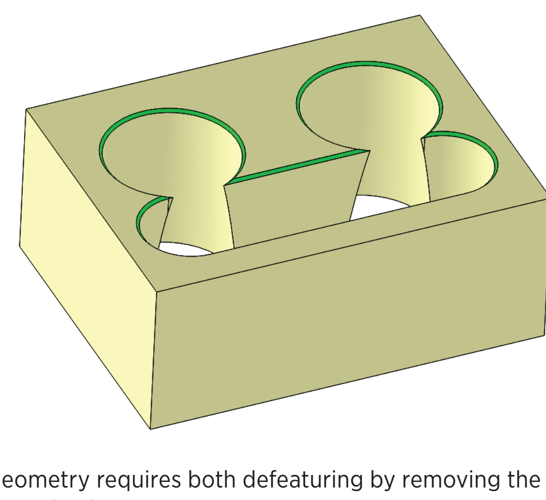


Figure 13.7 This geometry requires both defeaturing by removing the chamfer (green) and cleanup, as shown in Figure 13.8.

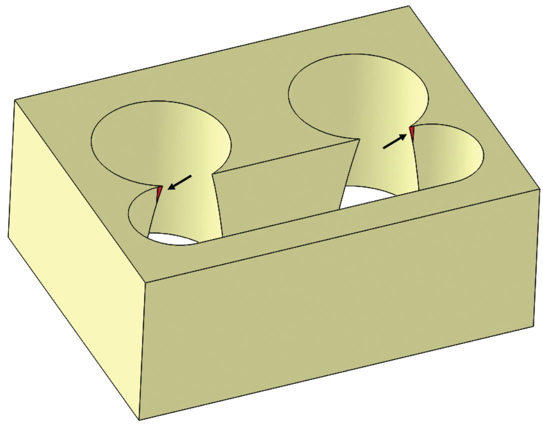


Figure 13.8 Two miniscule faces (red faces indicated by black arrows) should be removed prior to meshing; chamfer seen in Figure 13.7 has been suppressed.

The need for meshing places high-quality requirements on CAD geometry and meshing can actually double as geometry quality check. Therefore, mesh creation may be thought of as another geometry quality test worthwhile performing even if no FEA is intended. Usually geometry cleanup intended to facilitate meshing also results in a better manufacturing geometry. Geometry cleanup may be combined with defeaturing or may precede geometry idealization.

13.2 Common Meshing Problems

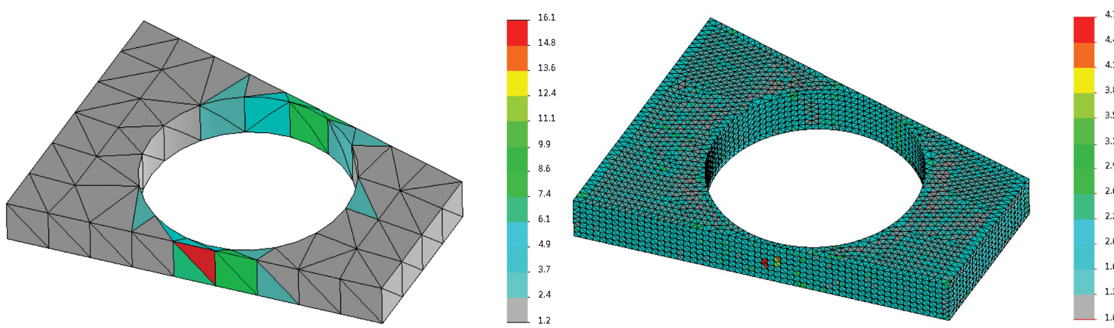
All the tasks described previously: idealization, defeaturing, and cleanup have one common goal: to produce geometry that captures all important features of the analyzed structure and, at the same time, is meshable into a correct, and preferably simple, finite-element mesh.

Meshing nowadays is always done with automeshers. But even though meshing is an automated process, it is not hands-off process and the user's input is still required to control element type and order, element size, and mesh bias.

Creating solid element mesh can be looked at as a process of filling up a volume with primitives of certain shape. Even though solid elements can be tetrahedral, pentahedral and hexahedral, automeshers most often are limited to generating tetrahedral elements. As it is not possible to represent any arbitrary geometry with an assembly of perfectly shaped, regular tetrahedrons, elements must undergo distortion in the process called

mapping. Therefore, the resulting mesh representing the analyzed geometry can be thought of as an assembly of tetrahedral elements with various degrees of distortion described in chapter 5. Complex CAD geometry invariably contains portions that are "difficult" to mesh with properly shaped elements and those portions often end up being represented with highly distorted elements. The same "difficult" portions of model geometry often coincide with areas of high stress where distorted elements will produce incorrect results.

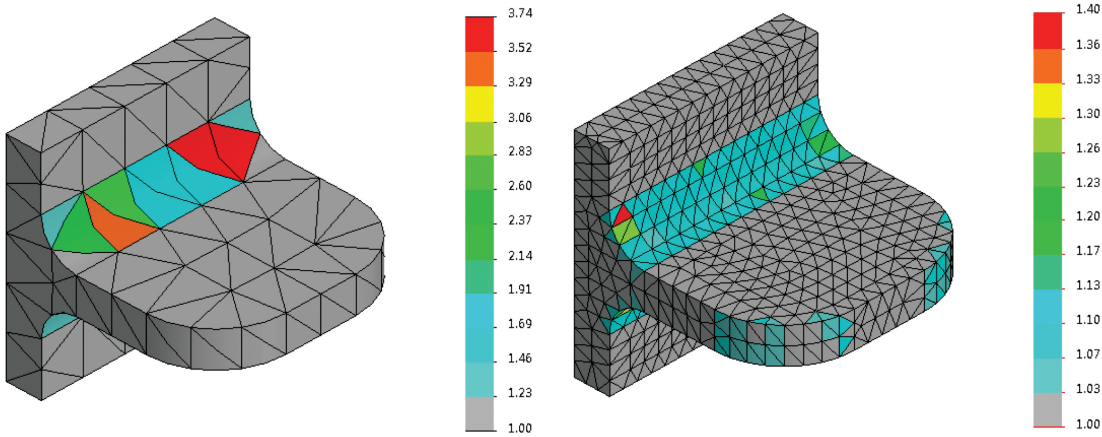
Excessive mesh distortion can be prevented by controlling default element size or by applying local mesh controls. Most FEA programs offer some kind of tools to check mesh quality and those tools should be used. The aspect ratio check is shown in Figure 13.9 and the curvature distortion check (also called Jacobian check) is shown in Figure 13.10. Sometimes, geometry makes it impossible to avoid excessively distorted elements without modifying the geometry itself (Figure 13.11).



Aspect ratio index in model with large elements

Aspect ratio index in model with small elements

Figure 13.9 The aspect ratio is one of measures of element quality; the aspect ratio for a perfect tetrahedral element is 1. Poorly shaped elements are created in thin portions of the model; these elements have higher aspect ratio as indicated by the color scale. The range of aspect ratios in the left model is 1.2–16.1. The mesh with better aspect ratio requires the use of smaller elements; the range of aspect ratios in the model on the right is 1.0–4.7.



Curvature distortion index in model with large elements

Curvature distortion index in model with small elements

Figure 13.10 Using large second-order elements to mesh curvilinear geometry may lead to excessive curvature distortion. These plots show Jacobian check, which is a measure of curvature distortion. The range of curvature distortion index in the model with large elements is 1.0–3.7. The range of curvature distortion index in the model with small elements is 1.0–1.4. Curvature distortion may be measured by turn angle.The turn angle along fillets in the model with large elements is 90° and the turn angle along fillets in model with small elements is 22.5°. This applies only to second or higher order elements whose edges and faces may acquire curvature during mapping of element on a curvilinear geometry.

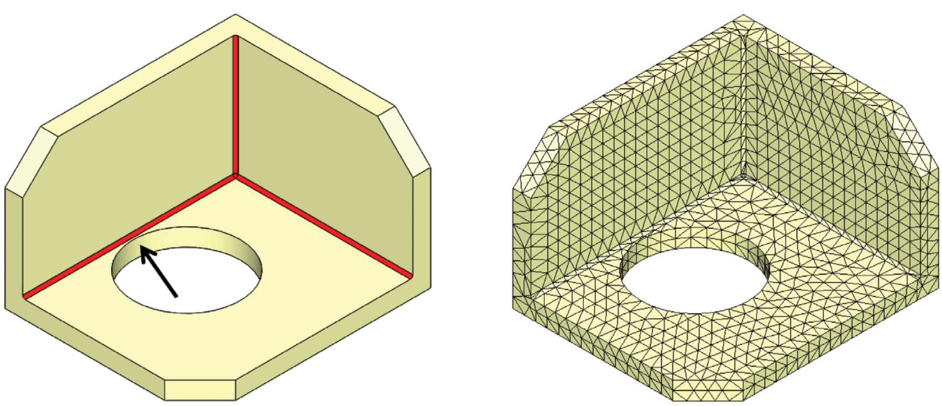


Figure 13.11 Meshing this geometry (left) will inherently result in poorly shaped elements (right). The model portion with tangent edge is shown by the arrow. The mesh has other problems too. Fillets are meshed with elements with excessive turn angle; one element covers the 90° angle, and hence the turn angle is 90°.

Even though we used solid elements to discuss meshing problems, similar meshing considerations apply to shell and beam elements. In the case of shell elements, element distortion applies to the element shape and warpage. Another important consideration in meshing with shell elements is the ratio between element size and the element thickness. In the case of beam elements, this is the ratio of element curvature and element length to the size of cross section.

13.3 Mesh Inadequacy

Even if finite-element mesh is free from excessive element distortion, it does not yet mean this is a good mesh. While elements may be correctly shaped, the number of elements combined with low element order may be unable to represent the expected pattern of data of interest like stresses or displacements. As opposed to element distortion, which can be controlled by mesh quality tools, the mesh adequacy can be only assessed based on understanding of the modeled problem. No software check will warn us against using nice looking but totally wrong mesh shown in Figure 13.12.

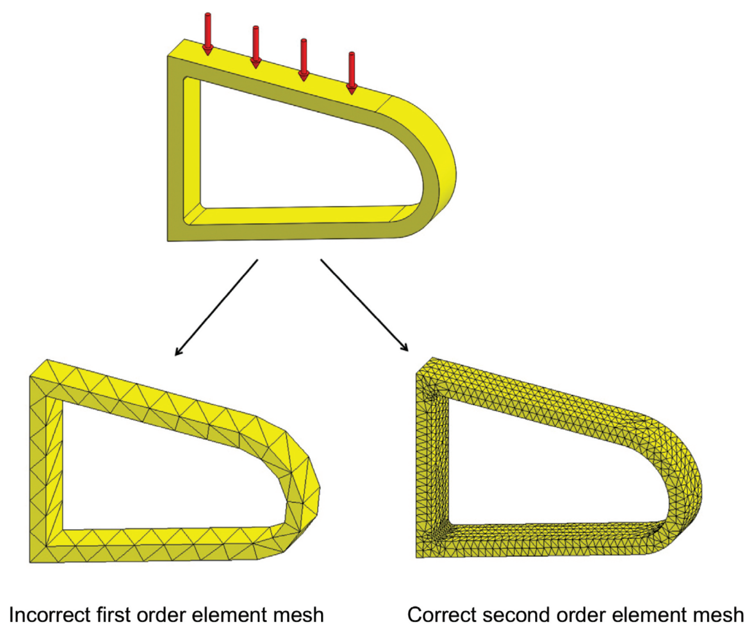


Figure 13.12 The loop model is supported along the vertical face and loaded along the top horizontal face (top). All first-order elements are properly shaped, but the mesh is incapable of modeling bending, which is the primary mechanism of load transfer in this structure. Several layers of first-order elements would be required to model bending stresses properly; second- (or higher) order elements should be used (bottom). Notice stress singularity in the form of sharp re-entrant edges, which is a modeling error and has nothing to do with meshing issues. The correct mesh shown has been created with two layers of second-order elements and stress singularities have been eliminated by modeling rounds and specifying mesh bias on those rounds.

13.4 Integration of CAD and FEA Software

Concurrent use of CAD and FEA software during the product development process means that geometry information is frequently exchanged between those two applications. This invariably brings up the issue of the desired level of integration between CAD and FEA software.

13.4.1 Stand-Alone FEA Software

Commercial FEA programs started as stand-alone applications in the early 1970s, long time before CAD became available. Those early FEA programs included rudimentary by today standards tools for geometry creation, adequate for creating and editing simple models in 2D or 3D. The advent of 3D solid, parametric, feature-based CAD, which offered much more powerful geometry creation tools than those included in early FEA programs, made it possible to use CAD for preparing FEA models. Geometry could be exchanged between CAD and FEA software utilizing neutral file formats like IGES. Later, direct interfaces enabled FEA software programs to read CAD geometry data directly from CAD models.

13.4.2 FEA Programs Integrated With CAD

FEA program intended as a tool for design engineer should be integrated with CAD for efficient exchange of information between design and analysis models and analysis results. In integrated CAD-FEA programs, all geometry creation and editing is done in CAD. If analysis-specific geometry is derived from CAD geometry by defeaturing and/or idealization, CAD offers the ability to switch between those two representations by feature suppression, automatic creation of mid-plane surfaces for shell element meshing, etc. In FEA portion of the integrated CAD-FEA system, users define type of analysis, assign material properties (material properties can be also defined in CAD and then transferred to FEA) loads, and supports, and create the mesh.

The integrated CAD–FEA program must reach a delicate balance between relieving the user from FEA-specific tasks like meshing and still allowing for the necessary level of control of FEA-specific task. Users should be given control over issues like mesh controls and element order. The software should also offer mesh quality checks and a choice of solution options most suitable for the analyzed problem.

13.4.3 Computer-Aided Engineering Programs

FEA is not the only application integrated with CAD. Modern computer-aided engineering (CAE) programs feature a large family of different engineering applications integrated with CAD in such a way that information can be exchanged both ways between application and CAD and between applications themselves. Figure 12–10 may

be expanded to include motion analysis and to show reciprocal exchange of information between all CAE applications, as seen in Figure 13.13. FEA, computational fluid dynamics (CFD), and motion analysis are only a few selected applications; the list keeps expanding with the progress in CAE.

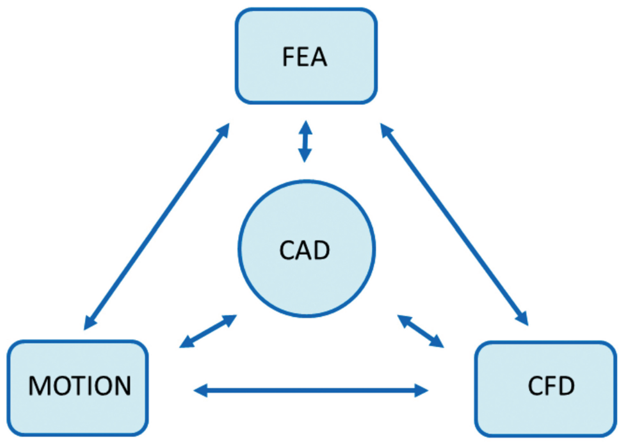


Figure 13.13 CAD integrated with applications such as FEA, CFD, and motion analysis form a CAE system available to design engineers.

13.5 FEA Implementation

13.5.1 Positioning of CAD and FEA Activities

We have already stated that FEA is one of many tools that design engineers use. It should be used concurrently with design process and results produced by FEA should help in making design decisions. This concurrent CAD–FEA process is illustrated in Figure 13.14 (left). Notice that design proceeds in manufacturing-specific geometry and FEA proceeds in FEA-specific geometry. Every time FEA is used, the interface line is crossed twice: for the first time when modifying CAD geometry to make it suitable for analysis with FEA and for the second time when making design decision based on FEA results. The interfacing requires substantial effort and is prone to errors. This significant interfacing effort may be avoided if, recognizing the differences between CAD and FEA geometries, new design is started and iterated in FEA-specific geometry. Only after a sufficient number of iterations, transfer is made to CAD geometry by adding manufacturing-specific features. In this way, the interfacing effort is reduced to just one switch from FEA to CAD geometry, as shown in Figure 13.14 (right).

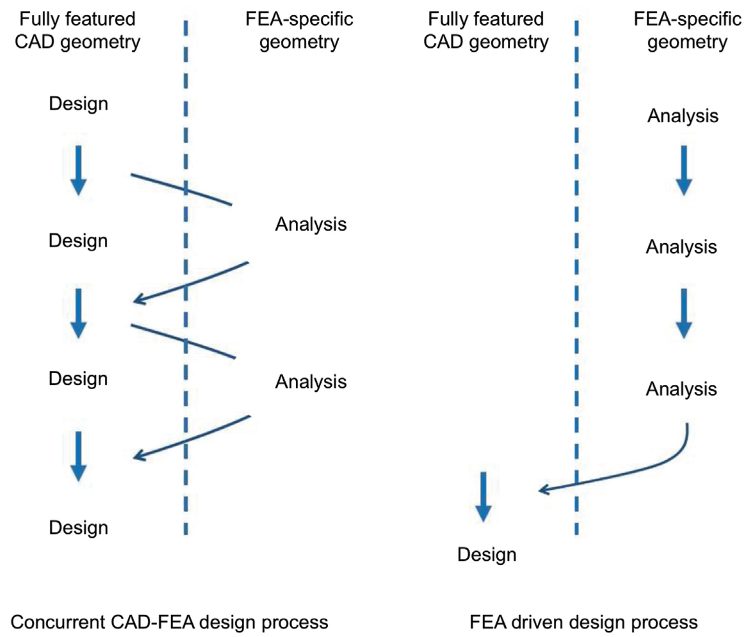


Figure 13.14 The concurrent CAD–FEA design process and the FEA-driven design process.

13.5.2 Personnel Training

Who should be in charge of FEA? How should we divide the analysis responsibilities between design engineers and analysts? Analysis used to be the exclusive domain of analysts, but this has been constantly changing in favor of design engineers. Experience indicates that design engineers, if provided with modern FEA software and with the appropriate training, are best positioned to analyze their design while it progresses through design iterations. The direct involvement of design engineers in analysis of their own designs assures that analysis results are correctly implemented into the design in progress. All this of course means significant expansion of responsibilities for design engineers. While traditionally drafting, designing and analysis were separate tasks executed by different people, the availability and power of both CAD and FEA programs, along with competitive pressures, extended responsibilities of design engineers first to design creation in CAD and then to design analysis with FEA (Figure 13.15). Analysts, of course, continue to play an important role assuming responsibilities for more complex types of analyses.

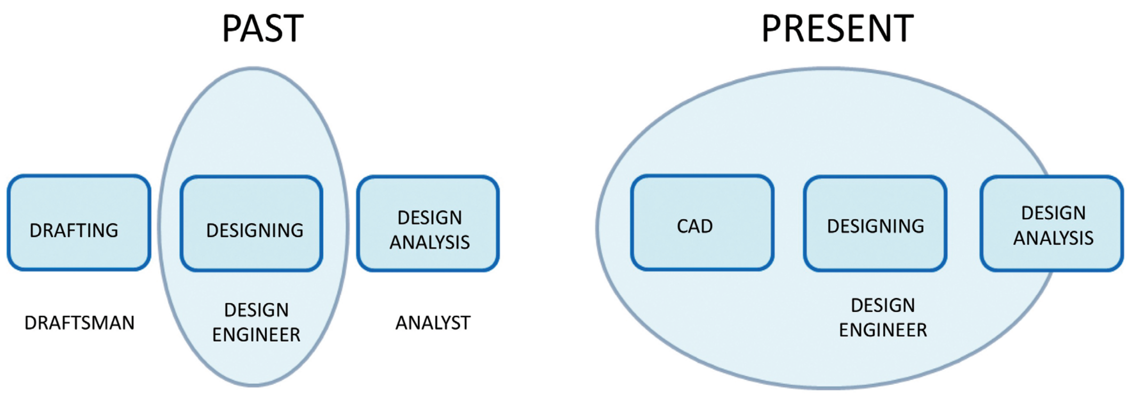


Figure 13.15 Increasing responsibilities of design engineers coming along with progress in CAE.

Even though we focus on FEA, all of the above applies to other CAE applications such as CFD or motion analysis.

The essential differences between FEA run by design engineers and analysts are summarized in Table 13.1. This table lists a typical split between types of analyses performed by design engineers and by analysts. However, growing experience of design engineers and better accessibility of FEA programs make design engineers engage in more advanced type of analyses, which used to be exclusive competency of analysts.

Table 13.1 Major differences between FEA performed by the design engineer and the analyst

	Design engineer	Analyst
Background	Design engineering	Design analysis
Types of analysis	Static Modal Buckling Steady-state thermal	Nonlinear Transient thermal Vibration
Degree of idealization of CAD geometry	Preferably all analysis done on geometry derived from CAD models with few modifications	Extensive idealization and use of shell elements, beam elements, 2D elements
Mode of analysis	Concurrent with the design process	“Offline”

How should we prepare design engineers for the added FEA responsibilities? Most of what design engineers need to know about the FEA is already well within their engineering background especially that most of undergraduate mechanical engineering programs include courses in FEA. Training in FEA does not have to introduce any exotic engineering knowledge or concepts. Topics from well-known subjects like machine design, mechanics of materials, vibrations, and heat transfer need to be assembled together and presented through their applicability to FEA.

The key issue in FEA training is providing users with conceptual understanding of FEA fundamentals: major assumptions, limitations, inherent errors as well as common mistakes, and traps and misconceptions. The training course should be run computers hands-on, so participants can benefit from synergy between acquiring software skills and familiarization with FEA background at the same time. Meaningful examples illustrating both FEA theory and software capabilities can be solved after a short introduction to FEA background and a quick familiarization with software. Geometry for all examples should be prepared in advance so that no time is wasted on CAD-specific tasks. The progressive familiarity with software should be worked into examples of increasing complexity. While extensive theoretical treatment is not required, a simplistic approach should also be avoided. A balance must be reached between FEA theory and software-specific functions because too much focus on "how to run software" may overshadow important issues of FEA fundamentals. It may also give a false impression that familiarity with the program equals expertise in FEA. Experience indicates that users who know fundamentals of FEA can easily figure out how to operate the program, but skills in operating the program do not necessarily lead to full understanding of FEA. After basic training, users should be given opportunity to try out their skills on simple projects under guidance of an experienced FEA user before more advanced training commences.

The highlights of FEA training for design engineer are summarized below:

- Focus on conceptual understanding of FEA.
- Make it hands-on with examples of progressing complexity.
- Have examples prepared in the form of CAD models.
- Combine theory with hands-on examples.
- Use integrated CAD–FEA program.
- Stress out that CAD and FEA even though they work together, are different applications.
- Follow up with a more advanced training.

A basic training should also be provided to all personnel who do not get involved with FEA directly but needs to supervise engineers performing FEA, request analysis, and interpret and implement FEA results. Training may be conducted in face-to-face format or as a webinar; in the author's opinion, none presents significant advantages over the other.

13.5.3 FEA Program Selection

Considering that so many different FEA programs are available on the market and all appear similar, how do we choose the one that is best suited for use as a design tool? Which one will bring us the best return on investment when implemented in the design process? We will point out several issues to consider when selecting FEA software:

- *Ability to Exchange Information Between FEA and CAD:* CAD and FEA need to be used side by side in product design processes: CAD for creating geometry of new design and FEA for analyzing it. We cannot select the FEA software for use by design engineers without considering which CAD system will be used. While it is possible to exchange information between CAD and FEA through neutral formats like IGES, STEP, or Parasolid, this method of communication is time consuming, prone to errors, and offers no associativity between CAD and FEA geometries. A direct interface where FEA software can read CAD file directly is better but still requires maintaining two separate databases: one for the CAD model and the other for the FEA model. To maximize the benefits of FEA used as a design tool, FEA program must be integrated with CAD in such a way that FEA is executed without leaving CAD program. The success of integrated CAD–FEA system depends on both capabilities of CAD and FEA modules. The integrated CAD–FEA duo works well only if solid, parametric, and feature-based solid CAD software is used. Using this type of CAD, geometric features can be temporarily suppressed without permanently deleting them and different design configurations can be easily examined by taking advantage of parametric formulation of the CAD model. CAD should be able to create meshable geometry and assign material properties, while the rest of FEA model definition (analysis type, loads, restraints, and mesh), solution, and result analysis should be handled by the FEA part of the CAD–FEA team. This not only allows for quick CAD–FEA iterations, but it also reduces FEA training time because all geometry-related functions are performed using the already familiar CAD rather than by FEA software.
- *FEA Software Should be Easy to use but User Should Still be Able to Control It:* The FEA should not be hidden from users. Therefore, the FEA program should offer control over meshing, type, and order of elements, idealization scheme, and the desired solution options. While in most cases the default choices offered by advanced FEA programs are acceptable, the user should be able to control FEA-specific tasks if such an intervention becomes necessary or desired.
- *FEA Software Should Have a Good Automesher and Fast Solver:* The quality of finite-element mesh is essential in producing quality FEA results and the fast solver is important to produce those results in a timely manner. A good mesher and a fast solver reduce the efforts spent on geometry preparation because even large CAD models mesh and solve fast providing results quick enough to keep up with the design process.
- *FEA Program Should Handle Common Types of Analyses:* How powerful should the FEA program be in terms of analysis capabilities? It may be tempting to buy FEA program that can handle all possible types of analyses, just in case if one day we are faced with a though analysis problem. However, those top end FEA programs are often difficult to use and/or are not integrated with CAD. For these reasons they perform poorly in when used be design engineers. Rather than selecting the most powerful program we can find, we should look for the one that addresses the majority of our analysis requirements and does it in the best possible way. Our

FEA program of choice should be easy to use and manage, should produce results fast, and be well integrated with CAD but does not have to support exotic types of analysis.

Most analysis types preformed in a concurrent CAD–FEA design process are the following types:

1. linear static analysis
2. nonlinear large displacements analysis
3. nonlinear material analysis (simple types of nonlinear material models)
4. contact stress analysis
5. modal analysis
6. linear buckling analysis
7. steady-state thermal analysis

The selected FEA software should support these types of analyses, while more advanced capabilities are optional. Chances are that occasional projects requiring advanced types of analysis will be either too complex or data will not be available to execute it concurrently with the design process. It is best to hand over those complex projects to analysts.

To facilitate data exchange between the design engineer and the analyst, the selected FEA program for design engineer should ideally be a subset of a larger FEA program. If a more advanced analysis becomes necessary, models prepared by the design engineer can be forwarded to the analyst without the need for geometry reconstruction or translation:

- *FEA Program Should Have Good Presentation Tools:* FEA program should incorporate tools to communicate design intended for the rest of organization in a clear and concise manner. Reports should be created automatically and results should be accessible to anybody with standard office environment software, without having to use the FEA software itself. Even the best results are useless unless, they are properly presented and presentation tools may actually be the most critical part of FEA software.
- *FEA Program Expertise Should be Available to Users:* In our selection process of FEA program for the design engineer, we should also check for the availability of expertise starting from on-line help through hotline support to user groups and consultants independent of the software vendor. The software vendor should not be the sole source of expertise in the chosen FEA software.
- *Cost of FEA Program:* Finally, the FEA program should not be costly either in terms of licensing cost or in terms of cost of implementation or training. However, the cost alone should not be the first consideration. The cost of buying an inappropriate FEA program outweighs all savings realized on the purchase price.

Let us now summarize the requirements placed on FEA as well as on CAD program.

A CAD system should satisfy the following requirements:

- be solid, parametric, and feature-based
- be able to create all geometry in both CAD-specific and FEA-specific configurations
- offer quick alternations between those two geometries while keeping them linked
- be able to send geometry and material properties data to FEA program

FEA system should obey the following requirements:

- be integrated with CAD
- support common CAD exchange formats in case data come from non-native CAD
- have an advanced mesher with provision for the user's control and a fast solver
- support common types of analyses like static, modal, buckling, thermal
- be scalable to top end "analyst style" FEA
- offer tools for communicating with the rest of organization
- have good users' support
- be reasonably priced

13.5.4 Hardware Selection

Ever since Windows® operating system has dominated the engineering hardware market in the 1990s, hardware selection has become by far a secondary issue compared with software selection. Generally, any computer capable of running solid, parametric, and feature-based CAD is also adequate to run FEA programs. Two important factors to consider are the amount of RAM memory (the more the better) and a good quality graphics (graphics card and monitor). In fact, the amount of RAM memory is more important than the processor speed. A slower processor will take longer to solve complex models, but not enough memory will make them impossible to solve.

13.5.5 Building Confidence in the FEA

Having acquired an appropriate FEA program and hardware and having provided the prospective users with training, we now need to demonstrate the usefulness of FEA to the rest of organization. This is best done in steps:

- Analyze typical successful designs now in production to establish the modeling approach and the acceptable safety factors.
- Validate each FEA results with testing and field results.
- Modify modeling approach if necessary.
- Add the case to the database.

- Keep track of the cost associated with the FEA and compare it with traditional design methods.

Building confidence in FEA requires that results be validated using analytical or experimental methods. Some of those experimental methods along with their advantages and disadvantages are summarized in Table 13.2.

Table 13.2 A partial list of methods available for validating results of FEA

Experimental method	Advantages	Disadvantages
Strain gauge test	Real loads, real structure, and reliable stress information	Provides only point information, and loads are often unknown
Photoelasticity	Provides field information	Expensive and time consuming; similitude rules are difficult to satisfy. Mostly of historical importance
Stress coating	Real loads, real structure relatively easy to apply	Good for qualitative information only, quantitative results unreliable
Brittle coating	Easy to use and fast	Limited to qualitative analysis
Results of laboratory / field tests	Reliable results	It may be too late to use in the FEA
Results of the previous similar FEA	Quick to assess relative improvement between two models	Link to reality still unknown if the first FEA was not supported by an experiment

13.5.6 Return-On Investment

The actual return on investment (ROI) depends on several factors that we have already discussed: what CAD and FEA programs are used, how is FEA implemented in the design process, and how well FEA users have been trained. Experience indicates that the cost of FEA program and training can be both recovered in a single implementation of FEA. Based on the author's experience in the field of automotive engineering, the cost of prototyping and testing of a simple part like engine bracket, pulley, or door hardware can easily run somewhere between 10000$ U.S. and 100000$ U.S. taking several weeks or months to complete. This compares with the cost of FEA software in the range of $4000–$8000 plus $2000 to $4000 for the user training. Even more importantly, results can be produced in a matter of hours or days as opposed to weeks or months required for prototyping and testing. The direct cost comparison is even greater in favor of FEA and against traditional prototyping and testing approach if more complex parts are analyzed. For complex parts, the combined cost of prototyping and testing of one design easily runs into hundreds of thousands of dollars and takes several months to complete. This cost comparison still does not account for savings from reduced warranty costs and

improved overall product quality. The ROI will certainly differ in each individual case and it actually might be easier to talk about losses caused by not using FEA.

Of course, we need to stress out that the aforementioned factors apply to correctly executed FEA. Erroneous FEA provides wrong results resulting in incorrect design decisions, which may be very expensive to rectify later in the design process.

13.6 FEA Project

We will now discuss steps in the FEA project from managerial point of view. While previously described steps: creating a mathematical model, creating an FEA model, solution, and analysis of results still apply, they must be appended by justification for analysis before any activity starts and implementation of results once analysis completes.

The following applies to structural analysis projects but may be easily expanded to thermal analysis projects.

13.6.1 Major Steps in FEA Project

Those of steps in an FEA project that require the involvement of management are marked with asterisks.

- *Do I Really Need the FEA?*:* This is the most fundamental question to address before any analysis starts. FEA consumes significant company resources to produce results and each use should be well justified. Providing answers to the following questions may help to decide if FEA should be executed:
  - Can I use previous test results or previous FEA results?
  - Is this a standard design so no analysis is necessary?
  - Are loads, restraints and material properties known well enough to make FEA worthwhile?
  - Would a simplified analytical model do?
  - Does my customer demand the FEA?
  - What shall I do with the results of the FEA?
  - Do I have enough time to implement results of the FEA?
- Should the analysis be done in house or should it be contracted out?*: Conducting analysis in-house and using consultants both have advantages and disadvantages. Consultants may produce results faster, while in-house analysis is conducive to establishing company expertise leading to long-term savings.
- *How Fast do I Need to Produce Results?**
  - Do I have enough time and resources to do complete FEA before design decisions must be made?
  - Is in-house expertise available?

  - Is my FEA program what my customer wants me to use?
- *Establish the Scope of Analysis:* Having established that we do need to conduct FEA, we need to decide on the extent of required analysis. The following list of questions may help in defining the scope of analysis:
  - A standard analysis of a new product from an established product line?
  - The last check of production-ready new design before final testing?
  - Quick check of design in progress to assist designer?
  - An aid to research and development (R&D) project (particular details of a design, gauge, fixture, etc.)?
  - Conceptual analysis to support a design at an early stage of development (R&D project)?
  - A simplified analysis (e.g., only a part of the structure) to help making a design decision?
  - What is an estimated number of model iterations, load cases, etc.?
  - How should I analyze results? What are evaluation criteria and safety factors?
  - How will I know whether the results can be trusted?
- Establish a cost effective modeling approach and define the mathematical model accordingly. Having established the scope of analysis, the finite-element model must now be prepared. The best model will be, of course, the simplest one that still provides the required results with an acceptable accuracy. Therefore, the model should be as simple as possible to minimize project cost and duration, yet it should account for essential characteristics of the analyzed object. We need to decide on the acceptable defeaturing and/or idealization. The goal is to produce a meshable geometry properly representing the analyzed problem. Further, loads and restraints must be formulated in accordance with the type of performed analysis. For example, vibration analysis will require loads and/or supports defined as a function of time. Large deformation analysis will require defining loads as following or nonfollowing, etc. Finally, material properties must also be defined in accordance with the type of analysis and degree of simplifications.
- *Create a Finite-Element Model and Solve It:* The finite-element model is created by discretization of a mathematical model; this is commonly called meshing. Although the term "meshing" implies that only geometry is discretized, discretization also affects loads and restraints. Meshing and solution are largely automated steps but still require the user's input depending on the software used, which may include the following:
  - element type(s) to be used
  - default element size and size tolerance
  - mesh controls (if any)

    - type of mesher to be used
    - type of solver and solver options to be used
- *Evaluate the Results:* FEA results must be critically reviewed prior to using them to make design decisions. This includes verification of assumptions and assessment of results. This is an iterative step that may require several loops to debug the FEA model and to establish confidence in results. Evaluating the results include the following:
    - Study animated displacements to verify restraints.
    - Check for rigid body motions.
    - Check overall stress levels (order of magnitude) using analytical methods to verify the applied loads.
    - Check reactions and compare them to reactions found using free-body diagrams.
    - Conduct the convergence process and review discretization errors.
    - Analyze stress concentrations and mesh ability to model them properly.
    - Review results in difficult-to-model locations like thin walls and high stress gradients.
    - Investigate the impact of element distortions on the data of interest.
- *Analysis of Results*:* The exact execution of this step depends, of course, on the objective of analysis. In all cases, however, results should be presented in a way suitable for using them to make a design decision. The analysis of results includes the following steps:
    - Present displacement results.
    - Present modal frequencies and associated modes of vibration (if applicable).
    - Present stress results and corresponding factors of safety.
    - Consider modifications to the analyzed structure to eliminate excessive stresses, to improve material utilization and manufacturability.
    - Discuss results and repeat iterations until the acceptable solution is found.

Management of the FEA project requires the manager's involvement during project execution. The correctness of FEA results cannot be established only by review of results. Involvement in project progress helps managers to keep in the loop and improve communication with person performing the analysis. Several checkpoints are suggested in Figure 13.16.

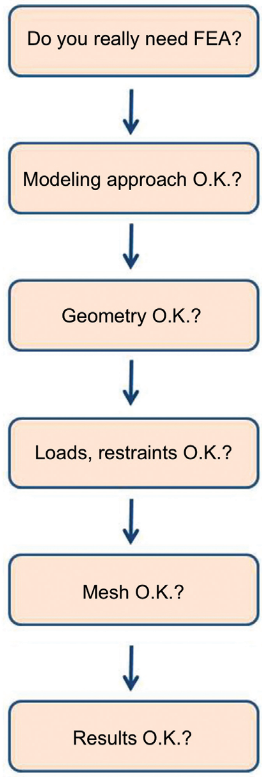


Figure 13.16 Checkpoints in an FEA project; the project is allowed to proceed after each step has been approved by the manager.

13.6.2 FEA Report

Even though each FEA project is unique, the structure of most FEA reports follows similar patterns. Reports should be self-explanatory and contain all information necessary to analyze results and recreate results, should that become necessary. The following are major components of a typical FEA report and their contents:

- *Executive Summary:* Objective of the project, part number, project number, essential assumptions, results and conclusions, software used including software release, information on where project backup is stored.
- *Introduction:* Describe the problem. Why did it require FEA? What kind of FEA? (static, contact stress, vibration analysis, etc.). What were the data of interest?
- *Description of Loads and Restraints:* Describe loads and restraints, include load diagrams (e.g., free-body diagrams), discuss any simplifications and assumptions, etc.
- *Description of Model Geometry:* Describe the model geometry and how it was created: from CAD geometry using an integrated CAD program, from CAD through an interface; describe and justify any geometry cleanup, defeaturing and idealization; and justify the modeling approach (solids, shells, beams, 2D, etc.).

- *Description of Mesh and Method Used to Control Discretization Error:* Describe the type of elements, global element size, any mesh control applied, number of elements, number of degrees of freedom, the type of automesher used, and the type of solver used. Justify why is this mesh adequate to model the data of interest. Describe the method used to control discretization error (mesh refinement, *h* adaptive solution, etc.)
- *Analysis of Results:* Present displacement and stress results including plots and animations. Justify stress measures used to present results (maximum principal stress, von Mises stress, maximum shear stress, etc.). Discuss errors of the results and present applicable safety factors.
- *Conclusions:* Make recommendations to the requester regarding structural integrity, necessary modifications, and further study needed. Recommend testing procedure (e.g., strain gauge test and fatigue life test). Make recommendation on future similar designs.
- *Project Documentation:* Building in-house expertise requires very good documentation of each project beside the project report itself. Therefore, significant time should be allowed to prepare project documentation, which should be self-explanatory and complete with all back-up files for easy recreation of results.
- *Follow-up:* After the tests have been completed, a report should be appended with test results; the correlation between analysis results and test results should be presented and corrective action be taken in case correlation is unsatisfactory. The corrective action may consist of an analysis of a revised model or a physical test.

13.6.3 Importance of Documentation and Backups

All FEA-related documents and document flow should be subjected to applicable document quality standards.

FEA reports and backups can be used for the following:

- audit of the work performed
- restart of the work
- basis for execution of modified analyses
- basis for personnel training
- establishing in-house expertise in FEA
- legal documents when liability is involved

A comprehensive project documentation, complete with backups and all additional documentation, is the major building block in the process of accumulating and retaining in-house FEA expertise. The project documentation should be sufficient to recreate the results or run a modified analysis without any need for verbal communications. Without proper project documentation, expertise gained in the project may be lost.

13.6.4 Contracting Out FEA Services

When FEA services are contracted out, the hired consultant does not necessarily become the analyst. As many as three parties may be involved when an FEA project in contracted out:

- in-house analyst who is directly responsible for the formulation of problem, data preparation, and final result analysis
- in-house supervisor responsible for the project
- consultant(s) who participates in selecting modeling approach, runs a particular analysis, and presents results to the analyst

Project definition has to be specific, and the client and consultant should know exactly what and why needs to be done. The in-house analyst should approve each step: definition of loads, restraints, mesh creation, result analysis, and conclusions. It is advisable to get a "warranty" in case results do not correlate with experimental data. Each FEA project, even if contracted out, should contribute to in-house experience in FEA.

Major steps in contracted out FEA projects are shown in Figure 13.17. Notice that a contracted out project still requires significant involvement of requestor and client–consultant communication must be open all the time.

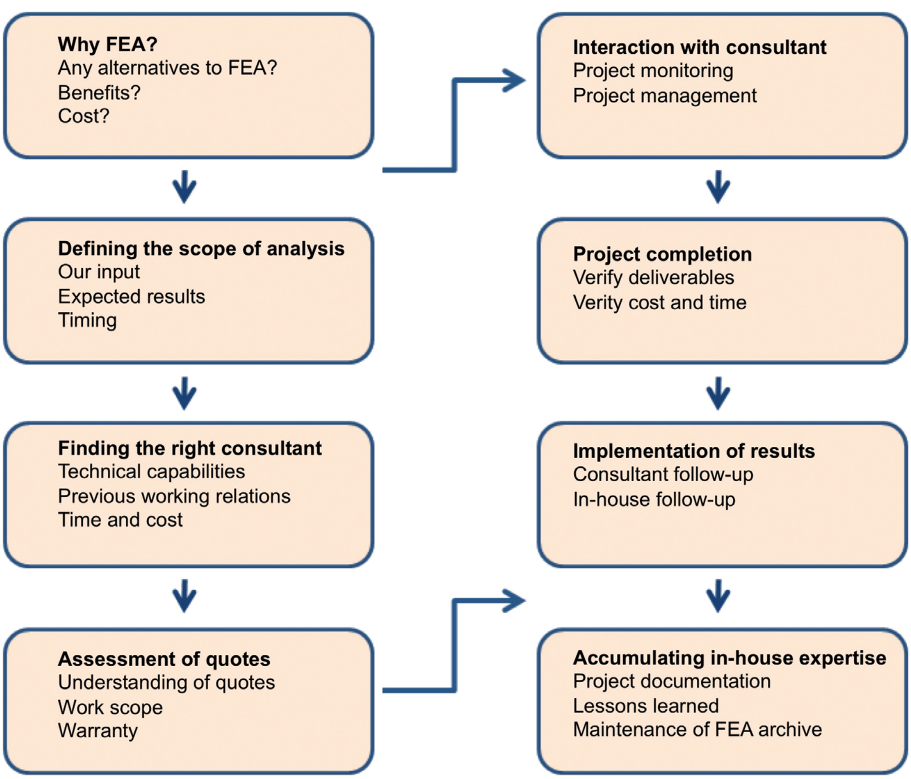


Figure 13.17 The major steps in a contracted-out FEA project.

13.6.5 Common Errors in the FEA Management

FEA projects are often challenging to manage and management errors are easily made. The following is a sample list of common errors with short comments. This is not an "all inclusive" list and is intended only to highlight some frequently encountered problems.

- *Please do FEA:* Project objectives are not clearly defined, neither the manager nor the person performing FEA knows exactly what project objectives and expected benefits are.
- *Too High Expectations Placed on FEA:* FEA results are viewed as an assurance of a sound design.
- *No Time Limit:* The project is allowed to take as much time as the person performing FEA wishes to take.
- *We do FEA on Everything:* FEA is done on models that could be analyzed with less expensive methods.
- *Proficiency in Software Confused With Expertise in FEA:* Skills in using particular FEA program are taken as a proof of expertise in FEA.
- *Confusing CAD and FEA:* The same measures are applied to assess the quality of CAD and FEA projects and personnel.
- *It Was Around When I Was at School so it Must be Good:* Obsolete programs are used.
- *Not Enough Time Allowed for Training in FEA:* Training in FEA is reduced to training in using FEA program.
- *Not Enough Time Allowed for Documentation:* Because of time pressure, projects are not documented properly. Without comprehensive report, good backup, and full documentation, corporate expertise gained in the project is lost.

Chapter 14
Misconceptions and Frequently Asked Questions

Here is a sample of statements and questions circulating in the FEA community, many of them reflect common misconceptions. All quotes are not edited, and in many cases, statements are formulated imprecisely, just the way they were posed. All are listed in a random order and the same topics occur in more than one question. We will start with a quiz and then we will answer some frequently asked questions.

14.1 FEA Quiz

1. The finer is the mesh, the better are your results: true/false?

   Usually true, if "better results" means more "accurate results" in terms of lower discretization error. Finer meshes are more expensive so a compromise should be found to combine acceptable accuracy with acceptable cost of analysis. Accuracy should be established based on the convergence process, not based on using a single mesh.

2. Geometry should be represented as accurately as possible: true/false?

   False. Only significant details should be included in the FEA model. Small and structurally (or thermally) insignificant details like chamfers, drafts, or company logos unnecessarily complicate mesh. Good understanding of the analyzed problem is required to distinguish between significant and insignificant details.

3. Solid elements give the best results because they accurately model the geometry: true/false?

False. Solid elements make impressive models. However, using solid elements does not guarantee the best or even good results. Also, not all geometries are suitable for meshing with solid elements. An example would be a sheet metal structure that is next to impossible to mesh with solid elements. When making meshing decisions, we should always consider alternatives to solids: shell elements, beam elements, or 2D representations (plane stress, plane strain, or axisymmetric elements).

4. Better (more expensive) FEA software gives better results: true/false?

   Of course, false. Good (i.e., correct results) depends on the user's skills. A "top of the line" software may produce terribly wrong results and vice versa.

5. Automeshing is better than manual meshing: true/false?

   This is true and false at the same time. Automeshing is for sure better because it takes little time compared with mapped or truly manual meshing. Automeshing is the only method available in modern FEA programs. So, is automeshing good? A mesh produced by an automesher is not automatically correct. The automesher does not know anything about your analysis objectives. All it does it fills up model geometry with elements. It is our responsibility to assure that mesh is refined where stress concentrations are expected, that we have enough element across members in bending, and so on. For more difficult geometry, automeshers tend to produce distorted elements and place elements with no regard to the laws of mechanics (i.e. one first order tetrahedral element across the wall in bending). Automeshers are much "safer" when used with second- (or higher) order elements; never use first-order elements unless you have a very good reason to use them. Do not rely on automesher with default settings; almost always user has to control element size and mesh bias.

6. High accuracy of FEA results from high processing accuracy of digital computer: true/false?

   False. First of all, who said that FEA is highly accurate? It may be, if it is used properly and if that high accuracy is desired. The accuracy of FEA cannot be confused with the accuracy of solvers. Most FEA programs use double-precision arithmetic for lower numerical error and, in most cases, so called "computer accuracy," or more precisely, the round-off error is small compared with other errors like modeling error and discretization errors.

7. If your FEA software reports no error, the solution will be correct: true/false?

   False. No error means only that the model is correct form the "solver point of view." The solver will solve the most incorrect model as long as it does not run into numerical problems.

8. You do not really need any error estimation, and the FEA is always accurate enough: true/false?

   False. One single run provides results with unknown error. The error may be very low, but we still do not know what that error is. Unless we have some previous

experience with similar problems, we need to run convergence analysis to estimate discretization error.

9. You should always make very fine mesh, so you do not have to worry about error: true/false?

   False. While fine mesh produces low discretization error, we still need more than one mesh to know what that discretization error is. Also, a fine mesh does nothing to modeling errors.

10. Higher order elements will give you more accurate results: true/false?

    True if “more accurate” means “lower discretization error.” Using higher order elements in place of lower order element is practically equivalent to mesh refinement. Comparing two otherwise identical models: first one with first-order elements and the second other one with second-order elements, the latter will provide more accurate results in terms of discretization error. Also, it is much easier to construct a correct mesh using second-order elements because second-order elements map better to curvilinear geometry and model linear stress field; first-order elements model constant stress field.

11. First-order tetrahedral solid elements are too stiff and should be avoided: true/false?

    Generally true. Using first-order tetrahedral element, we will need substantially more elements for proper modeling of stress pattern and for proper representation of curvilinear geometry. So again, do not use first-order elements unless you have a very good reason to use them.

12. Make a coarse mesh first to find stress concentration and then refine it as needed: true/false?

    True in most cases. However, the first “rough” mesh still needs to be fine enough to detect those stress concentrations. If the element size is large in comparison with the size of a “hot spot,” then stress concentration will not be modeled and we will never know that we should have refined the mesh in that particular location.

13. If FEA results correlate well with the experiment (e.g., strain gauge readings), then all FEA results are OK: true/false?

    False. Strain gauge may be placed in a spot that is modeled correctly in the finite-element model. However, correlation in one or more locations does not guarantee that everything is correct with the model. The opposite statement is, however, true: if FEA does not correlate with the experiment, then something is wrong either with the model or with the experiment (or our interpretation of results) or with both.

14. All major FEA codes have been extensively tested, so you do not need to do any benchmarking yourself: true/false?

    Generally true, if you use reputable software and your analysis involves typical cases. Still, it is recommended to run benchmark tests to get the feel of your program, particularly in more demanding applications.

15. Highly distorted elements are OK as long as they are far away from stress concentrations: true/false?

    False. Distorted elements tend to be too stiff and they affect global model stiffness. In other words, they "pollute" the model. The "pollution" may propagate to the point of interest rendering erroneous results.

16. If you study displacements, not stresses, then you can use a coarse mesh: true/false?

    True. Nodal displacements converge faster than stresses. Still, using a certain element size should be justified by the results of convergence analysis.

17. Modal analysis can use coarser mesh than stress analysis: true/false?

    True. Modal analysis find modal frequencies and shapes, which are global measures as opposed to local measures like stress concentrations. Therefore, global stiffness needs to be represented; the details can usually be ignored. However, if modal analysis provides prerequisite results to vibration analysis that finds stresses, then the mesh used in modal analysis should satisfy all meshing requirements applicable to stress analysis.

18. FEA offers a deceiving level of detail: true/false?

    True. Results come in ten digit numbers and every conceivable plot may be obtained along with impressive animations and graphs. It is easy to forget that results often rest on crude assumptions made about material properties, loads, supports, and other modeling simplifications.

19. Model geometry is the most readily controlled of all data, loads less so, restraints (supports) are the most difficult to control: true/false?

    True. Geometry is the most intuitive input and can be inspected by eye or using CAD tools. Loads are relatively easy to relate to as they are vectors and magnitudes are expressed in numbers. Restraints are most difficult to define and, perhaps because of that, proper definition of restraints is often neglected. Most people simply model them either as perfectly rigid or as hinged supports, while the truth is typically somewhere between.

20. Test data always have error and may be inconsistent with FEA assumptions: true/false?

    True. Loads and supports may be different between test and FEA, severe measurement errors may be present, etc.

21. Incompetent analysis gives at best unreliable results and at worst it is positively misleading; bad FEA gives you deceiving trust in the design: true/false?

    True. In the hands of an enthusiastic but unskilled user, the FEA is at best an expensive toy. Depending on the importance of the analysis, it may also be an outright dangerous tool.

14.2 Frequently Asked Questions

1. What is the objective of discretization of continuum?

   A continuous body has an infinite number of degrees of freedom (DOF). Discretization replaces it with a system with a finite number of DOF possible to solve with numerical methods, but discretization introduces a discretization error.

2. What are the major assumptions made in the design of an FE?

   The fundamental assumption in the formulation of an FE is that everything there to know about the element behavior is determined by nodal displacements (or temperatures). As soon as nodal displacements are found, we can find displacements at any point inside the element or along its edge or face using element displacement interpolation functions.

3. What are the basic steps in the FEA?

   a. *Step 1: Construction of a Mathematical Model*
      1. Definition of type of analysis
      2. Definition of geometry (solution domain)
      3. Definition and assignment of material properties
      4. Definition of restraints (essential boundary conditions)
      5. Definition of loads (natural boundary conditions)

   b. *Step 2: Creation of Finite-Element Model:* Discretizing (meshing) geometry into FEs and discretizing boundary conditions

   c. *Step 3: Solution*

   d. *Step 4: Analysis of Results*

4. What are the primary unknowns in FEA?

   The primary unknowns are nodal displacements in structural analysis and nodal temperatures in thermal analysis. Depending on the type of element, nodal displacements are described by all six displacement components: three translations and three rotations or by their subset. Nodal temperatures are scalars requiring only one piece of information. In structural analysis, secondary unknowns such as strains and stresses can be calculated based on displacement solution. In thermal analysis, secondary unknowns such as temperature gradient and heat flux are calculated based on temperature solution.

5. What is the nodal DOF?

   Nodal DOF is an unknown assigned to a node. The physical interpretations are displacements in structural problems and temperatures in thermal problems. In structural analysis, nodes have up to 6 DOF: three translations and three rotations. The actual number depends on the type of element. Nodes of 3D shell elements have the full set of 6 DOF, nodes of 3D solid elements usually have 3 DOF (translations),

and nodes of 2D elements have 2 DOF (translations). In thermal analysis, each node has just 1 DOF (translations): temperature. By defining supports at certain nodes, some nodes are assigned a prescribed value. For example, a rigid support is modeled by prescribing zero displacements to all displacement components of a given node: $x = 0$, $y = 0$, $z = 0$, $rotx = 0$, $roty = 0$, and $rotz = 0$.

6. What is the relation between the number of nodes and the number of DOF in the finite element model?

   Assuming that all elements are of the same type, the number of DOF may be calculated by multiplying the number of nodes by the number of DOF per node minus the number of DOF removed by restraints.

7. What characterizes the state of the minimum total potential energy?

   The total potential energy of the finite element model may be defined as a function of nodal displacements. The state of the minimum total potential energy of an elastic body under load is also the state of equilibrium of the same body. Thus, in the finite element model, equilibrium can be found by looking for this set of nodal displacements that minimizes the total potential energy of the model.

8. How are fundamental equations of FEA formulated?

   Fundamental FEA equations are formulated by minimizing the total potential energy of the model; another method may also be used to formulate these equations. The equations take the form of linear algebraic equations. In a static problem, the equations may be written using matrix notation as

   $$[K] * [d] = [F]$$

   where $[K]$ is a known stiffness matrix, $[d]$ is an unknown vector of nodal displacements, and $[F]$ is a known vector of nodal loads.

9. What is a displacement interpolation function?

   A displacement interpolation function is a formula used to calculate displacement in the given location of element using information on nodal displacements found by solving FEA equations. Displacement interpolation functions are defined on element edges, faces, and the element interior. Displacement interpolation functions must satisfy several requirements:

   a. Internal compatibility; displacements must be continuous over the entire element.

   b. Inter-element compatibility; displacement along the common edge between two elements must be described by the same shape function, so no “cracks” or “overlaps” can form in-between elements.

   c. Rigid body motion; if the element is displaced as a rigid body, the element must show zero strain; this is called patch test.

   d. Constant strain; displacement interpolation functions must be able to model constant strain cases.

If conditions a and b are satisfied, the element is “conforming.” If conditions c and d are also satisfied, the element is called “complete.” If elements are both conforming and complete, the solution of a finite element model will converge to the exact solution (of the continuous mathematical model) during the convergence process. Notice that displacement interpolation functions impose artificial restraints on the structure, so the finite element model is always stiffer than the mathematical. The added stiffness decreases with mesh refinement and/or increases in element order.

10. What is the difference between the first-order and second-order elements? What is a hybrid element?

    The order of an element is determined by the order of its displacement interpolation function. Displacement interpolation functions are polynomials because this makes numerical operations easy to program. The first-order element has first-order (linear) displacement interpolation functions, the second-order element has second-order (quadratic) displacement interpolation functions, etc. The first-order element has straight edges and nodes placed only in corners. The second-order element has midside nodes (along element’s edge) in addition to corner nodes; consequently, it can have curved edges and faces.

    The shape of all edges and faces of an element are defined by functions of the same order as the element displacement interpolation functions, then the element is called isoparametric. The higher the element order is, the more computationally intensive it becomes. At the same time, higher order elements have important advantages. Fewer elements are needed to model given displacement and stress field; curvilinear edges are easier to work with when modeling complex geometry and results produced by higher order elements converge faster with mesh refinement. This brings us to hybrid elements that have been designed to combine computational simplicity of lower order elements with the advantages of higher order elements. Hybrid elements look like first-order elements “from outside” but are capable of modeling quadratic displacement fields; this is called a subparametric element. The exact definition varies from one program to another as element designers use different numerical methods to improve element performance.

11. How do elements “communicate” with each other?

    The only way elements interact with each other is through common nodes. Why then do elements stay together when loaded, without forming “cracks” along edges or overlapping each other? This is taken care of by element compatibility. If the same displacement interpolation function is used by two elements along the shared edge or face, then displacements along both edges or faces will be identical and displacements across the border between elements with be continuous.

12. How is mass assigned to the FE model?

    Mass is assigned to nodes of the finite element mesh. Therefore, mass distribution and inertial properties of the finite element model will not be the same as in a continuous model before meshing.

13. What are different types of finite elements?

    Finite Elements can be classified in many different ways:

    a. by application—structural, thermal
    b. by dimensionality—3D and 2D (plane stress, plane strain, and axisymmetric)
    c. by type—solid, shell, beam, and plate (2D)
    d. by shape—triangular, quadrilateral, tetrahedral, hexahedral, and prism
    e. by order—first order, second order, and higher order
    f. by FEA method—*h*-elements and *p*-elements
    g. by integration scheme—isoparametric, subparametric, and superparametric
    h. By type of analysis they support—linear, nonlinear material, large displacements, large strain, etc.
    i. By assumptions made in element design—thin shell, thick shell, axisymmetric, etc.
    j. Special elements—mass, spring, gap, damper, etc.

14. What are compatible elements and noncompatible elements?

    Compatible elements produce continuous displacement (or temperature) field when assembled in a mesh because the same displacement interpolation functions are used by both elements along the shared edge or face. Compatible elements will not form a "crack" or an "overlap." Non-compatible elements can be forced into compatibility by defining links. This is how non-compatible elements may be used in the same mesh. Links, however, should not be used in areas where accurate results ae required. Stress averaging should not be done between non-compatible elements.

15. What is a distorted element?

    The element has been designed to work within a certain range of shape distortion. The "natural" shape for triangular element is an equilateral triangle and for quadrilateral element, it is square, etc. The actual shape that element assumes after mapping onto model geometry differs from the natural shape. Common types of distortion are aspect ratio and curvature distortion. A finite element mesh should be run through an element quality check and excessive distortions should be eliminated.

16. What is the difference between *h*-elements and *p*-elements?

    h-elements are those elements whose order does not change order during analysis. *h*-elements have fixed order displacement interpolation functions. Convergence analysis in the model using *h*-elements is done by mesh refinement. The name "*h*-element" comes from the element characteristic size denoted by *h*, which is reduced during the process of mesh refinements.

p-elements do not have fixed-order displacement interpolation functions. The solution of a *p*-element model is done in several iterative loops, while the order of displacement interpolation functions is upgraded until the difference in results between consecutive iterations becomes less than the requested accuracy. The name "*p*-element" comes from the *p*-order of polynomial displacement interpolation functions, which are upgraded during the iterative solution. In most programs using *p*-elements, displacement interpolation functions are upgraded only where required, i.e., where differences in certain measures between two consecutive runs are larger than the user's specified accuracy and *p*-elements have the ability to use displacement interpolation functions of different orders along different edges and faces. This requires introduction of blending functions, which are used to determine the field of displacements inside the element based on displacements along all edges and faces.

17. What are the different types of convergence process?

    An *h*-convergence is done by mesh refinement through reducing element size. That reduction can be done globally by refining mesh everywhere in the model or locally by refining the mesh only where the stress concentration or heat flux is expected. *h*-convergence may be performed by the user who runs solution, refines the mesh, compares results between the consecutive runs, etc. Mesh refinement can be done globally or locally. Some programs offer automated *h*-convergence capabilities.

    A *p*-convergence does not affect element size; the mesh stays the same throughout the entire convergence process, but the element order is upgraded. A *p*-convergence is done automatically in an iterative solution until the user-specified convergence criterion is satisfied; the only input required from the user is a convergence criterion (or criteria) and the desired accuracy. A *p*-convergence is an adaptive process and requires elements capable of upgrading the order of selected edges and faces.

    Sometimes, the desired accuracy cannot be achieved even with the highest available element *p*-order. In this case, the user has to refine the mesh manually in a fashion similar to the traditional *h*-convergence and then run the *p*-convergence process again. This is called *p*–*h* or *h*–*p* convergence.

18. What is the objective of convergence analysis?

    FEA gives only an approximate solution. Convergence analysis estimates the discretization error. The discretization error in FEA solution does not have to be minimized; it should be low enough to satisfy the user's requirements.

    Common Convergence Criteria:

    a. strain energy
    b. RMS strain energy (local or global)
    c. maximum stress
    d. displacement (local or global)

19. What is the difference between verification and validation of FEA results?

Verification checks if the mathematical model, as submitted to be solved with FEA, has been correctly discretized and solved; verification deals with the numerical solution of the mathematical model. Validation determines if results correctly represent the reality from the perspective of the intended use of the model. It checks if results correctly describe the real-life behavior of the analyzed object.

20. What types of errors affect the accuracy of FEA results?

    a. *Modeling Error:* Modeling error originates from the fact that FEA does not work on the real structure but on its idealized mathematical model. These idealizations introduce modeling errors, which can be reduced (but never eliminated) by good modeling practices. The modeling error cannot be estimated by any FEA technique, because FEA provides an approximate solution of whatever mathematical model was chosen. Modeling error happens before FEA enters the stage. Study of modeling errors answers the question, "How well does the mathematical model represent the reality?"

    b. *Discretization Error:* Discretization error results from using a finite number of DOF to approximate a solution characterized by infinite number of DOF. Discretization is accomplished by meshing. Discretization imposes certain assumptions on the data of interest (e.g., displacement field inside element must be of second order). Discretization errors are assessed in the convergence process. Discretization error can be defined as convergence error or solution error. Convergence error is the difference in results between consecutive steps of the convergence process. This error can be calculated explicitly. Solution error is the difference between FEA results and results that would have been provided by a continuous mathematical model. Solution error can be only estimated but not calculated exactly. To estimate the solution error, one has to assess the rate of convergence and predict changes in results within the several next iterations as if they were performed. Analyzing of discretization error answers the question, "How well has the mathematical model been solved?"

    c. *Numerical Error:* Numerical error of FEA results is the round-off error accumulated by the solver; usually it is low. Some FEA programs provide various measures to control this error.

    d. *Error of Interpretation of Results:* A typical example would be using von Mises stress to find the factor of safety of brittle material. The error of interpretation of results may be completely eliminated by proper user's training.

21. Can you estimate discretization error without convergence analysis?

    Strictly speaking no; a single solution provides results with an unknown discretization error. However, there are cases where skipping convergence analysis may be OK. If you have successfully analyzed similar problems in the past (including error analysis and, ideally, experimental validation), you can analyze the percentage of change, not the actual magnitudes of results. An error analysis

may be skipped as long as your model is "reasonably" close to the former one. You can also use global or local error estimators provided in your software. These error estimators are related to the difference between averaged and nonaveraged stresses.

22. What is stress averaging?

    Stress averaging is a technique used in presenting stress results. Implementation may vary in different FEA programs; generally, the stress averaging procedure can be explained as follows. Stress results are calculated for each element in certain locations called Gauss points. If more than one Gauss point is present in the element (as is the case with second- and higher order elements), one averaged stress value is reported for the element. Even though these stresses are averaged between Gauss points, they are called nonaveraged stresses or element stresses, because averaging is done internally within the same element. Alternatively, stresses in Gauss points may be extrapolated to element nodes without prior averaging between themselves. Most often, one node is shared by several elements, and each element reports different stress at the shared node. Stress is then averaged on node producing averaged (or nodal) stress results.

    To illustrate the difference between the averaged and nonaveraged stress results, we examine results in Figures 14.1 and 14.2. For clarity of these illustrations, a purposely coarse mesh is used; mesh this coarse could not be used for analysis. The mesh consists of second-order shell elements. Both averaged and nonaveraged stresses should be examined during the analysis of stress results. A large difference between averaged and nonaveraged stress in certain portion of the model indicates that mesh needs to be refined there.

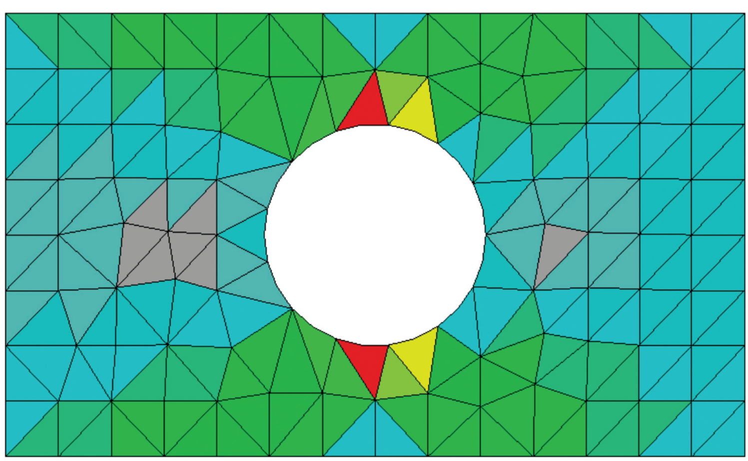


Figure 14.1 Non-averaged von Mises stress results also called element stresses. Stress distribution is that each element is constant because results are averaged internally in elements.

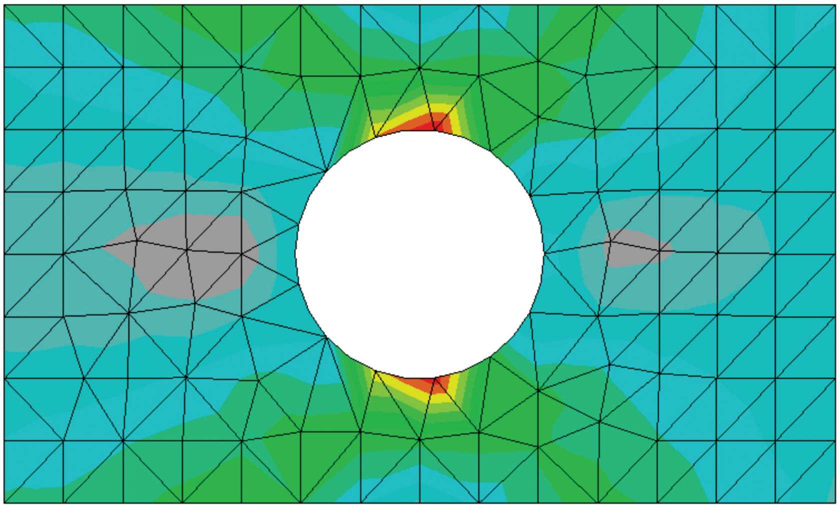


Figure 14.2 Averaged von Mises stress results also called nodal stresses. The stress plot shows continuous stress distribution because it is based on stresses extrapolated to nodes and then averaged on nodes.

23. What is the difference between loads and boundary conditions?

    Surface loads (loads applied to the boundary of solution domain) and restraints (supports) are both boundary conditions. Loads are called natural boundary conditions and restraints are called essential boundary conditions. The only type of loads that cannot be classified as boundary conditions are volume loads like gravity or inertial load.

24. How many sets of loads and restraints may be defined for one linear static analysis?

    You can define as many different load cases as you wish (if your software allows that), because the stiffness matrix remains the same, no matter what load is applied. However, only one set of restraints may be defined because the stiffness matrix changes from one set of restraints to the next.

25. What is the difference between boundary conditions applied to the geometry and the FEA model?

    The common way to apply loads and restraints is to define them on CAD geometry. These loads and restraints are converted into nodal loads and restraints during the process of meshing. Notice that meshing is commonly understood as discretization of geometry, but everything else in the mathematical model is also discretized; loads, restraints, and mass are discretized and applied to nodes. There is nothing that would be continuous in a finite element model.

    a. *Real Loads and Restraints:* Real loads acting on a structure are continuous like pressure, own weight, inertial forces, etc. Point loads or line load do not exist in nature. The same applies to restraints.

b. *Loads and restraints in the FE model:* Loads in the finite element model is represented as a set of forces and moments acting on nodes. Restraints are defined on nodes as prescribed displacement; a fixed restraint defines displacements equal to zero.

26. What are the major types of FEA?

    This depends on how FEA is classified. There are structural and thermal FEA, linear and nonlinear FEA, static and vibration (dynamic) FEA, etc. Also, we can classify FEA by objective like design FEA when it is used as a design tool, forensic analysis to find out why a structure failed, or an optimization exercise to improve the existing design.

27. What are the types of nonlinearity?

    Nonlinear analysis is the one in which the stiffness of the structure changes during the process of load application. That change needs to be modeled in the process of iterative solution required in nonlinear analysis. There are different sources of nonlinearity; the following is not a complete list:

    a. nonlinear supports
    b. contact stress
    c. tension/compression only links
    d. nonlinear elastic foundation
    e. offset or eccentric loads
    f. loads causing the loss of elastic stability (buckling)
    g. geometric nonlinearity
    h. material nonlinearity

28. How to reduce the size of the finite-element model

    a. *Taking Advantage of Repetitive Geometry:* Sometimes, geometry and boundary conditions (loads and restraints) are characterized by a certain symmetry such as symmetry, antisymmetry, and cyclic symmetry. In those cases, it is enough to model only one portion of the model and simulate the remaining portions with proper boundary conditions. The extra time spent on the modification of geometry is usually well worth it. Symmetry and anti-symmetry boundary conditions should be used very carefully in modal analysis; symmetry boundary conditions will eliminate all anti-symmetric modes and anti-symmetry boundary conditions will eliminate symmetric modes.

    b. *Defeaturing Geometry:* Defeaturing of geometry, meaning deleting structurally insignificant details, is particularly important when working with complex CAD geometry. Geometry produced in CAD is usually not oriented toward the FEA. If used "as is," it may result in very complicated models that are too large to solve. Defeaturing has to be done very carefully to avoid removing features important for the objective of analysis.

c. *Using Shell or Beam Elements in Place of Solid Elements:* Some thin walled models such as sheet metal lend themselves to meshing with shell elements rather than with solids. This requires significant idealization effort in order to represent solid geometry with surfaces but results in significant simplification of the model. For certain classes of geometry, for example, weldments, meshing with solid elements is out of question and using the beam element is the only choice.

d. *Using 2D Representations (Plane Stress, Plane, Strain Axisymmetric):* Working with 2D models tremendously simplifies model generation, solution, and analysis of results, but is applicable only to a narrow class of problems.

29. What is the bandwidth of the stiffness matrix, why is it important, and how can it be reduced?

    Bandwidth is an important consideration when solving FEA equations with a banded solver. Bandwidth is a parameter describing the stiffness matrix of the model. The stiffness matrix is always symmetric and, in most cases, all nonzero elements of this matrix are grouped along the main diagonal. The bandwidth is the highest number of nonzero elements in any row. The larger the bandwidth is, the more time consuming and numerically demanding is the solution. For low bandwidth, nodes should be numbered in such a way that the numbering of nodes belonging to each element is as close as possible. Renumbering of nodes to minimize the bandwidth is done automatically by software and is transparent to the user. In a case where a frontal solver is used, element numbers rather than node numbers should be as close as possible.

30. Where mistakes are often made?

    Mistakes can be made just about anywhere during an FEA project, but the most common of them are as follows:

a. *Improper Use of Elements in a Mesh:* For example, one layer of first-order elements is placed across the thickness of wall in bending or elements incorrectly map to geometry.

b. *Using Excessively Distorted Elements:* Excessively distorted are often produced with an automatic mesh generator running on default settings; the user's input into meshing is almost always required.

c. *Too Few Elements in a Mesh:* If mesh is too coarse, it will not be able to model the structure properly; stress concentrations may not be detected.

d. *Improper Restraints:* A common mistake is an overly restrained model resulting in an overly stiff structure. Displacement and, consequently, stresses are then underestimated.

e. *Hasty Result Analysis, No Lessons Learned:* FEA provides a wealth of results, which need to be analyzed properly. Each project should contribute to the accumulated corporate experience and mechanism to assure that it should be put in place.

31. What are alternatives to FEA?

    As far as computer-aided engineering is concerned, we can use other numerical tools like finite difference method or boundary element method. Also, meshless methods are being developed and look very promising.

    However, let us not forget about obvious ones like long-hand calculations, computerized tables, and testing. These methods are less expensive and often more reliable. You should always take advantage of hand calculations, handbooks, or other non-numerical methods to determine the the order of magnitude of expected results at least.

32. What is the golden rule of FEA?

    An FE model (or any model for that matter) may never be accepted as a final and true description of the system. At best it may be regarded as a good enough description of certain aspects that are of particular interest to us. The ultimate objective of the FEA is not to find exact solution such as displacement or stress. The objective is to obtain a reasonable approximation of reality that will allow making a correct design decision.

Chapter 15
FEA Resources

There are many sources of finite element analysis (FEA) expertise available to users that include but are not limited to the following:

- engineering textbooks
- software specific manuals
- engineering journals
- professional development courses that include face to face and distant learning
- FEA users' groups
- government organizations

With so many applications for FEA, attempts have been made to create a governing body overlooking FEA standards and practices. One of the leading organizations in this field is the National Agency for Finite Element Methods and Standards (NAFEMS). It was founded in the United Kingdom in 1983 with the specific objective: "To promote the safe and reliable use of finite element and related technology."

The NAFEMS has published many excellent handbooks such as the following:

- a finite-element primer
- a finite-element dynamics primer
- guidelines to finite-element practice
- background to benchmarks

Many professional organizations offer professional development courses in the field of FEA. In particular, the SAE International offers on-line training seminars closely related to the topics of this book:

- "Finite Element Analysis for Design Engineers" WB1241
- "Vibration Analysis with FEA" WB1401

More information may be found at http://training.sae.org/webseminars/.

References

Engineering literature offers a large selection of reference material on FEA and on engineering analysis closely related to FEA; we will list only a few of them.

1. Adams V. and Askenazi A. Building Better Products With Finite Element Analysis. OnWord Press: New Mexico. 1999.
2. Incropera F., Dewitt D., Bergman T., and Lavine A. *Fundamentals of Heat and Mass Transfer*, fifth edition. John Wiley & Sons, Inc.:New Jersey. 2007.
3. Inman D. *Engineering Vibration*. Prentice Hall:New Jersey. 1994.
4. Kim N.H. and Sankar B.V. *Introduction to Finite Element Analysis and Design*. John Wiley & Sons, Inc:New Jersey. 2008.
5. Logan D. *A First Course in the Finite Element Method*, fourth edition. Brooks/Cole:Boston. 2007.
6. Spyrakos C. *Finite Element Modeling in Engineering Practice*. West Virginia University Printing Services:West Virginia. 1996.
7. Szabo B. and Babuska I. *Finite Element Analysis*. John Wiley & Sons, Inc:New Jersey. 1991.
8. Zienkiewicz O. and Taylor R. *The Finite Element Method*. McGraw-Hill Book Company:New York. 2000.

Chapter 16
Glossary of Terms

Automeshing: Process of automatic mesh creation.

Averaged stresses: These stresses are calculated at nodes by averaging stresses at a node as reported by all elements sharing that node. Nodal stresses are "smoothed out" and by virtue of averaging produce continuous stress distributions in the model, also called nodal stresses.

Beam element: Element produced by meshing line (may be straight or curved).

Black body: A black body is a theoretically ideal radiator and an absorber of energy at all electromagnetic wavelengths.

Boundary Element Method: An alternative to the FEA method for solving field problems, only the boundary of the solution domain needs to be discretized. Boundary element method is very efficient for analyzing compact 3D shapes, but difficult to use on more "spread out" shapes.

CAD: Computer Aided design.

CAD specific geometry: Geometry with all details required for manufacturing.

Cleanup: Removing and/or repairing geometric features that would prevent the mesher from creating the mesh or would result in an incorrect mesh (with highly distorted or degenerated elements).

Conjugate heat transfer: Heat transfer process that involves variations of temperature within solids and fluids, due to thermal interaction between the solids and fluids.

Convergence criterion: A condition that must be satisfied in order for an automated convergence process to stop.

Convergence process: This is a process of systematic changes in the mesh in order to see how the data of interest change with the choice of the mesh and (hopefully) prove that the data of interest are not significantly dependent on the choice of discretization. A convergence process, or analysis, can be performed as *h*-convergence or *p*-convergence.

Defeaturing: The process of removing (or suppressing) geometric features from CAD geometry in order to simplify the finite element mesh.

Degree of freedom (DOF): Ability of node to perform generalized displacement. In structural analysis, node may have up to 6 DOF depending on element type.

Discretization: This defines the process of splitting up a continuous mathematical model into discrete "pieces" called elements. A visible effect of discretization is the finite element mesh. However, mass, loads, and restraints are also discretized.

Discretization error: This type of error affects FEA results because FEA works on an assembly of discrete elements (mesh) rather than on a continuous structure. The finer the finite element mesh, the lower the discretization error, but the solution takes more time to complete.

Displacement interpolation function: A polynomial function defined on nodal displacements. It calculates displacements (or temperatures) inside an element (in element volume, on faces and along edges) based on known nodal displacements.

Distortion of element: Departure from element ideal shape; happens as a result of mapping onto model geometry.

Emissivity: Emissivity is the ratio of the energy radiated from a material's surface to that radiated from a blackbody.

FEA: Finite element analysis.

FEA specific geometry: Geometry intended for FEA, usually derived from CAD geometry by defeaturing, idealization, and cleanup.

Finite Difference Method: An alternative to the FEA; solution domain is discretized into a grid. The finite difference method is generally less efficient for solving structural and thermal problems, but is often used in fluid dynamics problems.

Finite Element: the building blocks of a mesh, defined by position of their nodes and by functions approximating distribution of sought for quantities, such as displacements or temperatures.

Finite Volumes Method: This is yet another alternative to the FEA method of solving field problem, similar to the finite difference method.

Following load: Load that retains is orientation in relation to deforming model. Distinction between the following and nonfollowing loads is important in nonlinear geometry with large displacements analysis.

Frequency response analysis: Type of vibration analysis where excitation is harmonic and it's magnitude is a function of the excitation frequency.

Geometric stiffness: Stiffness that characterizes structural response to an applied load. Geometric stiffness is a function of shape, material properties, and restraints.

***h*-element:** Elements, for which the order does not change during analysis. This means that a first-order element remains a first order throughout solution. Convergence analysis of a model using h-elements is done by refining the mesh and comparing

results (like displacement and stresses) before and after refinement. The name *h*-element comes from the element characteristic dimension *h*, which is reduced in consecutive mesh refinements.

Idealization: This refers to making simplifying assumptions in the process of creating a mathematical model of an analyzed structure. Idealization may involve simplifying geometry, replacing thin walls with midplane surfaces or using 2D simplifications.

Idealization error: This type of error results from the fact that analysis is based on and idealized mathematical and not on a real-life object. Geometry, material properties, loads, and restraints all are idealized in models submitted to FEA.

Linear material: This is a type of material where stress is linearly proportional to strain.

Mapping: Process of changing element shape from its ideal shape to the shape it assumes in the mesh, element mapping takes place during meshing.

Membrane element: Element produced by meshing surface, and as opposed to shell element, membrane element does not model bending stresses; it has no bending stiffness.

Mesh diagnostic: Feature of FEA program that determines (in cases when meshing fails) which geometric entities prevented meshing.

Meshing: Process of discretizing the model geometry. As a result of meshing, the originally continuous geometry is represented by an assembly of finite elements.

Modal analysis: Also called "frequency analysis." A modal analysis calculates the natural frequencies of a structure as the associated modes (shapes) of vibration. Modal analysis does not calculate displacements or stresses.

Modeling error: This type of error results from idealizations that are introduced in the process of creating a mathematical model. See idealization error.

Non-averaged stresses: Stresses reported for element without averaging them with stresses reported by neighboring elements, also called element stresses.

Non-following load: Load that retains its orientation in relation to the external coordinate system and not to the deforming model. Distinction between the following and nonfollowing loads is important in nonlinear geometry analysis with large displacements.

Numerical error: The accumulated rounding off of numbers causes this type of error by the numerical solver in the solution process. The value of numerical errors is usually very low.

***p*-adaptive solution:** Solution method using *p*-elements. A *p*-adaptive solution provides results with specified accuracy, but is more time consuming than *h* solution using *h* elements.

***p*-element:** *p*-elements are elements that do not have predefined order. Solution of a *p*-element model requires several iterations, while element order is upgraded until the difference in user-specified measures (e.g., global strain energy and RMS stress) becomes less than the requested accuracy. The name *p*-element comes from the *p*-order of

polynomial displacement interpolation functions which are gradually upgraded during the iterative solution along all edges.

Preload: Preload is a load that modifies the stiffness of a structure. Preload is important in a modal analysis where it may significantly change natural frequencies. In some cases, preload must be also considered in static analysis.

Restraints: Any condition defined for displacements of model. Supports, symmetry boundary conditions, etc. are examples of restraints.

Rigid body mode: A mode with zero frequency found in structures that are not fully restrained or not restrained at all. A structure with no restraints has six rigid body modes. Also see rigid body motion.

Rigid body motion: Rigid body motion is the ability to move without experiencing any deformation. In the case of a fully supported structure, the only way it can move under load is to deform its shape. If a structure is not fully supported, it can move as a rigid body without any deformation.

Root Mean Square (RMS) stress: Square root of the mean of the squares of stress; RMS stress may be used as a convergence criterion.

Shell element: Shell elements are intended for meshing surfaces of 3D model. Triangular shell elements have three corner nodes. If this is a second-order triangular element, it also has midside nodes, making the total number of nodes equal to six. Each node of a shell element has 6 DOF.

Singularity: A condition when a mathematical model has no solution in certain location; common stress singularity is a sharp re-entrant edge where stress is infinitely high (singular), meaning that no stress solution exists in that location.

Small displacement assumption: Analysis based on small displacements assumes that displacements caused by the applied load are small enough as not to significantly change structure stiffness. Analysis based on this assumption of small displacements is also called linear geometry analysis. However, the magnitude of displacements itself is not the deciding factor in determining whether or not those small displacement analysis may be used. What matters is whether or not those displacements significantly change the stiffness of the analyzed structure.

Solid element: 3D element produced by meshing volume.

Steady state thermal analysis: Thermal analysis that assumes that temperature does not change with time.

Symmetry boundary conditions: These refer to displacement conditions defined on a flat model boundary allowing only for in-plane displacement and restricting any out-of-plane displacement components. Symmetry boundary conditions are very useful in structural analysis for reducing model size if model geometry, load, and supports are all symmetric.

Tetrahedral element: This is a type of element intended for meshing volumes of 3D models. It belongs to the family of solid elements beside hexahedral (brick) and pentahedral (wedge) elements. It is the most widely used solid element. A tetrahedral element has four triangular faces and four corner nodes. If this is used as a second-order element, it also has midside nodes, making then the total number of nodes equal to 10. Each node of a tetrahedral element has 3 DOF.

Thermal analysis: Thermal analysis finds temperature distribution and heat flow in a structure.

Time response analysis: Type of vibration analysis where excitation is a function of time.

Transient thermal analysis: Thermal analysis where temperature and other heat transfer parameters change with time.

Ultimate strength: This refers to the maximum stress that may occur in a structure. If the ultimate strength is exceeded, failure will take place (the part will break). Ultimate strength is usually much higher than tensile strength.

Validation: Validation determines if an FEA model correctly represents the reality from the perspective of the intended use of the model. It checks if results correctly describe the real-life behavior of the analyzed object.

Verification: Verification checks if the mathematical model, as submitted to be solved with FEA, has been correctly discretized and solved.

Von Mises stress: This is a stress measure that takes into consideration all six stress components of a 3D state of stress. Von Mises stress, also called Huber stress, is a very convenient and popular way of presenting FEA results because it is a scalar non-negative value and because the magnitude of von Mises stress can be used to determine safety factors for materials exhibiting elastic–plastic properties, such as steel, aluminum, or cooper.

Yield strength: This refers to the maximum stretching that can be allowed in a model before plastic deformation takes place.

Chapter 17
List of Exercises

Chapter	Parasolid	SOLIDWORKS
4	4.01.HOLLOW_PLATE.x_t	4.01.HOLLOW_PLATE.sldprt
	4.02.L_BRACKET.x_t	4.02.L_BRACKET.sldprt
	4.03.2D_BEAM.x_t	4.03.2D_BEAM.sldprt
5	5.01.BRACKET01.x_t	5.01.BRACKET01.sldprt
	5.02.CANTILEVER_BEAM.x_t	5.02.CANTILEVER_BEAM.sldprt
6	6.01.BRACKET02.x_t	6.01.BRACKET02.sldprt
	6.02.SHAFT.x_t	6.02.SHAFT.sldprt
	6.03.PRESSURE_TANK.x_t	6.03.PRESSURE_TANK.sldprt
	6.04.RING.x_t	6.04.RING.sldprt
	6.05.LINK.x_t	6.05.LINK.sldprt
7	7.01.CANTILEVER_BEAM.x_t	7.01.CANTILEVER_BEAM.sldprt
	7.02.TORSION_SHAFT.x_t	7.02.TORSION_SHAFT.sldprt
	7.03.ROUND_PLATE.x_t	7.03.ROUND_PLATE.sldprt
	7.04.LINK.x_t	7.04.LINK.sldprt
	7.05.SLIDING_SUPPORT.x_t	7.05.SLIDING_SUPPORT.sldprt
	7.06.CLAMP01.x_t	7.06.CLAMP01.sldprt
	7.07.CLAMP02.x_t	7.07.CLAMP02.sldprt
	7.08.SHRINK_FIT.x_t	7.08.SHRINK_FIT.sldasm
8	8.01.BRACKET_NL.x_t	8.01.BRACKET_NL.sldprt
	8.02.L_BRACKET.x_t	8.02.L_BRACKET.sldprt
9	9.01.TUNING_FORK.x_t	9.01.TUNING_FORK.sldprt
	9.02.BOX.x_t	9.02.BOX.sldprt
	9.03.AIRPLANE.x_t	9.03.AIRPLANE.sldprt
	9.04.BALL.x_t	9.04.BALL.sldprt
	9.05.LINK.x_t	9.05.LINK.sldprt
	9.06.BLADE.x_t	9.06.BLADE.sldprt
	9.07.COLUMN.x_t	9.07.COLUMN.sldprt
	9.08.BRACKET.x_t	9.08.BRACKET.sldprt

Chapter	Parasolid	SOLIDWORKS
10	10.01.NOTCHED_COLUMN. x_t	10.01.NOTCHED_COLUMN. sldprt
	10.02.BUTTON.x_t	10.02.BUTTON.sldprt
	10.03.CURVED_COLUMN.x_t	10.03.CURVED_COLUMN.sldprt
	10.04.STAND. x_t	10.04.STAND. sldasm
	10.05.CURVED_SHEET.x_t	10.05.CURVED_SHEET.sldprt
11	11.01.HAMMER. x_t	11.01.HAMMER. sldprt
	11.02.ELBOW_PIPE.x_t	11.02.ELBOW_PIPE.sldprt
	11.03.CENTRIFUGE.x_t	11.03.CENTRIFUGE.sldasm
	11.04.PLANK.x_t	11.04.PLANK.sldprt
12	12.01.BRACKET.x_t	12.01.BRACKET.sldprt
	12.02.HEAT_SINK.x_t	12.02.HEAT_SINK.sldasm
	12.02.HEAT_SINK_sharp_edges.x_t	
	12.03.CHANNEL.x_t	12.03.CHANNEL.sldprt
	12.04.SPACE_HEATER.x_t	12.04.SPACE_HEATER.sldasm

Index

Note: Locators followed by '*f*' and '*t*' refer to figures and tables, respectively.

About the Author

Dr. Paul Kurowski

Dr. Paul Kurowski obtained his M.Sc. and Ph.D. in Applied Mechanics from Warsaw Technical University, Warsaw, Poland. He completed postdoctoral work at Warsaw Technical University, Kyoto University, and the University of Western Ontario. Paul Kurowski is professor in the Faculty of Engineering at the University of Western Ontario. He teaches undergraduate and graduate courses in Product Design, Finite Element Methods, Computer Aided Engineering, Vibration Analysis and others. Paul is also the President of Design Generator Inc., a consulting firm with expertise in Product Development and training in Computer Aided Engineering methods. He has published many technical papers and created and taught professional development seminars in the field of Finite Element Analysis for SAE International, ASME, Professional Engineers of Ontario, Parametric Technology Corporation (PTC), Rand Worldwide, SOLIDWORKS Corporation and others.

His professional interests revolve around finding the best ways of using Computer Aided Engineering (CAE) methods for faster and more effective product development processes where computer models replace physical prototypes.

Paul is a member of the Association of Professional Engineers of Ontario and the SAE International. He may be contacted at HYPERLINK "http://www.designgenerator.com" www.designgenerator.com.

Finite Element Analysis for Design Engineers

Second Edition

Paul M. Kurowski

Finite Element Analysis (FEA) has been widely implemented by the automotive industry as a product design tool for design engineers who use it to reduce product development time and cost. This book serves as a guide for FEA users and addresses the specific needs of design engineers. It provides clear presentation of FEA fundamentals and implementation that will help practitioners to avoid misapplication of this tool. Easy to follow examples of FEA fundamentals are clearly presented and are ready for implementation during the product design process. The FEA is fully explored in this practical approach that includes:

- Understanding FEA basics
- Commonly used modeling techniques
- Commonly used types of FEA analyses
- Frequently made FEA errors and their effect on the quality of results
- Implementation of FEA in the design process
- Hands-on simple and informative exercises

This indispensable guide provides design engineers with proven methods to analyze their own work while it is still in the form of easily modifiable CAD models. Exercises provide examples for improving the FEA hands-on process to deliver quick turnaround times and prompt implementation.

About the Author:

Dr. Paul Kurowski is a professor in the Department of Mechanical and Materials Engineering at the University of Western Ontario in London, Ontario, Canada. His teaching includes finite element analysis, machine design, kinematics and dynamics of machines, mechanical vibration, computer aided engineering and product design. He is also the President of Design Generator Inc., a consulting firm specializing in design analysis and training in Computer Aided Engineering methods. Dr. Kurowski has published multiple technical papers and taught professional development seminars for the SAE International, the American Society of Mechanical Engineers, the Association of Professional Engineers of Ontario, the Parametric Technology Corp. (PTC), Rand Worldwide, SOLIDWORKS Corp. and other companies and professional organizations. He is a member of the SAE International and the Association of Professional Engineers of Ontario. Dr. Kurowski obtained his M.Sc. and Ph.D. in Applied Mechanics from Warsaw Technical University and completed postdoctoral work at Kyoto University.

R-449