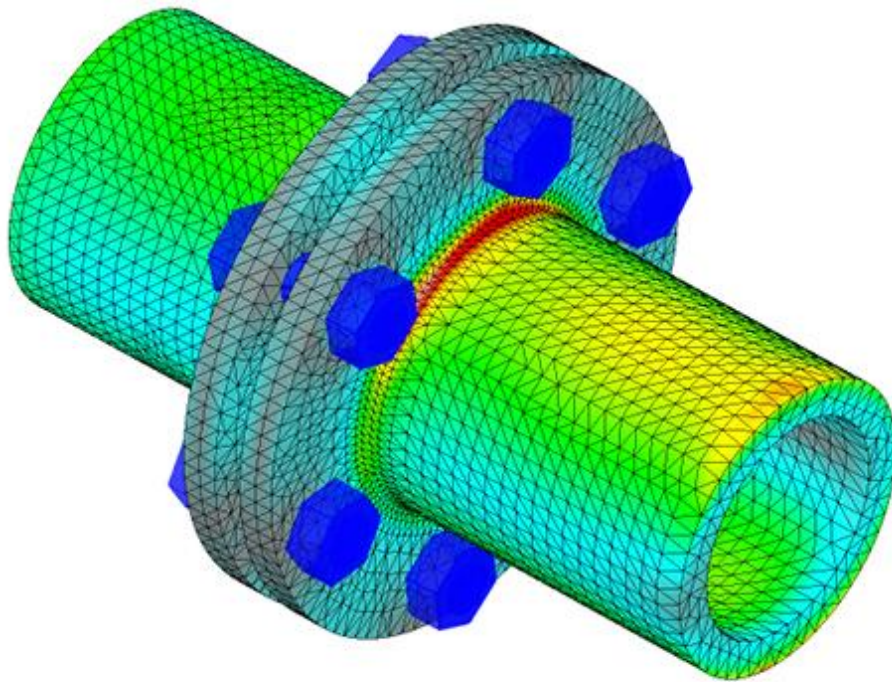


Engineering Analysis

with SOLIDWORKS® Simulation 2015



Paul M. Kurowski



Design Generator Inc.



Better Textbooks. Lower Prices.
www.SDCpublications.com

Table of contents

About the Author	Error! Bookmark not defined.
Acknowledgements	Error! Bookmark not defined.
Table of contents	1
Before You Start	7
Notes on hands-on exercises and functionality of Simulation	
Prerequisites	
Selected terminology	
1: Introduction	11
What is Finite Element Analysis?	
Finite Element Analysis used by Design Engineers	
Objectives of FEA for Design Engineers	
What is SOLIDWORKS Simulation?	
Fundamental steps in an FEA project	
Errors in FEA	
A closer look at finite elements	
What is calculated in FEA?	
How to interpret FEA results	
Units of measure	
Using online help	
Limitations of Static studies	
2: Static analysis of a plate	37
Using the SOLIDWORKS Simulation interface	
Linear static analysis with solid elements	
Controlling discretization error with the convergence process	
Finding reaction forces	
Presenting FEA results in a desired format	

3: Static analysis of an L-bracket Error! Bookmark not defined.

- Stress singularities
- Differences between modeling errors and discretization errors
- Using mesh controls
- Analysis in different SOLIDWORKS configurations
- Nodal stresses, element stresses

4: Static and frequency analyses of a pipe support Error! Bookmark not defined.

- Use of shell elements
- Frequency analysis
- Bearing load

5: Static analysis of a link Error! Bookmark not defined.

- Symmetry boundary conditions
- Preventing rigid body motions
- Limitations of the small displacements theory

6: Frequency analysis of a tuning fork and a plastic part Error! Bookmark not defined.

- Frequency analysis with and without supports
- Rigid body modes
- The role of supports in frequency analysis
- Symmetric and anti-symmetric modes

7: Thermal analysis of a pipe connector and a heater Error! Bookmark not defined.

- Analogies between structural and thermal analysis
- Steady state thermal analysis
- Analysis of temperature distribution and heat flux
- Thermal boundary conditions
- Thermal stresses
- Vector plots

8: Thermal analysis of a heat sink Error! Bookmark not defined.

- Analysis of an assembly
- Global and local Contact conditions
- Steady state thermal analysis
- Transient thermal analysis
- Thermal resistance layer
- Use of section views in result plots

- 9: Static analysis of a hanger** Error! Bookmark not defined.
Global and local Contact conditions
Hierarchy of Contact conditions
- 10: Thermal stress analysis of a bi- metal loop** Error! Bookmark not defined.
Thermal deformation and thermal stress analysis
Eliminating rigid body motions
Converting Sheet Metal bodies to Solid bodies
"Parasolid" round trip
Saving model in deformed shape
- 11: Buckling analysis of I-beam** Error! Bookmark not defined.
Buckling analysis
Buckling load safety factor
Stress safety factor
- 12: Static analysis of a bracket using adaptive solution methods** Error! Bookmark not defined.
h-adaptive solution method
p-adaptive solution method
Comparison between h-elements and p-elements
- 13: Drop test** Error! Bookmark not defined.
Drop test analysis
Stress wave propagation
Direct time integration solution
- 14: Selected nonlinear problems** Error! Bookmark not defined.
Large displacement analysis
Analysis with shell elements
Membrane effects
Following and non-following load
Nonlinear material analysis
Residual stress
- 15: Mixed meshing problem** Error! Bookmark not defined.
Using solid and shell elements in the same mesh
Mixed mesh compatibility
Manual and automatic finding of contact sets
Shell Manager

16: Analysis of a weldment using beam elements Error! Bookmark not defined.

- Different levels of idealization implemented in finite elements
- Preparation of a SOLIDWORKS model for analysis with beam elements
- Beam elements and truss elements
- Analysis of results using beam elements
- Limitations of analysis with beam elements

17: Review of 2D problems Error! Bookmark not defined.5

- Classification of finite elements
- 2D axi-symmetric element
- 2D plane stress element
- 2D plane strain element

18: Vibration Analysis - Modal Time History and Harmonic Error! Bookmark not defined.

- Modal Time History analysis (Time Response)
- Harmonic analysis (Frequency Response)
- Modal Superposition Method
- Damping

19: Analysis of random vibration Error! Bookmark not defined.1

- Random vibration
- Power Spectral Density
- RMS results
- PSD results
- Modal excitation

20: Miscellaneous topics	Error! Bookmark not defined.1
Mesh quality	
Solvers and solvers options	
Displaying mesh in result plots	
Automatic reports	
E drawings	
Non uniform loads	
Frequency analysis with pre-stress	
Interference fit analysis	
Rigid connector	
Pin connector	
Bolt connector	
Remote load/mass	
Weld connector	
Bearing connector	
Cyclic symmetry	
Strongly nonlinear problem	
Submodeling	
Terminology issues in the Finite Element Analysis	
21: Implementation of FEA into the design process	Error! Bookmark not defined.5
Verification and Validation of FEA results	
FEA driven design process	
FEA project management	
FEA project checkpoints	
FEA reports	
22: Glossary of terms	Error! Bookmark not defined.3
23: Resources available to FEA users	Error! Bookmark not defined.1
24: List of exercises	Error! Bookmark not defined.7

Before You Start

Notes on hands-on exercises and functionality of Simulation

This book goes beyond a standard software manual because its unique approach concurrently introduces you to **SOLIDWORKS Simulation** software and the fundamentals of Finite Element Analysis (FEA) through hands-on exercises. We recommend that you study the exercises in the order presented in the book. As you go through the exercises, you will notice that explanations and steps described in detail in earlier exercises are not repeated in later chapters. Each subsequent exercise assumes familiarity with software functions discussed in previous exercises and builds on the skills, experience, and understanding gained from previously presented problems. Exceptions to the above are chapters 21, 22, 23, 24 which do not include hands-on exercises.

Exercises in this book require different levels of **SOLIDWORKS Simulation** functionality and this depends on which **Simulation** product is used. The **SOLIDWORKS Simulation Product Matrix** document is available at:

<http://www.SOLIDWORKS.com/sw/products/simulation/simulation-matrices.htm>

This book deals with structural analysis using **SOLIDWORKS Simulation**. Therefore, **Motion** analysis won't be covered. **SimulationXpress** is a simplified version of **SOLIDWORKS Simulation** and will not be covered either.

All exercises in this book use **SOLIDWORKS** models, which can be downloaded from www.SDCpublications.com. These exercises do not contain any **Simulation** studies; you are expected to create all studies, results plots, and graphs yourself. The only exceptions are exercises in chapter 4 (MISALIGNMENT), 20 and 21 which come with **Simulation** studies fully or partially defined. All problems presented here have been solved with **SOLIDWORKS Simulation Premium** running on Windows 7 in a 64 bit operating environment.

We encourage you to explore each exercise beyond its description by investigating other options, other menu choices, and other ways to present results. You will soon discover that the same simple logic applies to all functions in **SOLIDWORKS Simulation**.

“Engineering Analysis with SOLIDWORKS Simulation” is an introductory text. The focus is more on understanding Finite Element Analysis than presenting all software capabilities. This book is not intended to replace software manuals. Therefore, not all **Simulation** capabilities will be covered, especially those of fatigue, design studies, optimization, advanced nonlinear thermal and dynamic analyses.

Readers of "Engineering Analysis with SOLIDWORKS Simulation" may wish to review the book “Thermal Analysis with SOLIDWORKS Simulation” (Figure 23-1) and “Vibration Analysis with SOLIDWORKS” (Figure 23-2), both published by the SDC Publications. These books are not introductory texts; they are designed for users who are familiar with topics presented in “*Engineering Analysis with SOLIDWORKS Simulation*”.

The knowledge acquired by the readers of "Engineering Analysis with SOLIDWORKS Simulation" will not be software specific. The same concepts, tools and methods will apply to any FEA software.

Prerequisites

The following prerequisites are recommended:

- An understanding of Statics, Kinematics and Dynamics
- An understanding of Mechanics of Materials
- An understanding of Heat Transfer
- An understanding of Mechanical Vibrations
- Experience with parametric, feature based solid modeling using SOLIDWORKS
- Familiarity with the Windows Operating System

Selected terminology

The mouse pointer plays a very important role in executing various commands and providing user feedback. The mouse pointer is used to execute commands, select geometry, and invoke pop-up menus. We use Windows terminology when referring to mouse-pointer actions.

Item	Description
Click	Self-explanatory
Double-click	Self-explanatory
Click-inside	Click the left mouse button. Wait a second, and then click the left mouse button inside the pop-up menu or text box. Use this technique to modify the names of items in SOLIDWORKS Simulation Manager .
Drag and drop	Use the mouse to point to an object. Press and hold the left mouse button down. Move the mouse pointer to a new location. Release the left mouse button.
Right-click	Click the right mouse button. A pop-up menu is displayed. Use the left mouse button to select a desired menu command.

All **SOLIDWORKS** file names appear in CAPITAL letters, even though the actual file names may use a combination of capital and small letters. Selected menu items and **SOLIDWORKS Simulation** commands appear in **bold**. **SOLIDWORKS** configurations, **SOLIDWORKS Simulation** folders, icon names and study names appear in *italics* except in captions and comments to illustrations. **SOLIDWORKS** and **Simulation** also appear in bold font. Bold font may also be used to draw reader's attention to particular term.

1: Introduction

What is Finite Element Analysis?

Finite Element Analysis, commonly called FEA, is a method of numerical analysis. FEA is used for solving problems in many engineering disciplines such as machine design, acoustics, electromagnetism, soil mechanics, fluid dynamics, and many others. In mathematical terms, FEA is a numerical technique used for solving field problems described by a set of partial differential equations.

In mechanical engineering, FEA is widely used for solving structural, vibration, and thermal problems. However, FEA is not the only available tool of numerical analysis. Other numerical methods include the Finite Difference Method, the Boundary Element Method, and the Finite Volume Method to mention just a few. However, due to its versatility and numerical efficiency, FEA has come to dominate the engineering analysis software market, while other methods have been relegated to niche applications. When implemented into modern commercial software, both FEA theory and numerical problem formulation become completely transparent to users.

Finite Element Analysis used by Design Engineers

FEA is a powerful engineering analysis tool useful in solving many problems ranging from very simple to very complex. Design engineers use FEA during the product development process to analyze the design-in-progress. Time constraints and limited availability of product data call for many simplifications of computer models. On the other hand, specialized analysts implement FEA to solve very complex problems, such as vehicle crash dynamics, hydro forming, and air bag deployment.

This book focuses on how design engineers use FEA, implemented in **SOLIDWORKS Simulation**, as a design tool. Therefore, we highlight the most essential characteristics of FEA as performed by design engineers as opposed to those typical for FEA performed by analysts.

FEA for Design Engineers: Another design tool

For design engineers, FEA is one of many design tools that are used in the design process and include CAD, prototypes, spreadsheets, catalogs, hand calculations, text books, etc.

FEA for Design Engineers: Based on CAD models

Modern design is conducted using CAD, so a CAD model is the starting point for analysis. Since CAD models are used for describing geometric information for FEA, it is essential to understand how to prepare CAD geometry in order to produce correct FEA results, and how a CAD model is different from an FEA model. This will be discussed in later chapters.

FEA for Design Engineers: Concurrent with the design process

Since FEA is a design tool, it should be used concurrently with the design process. It should drive the design process rather than follow it.

Limitations of FEA for Design Engineers

An obvious question arises: would it be better to have a dedicated specialist perform FEA and let design engineers do what they do best – design new products? The answer depends on the size of the business, type of products, company organization and culture, and many other tangible and intangible factors. A general consensus is that design engineers should handle relatively simple types of analysis, but do it quickly and of course reliably. Analyses that are very complex and time consuming cannot be executed concurrently with the design process, and are usually better handled either by a dedicated analyst or contracted out to specialized consultants.

Objectives of FEA for Design Engineers

The ultimate objective of using FEA as a design tool is to change the design process from repetitive cycles of “design, prototype, test” into a streamlined process where prototypes are not used as design tools and are only needed for final design validation. With the use of FEA, design iterations are moved from the physical space of prototyping and testing into the virtual space of computer simulations (Figure 1-1).

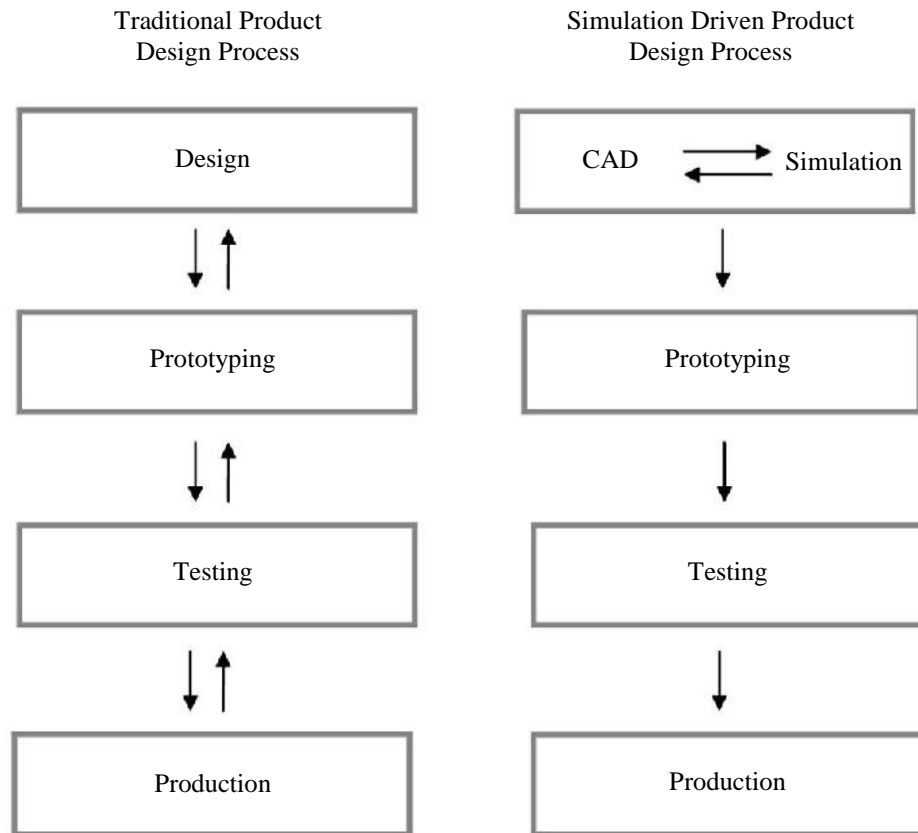


Figure 1-1: Traditional and Simulation* driven product development.

Traditional product development needs prototypes to support a design in progress. The process in Simulation-driven product development uses numerical models, rather than physical prototypes to drive development. In an Simulation driven product design process, the prototype is no longer a part of the iterative design loop.

**Simulation means here Finite Element Analysis.*

What is SOLIDWORKS Simulation?

SOLIDWORKS Simulation is a commercial implementation of FEA capable of solving problems commonly found in design engineering, such as the analysis of displacements, stresses, natural frequencies, vibration, buckling, heat flow, etc. It belongs to the family of engineering analysis software products originally developed by the Structural Research & Analysis Corporation (SRAC). SRAC was established in 1982 and since its inception has contributed to innovations that have had a significant impact on the evolution of FEA. In 1995 SRAC partnered with the **SOLIDWORKS** Corporation and created COSMOSWorks, one of the first **SOLIDWORKS** Gold Products, which became the top-selling analysis solution for the **SOLIDWORKS** Corporation. The commercial success of COSMOSWorks integrated with **SOLIDWORKS** CAD software resulted in the acquisition of SRAC in 2001 by Dassault Systèmes, parent of **SOLIDWORKS** Corporation. In 2003, SRAC operations merged with the **SOLIDWORKS** Corporation. In 2009, COSMOSWorks was re-named **SOLIDWORKS Simulation**.

SOLIDWORKS Simulation is integrated with **SOLIDWORKS** CAD software and uses **SOLIDWORKS** for creating and editing model geometry. **SOLIDWORKS** is a solid, parametric, feature-driven CAD system developed specifically for the Windows Operating System. Many other CAD and FEA programs were originally developed in a UNIX environment and only later ported to Windows, and therefore are less integrated with Windows than **SOLIDWORKS** and **SOLIDWORKS Simulation**.

Fundamental steps in an FEA project

The starting point for any **SOLIDWORKS Simulation** project is a **SOLIDWORKS** model, which can be a part or an assembly. First, material properties, loads, and restraints are defined. Next, as is always the case with using any FEA-based analysis tool, the model geometry is split into relatively small and simply shaped entities called finite elements. The elements are called “finite” to emphasize the fact that they are not infinitesimally small, but relatively small in comparison to the overall model size. Creating finite elements is commonly called meshing. When working with finite elements, the **SOLIDWORKS Simulation** solver approximates the sought solution (for example stress) by assembling the solutions for individual elements.

From the perspective of FEA software, each application of FEA requires three steps:

- ❑ Preprocessing of the FEA model, which involves defining the model and then splitting it into finite elements
- ❑ Solving for desired results
- ❑ Post-processing for results analysis

We will follow the above three steps in every exercise. From the perspective of FEA methodology, we can list the following FEA steps:

- ❑ Building the mathematical model
- ❑ Building the finite element model by discretizing the mathematical model
- ❑ Solving the finite element model
- ❑ Analyzing the results

The following subsections discuss these four steps.

Building the mathematical model

The starting point to analysis with **SOLIDWORKS Simulation** is a **SOLIDWORKS** model. Geometry of the model needs to be meshable into a correct finite element mesh. This requirement of mesh-ability has very important implications. We need to ensure that the CAD geometry will indeed mesh and that the produced mesh will provide the data of interest (e.g. stresses, displacements or temperature distribution) with acceptable accuracy.

The necessity to mesh often requires modifications to the CAD geometry, which can take the form of defeaturing, idealization, and/or clean-up:

Term	Description
Defeaturing	The process of removing geometry features deemed insignificant for analysis, such as outside fillets, chamfers, logos, etc.
Idealization	A more aggressive exercise that may depart from solid CAD geometry by, for example, representing thin walls with surfaces and beams with lines.
Clean-up	Sometimes needed because geometry must satisfy high quality requirements to be meshable. To clean-up, we can use CAD quality control tools to check for problems like, for example, sliver faces, multiple entities, etc. that could be tolerated in the CAD model, but would make subsequent meshing difficult or impossible.

It is important to mention that we do not always simplify the CAD model with the sole objective of making it meshable. Often we must simplify a model even though it would mesh correctly “as is”, because the resulting mesh would be large (in terms of the number of elements) and consequently, the meshing and the analysis would take too long. Geometry modifications allow for a simpler mesh and shorter meshing and computing times.

Sometimes, geometry preparation may not be required at all. Successful meshing depends as much on the quality of geometry submitted for meshing as it does on the capabilities of the meshing tools implemented in the FEA software.

Having prepared a meshable, but not yet meshed geometry, we now define material properties (these can also be imported from a CAD model), loads and restraints, and provide information on the type of analysis that we wish to perform. This procedure completes the creation of the mathematical model (Figure 1-2). Notice that the process of creating the mathematical model is not FEA specific. FEA has not yet entered the picture.

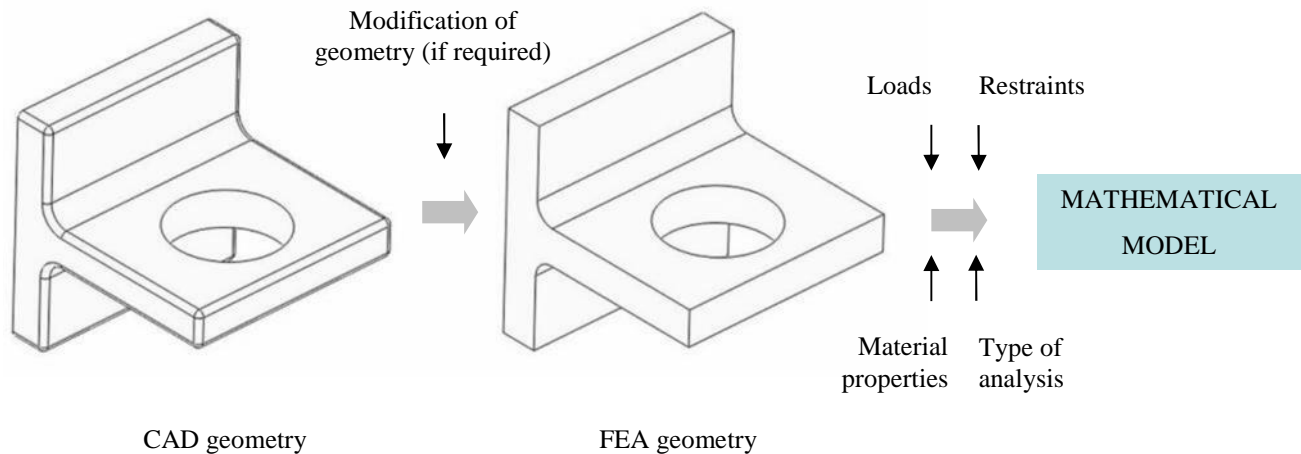


Figure 1-2: Building the mathematical model.

The process of creating a mathematical model consists of the modification of CAD geometry (here removing outside fillets), definition of loads, restraints, material properties, and definition of the type of analysis (for example linear static) that is to be performed.

You may review the differences between CAD geometry and FEA geometry using part model BRACKET DEMO in its two configurations: *01 fully featured* and *02 defeatured*.

Building the finite element model

The mathematical model now needs to be split into finite elements using the process of discretization, more commonly known as meshing (Figure 1-3). Geometry, loads, and restraints are all discretized. The discretized loads and restraints are applied to the nodes of the finite element mesh which is a discretized representation of geometry.

Solving the finite element model

Having created the finite element model, we now utilize a solver provided in **SOLIDWORKS Simulation** to produce the desired data of interest (Figure 1-3).

Analyzing the results

Often the most difficult step of FEA is analyzing the results. Proper interpretation of results requires that we understand all simplifications (and errors they introduce) in the first three steps: defining the mathematical model, meshing, and solving.

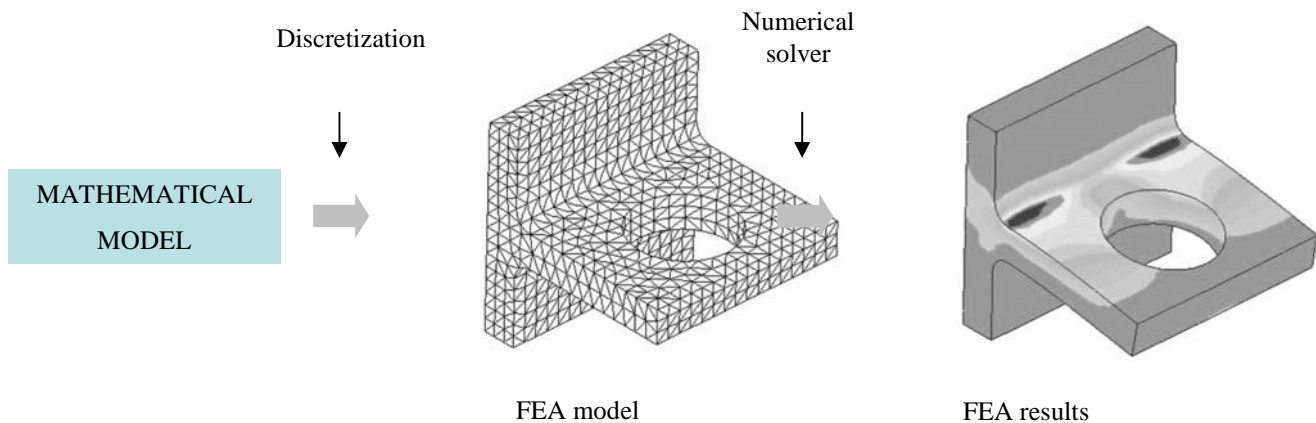


Figure 1-3: Building the finite element model.

The mathematical model is discretized into a finite element model. This completes the pre-processing phase. The FEA model is then solved with one of the numerical solvers available in SOLIDWORKS Simulation.

Errors in FEA

The process illustrated in Figure 1-2 and Figure 1-3 introduces unavoidable errors. Formulation of a mathematical model introduces modeling errors (also called idealization errors), discretization of the mathematical model introduces discretization errors, and solving introduces solution errors. Of these three types of errors, only discretization errors are specific to FEA. Modeling errors affecting the mathematical model are introduced before FEA is utilized and can only be controlled by using correct modeling techniques. Solution errors are caused by the accumulation of round-off errors.

A closer look at finite elements

Meshing splits continuous mathematical models into finite elements. The type of elements created by this process depends on the type of geometry meshed. **SOLIDWORKS Simulation** offers three types of three dimensional (3D) elements: solid elements for meshing solid geometry, shell elements for meshing surface geometry and beam elements for meshing wire frame geometry. **SOLIDWORKS Simulation** also works with two dimensional (2D) elements: plane stress elements, plane strain elements, and axi-symmetric elements.

Before proceeding, we need to clarify an important terminology issue. In CAD terminology, “solid” denotes the type of geometry: solid geometry (as opposed to surface or wire frame geometry). In FEA terminology, “solid” denotes the type of element used to mesh the solid CAD geometry.

Solid elements

The type of geometry that is most often used for analysis with **SOLIDWORKS Simulation** is solid CAD geometry. Meshing of this geometry is accomplished with tetrahedral solid elements, commonly called “tets” in FEA jargon. The tetrahedral solid elements in **SOLIDWORKS Simulation** can either be first order elements (“draft quality”), or second order elements (“high quality”). The user decides whether to use draft quality or high quality elements for meshing. However, as we will soon prove, only high quality elements should be used for an analysis of any importance. The difference between first and second order tetrahedral elements is illustrated in Figure 1-4.

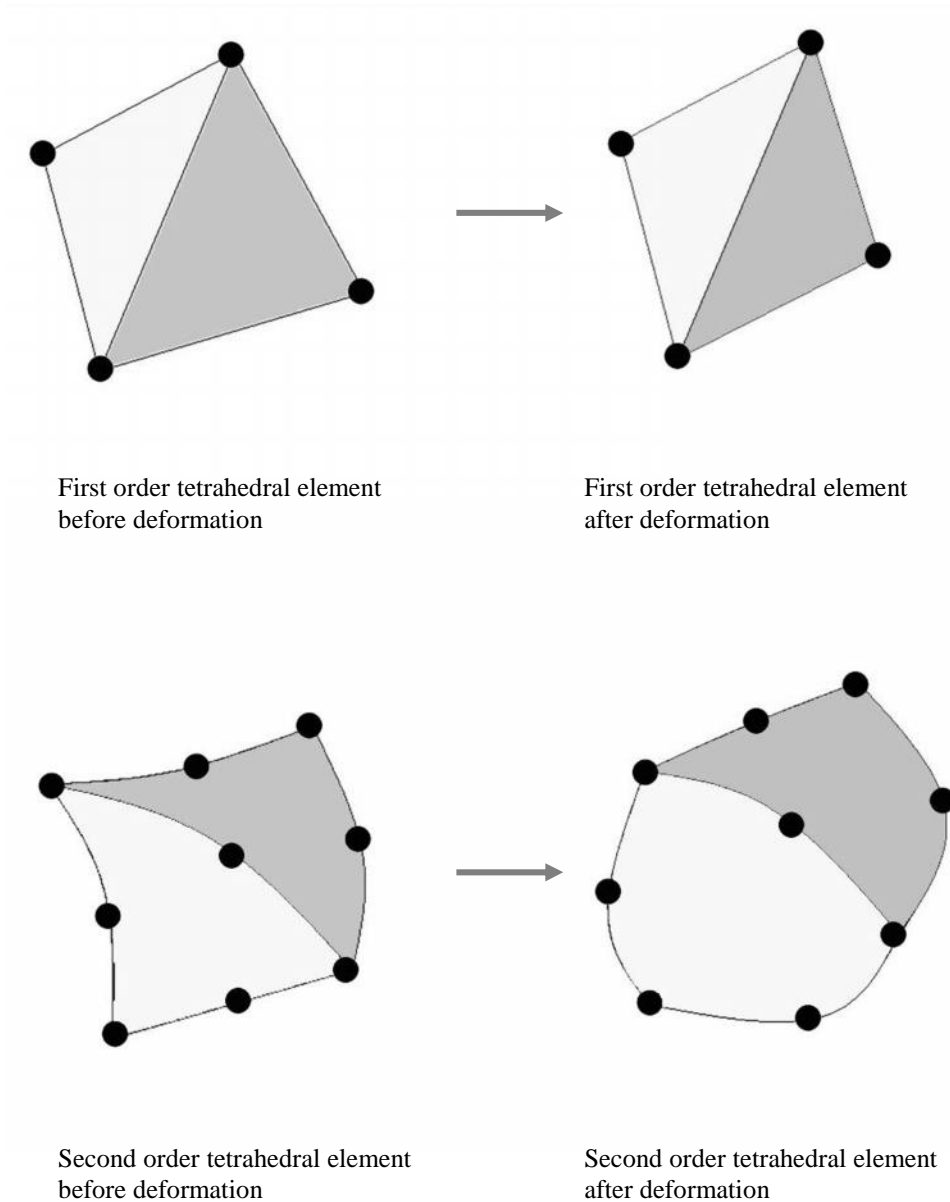


Figure 1-4: Differences between first and second order tetrahedral elements.

First and second order tetrahedral elements are shown before and after deformation. Notice that the first order element has corner nodes only, while the second order element has both corner and mid-side nodes (one mid-side node is not visible for the second order element in this illustration). Single elements seldom experience deformations of this magnitude, which are exaggerated in this illustration.

In a first order element, edges are straight and faces are flat. After deformation the edges and faces must retain these properties.

The edges of a second order element before deformation may either be straight or curvilinear, depending on how the element has been mapped to model the actual geometry. Consequently, the faces of a second order element before deformation may be flat or curved.

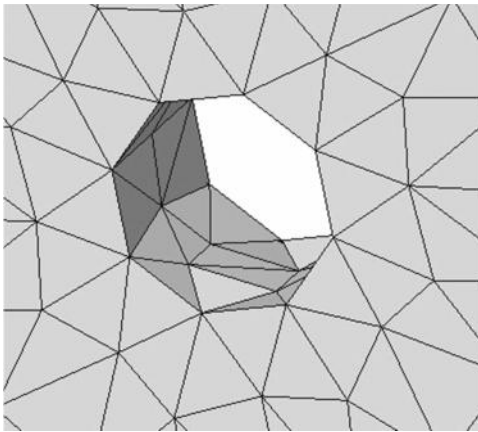
After deformation, edges of a second order element may either assume a different curvilinear shape or acquire a curvilinear shape if they were initially straight. Consequently, faces of a second order element after deformation can be either flat or curved.

First order tetrahedral elements model the linear field of displacement inside their volume, on faces, and along edges. The linear (or first order) displacement field gives these elements their name: first order elements.

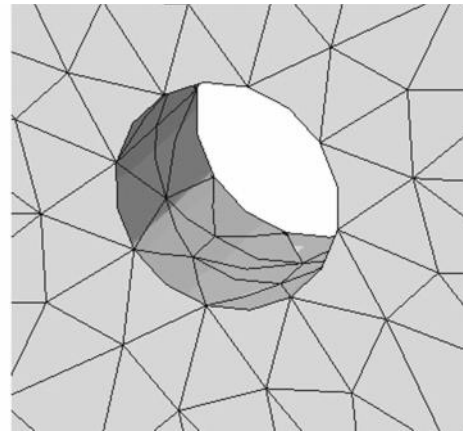
If you recall from Mechanics of Materials, strain is the first derivative of displacement. Since the displacement field is linear, the strain field is constant. Consequently the stress field is also constant in first order tetrahedral elements. This situation imposes a very severe limitation on the capability of a mesh constructed with first order elements to model the stress distribution of any complex model. To make matters worse, straight edges and flat faces cannot map properly to curvilinear geometry, as illustrated in Figure 1-5, left.

Second order tetrahedral elements have ten nodes (Figure 1-4) and model the second order (parabolic) displacement field and first order (linear) stress field in their volume, on faces and along edges. The edges and faces of second order tetrahedral elements can be curvilinear before and after deformation, therefore these elements can be mapped precisely to curved surfaces, as illustrated in Figure 1-5 right. Even though these elements are more computationally demanding than first order elements, second order tetrahedral elements are used for the majority of analyses with **SOLIDWORKS Simulation** because of their better mapping and stress modeling capabilities.

A tetrahedral solid element is the only type of solid elements available in **SOLIDWORKS Simulation**.



Mapping a hole with first order solid tetrahedral elements.



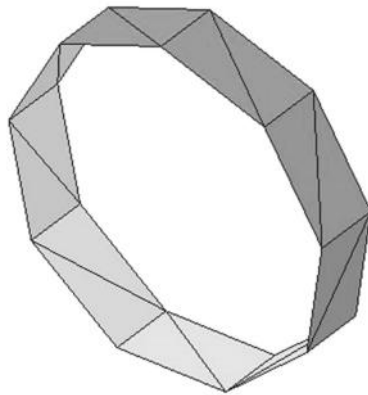
Mapping a hole with second order solid tetrahedral elements.

Figure 1-5: Failure of straight edges and flat faces to map to curvilinear geometry when using first order elements (left), and precise mapping to curvilinear geometry using second order elements (right).

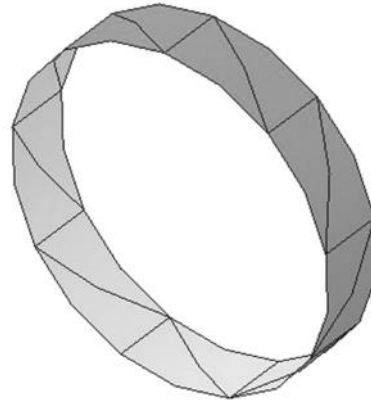
Notice the imprecise first order element mapping of the hole; flat faces approximate the face of the curvilinear geometry. Second order elements map well to curvilinear geometry.

Shell elements

Shell elements are created by meshing surfaces or faces of solid geometry. Shell elements are primarily used for analyzing thin-walled structures. Since surface geometry does not carry information about thickness, the user must provide this information. Similar to solid elements, shell elements also come in draft and high quality with analogous consequences with respect to their ability to map to curvilinear geometry, as shown in Figure 1-6.



Mapping a cylinder with first order triangular shell elements



Mapping a cylinder with second order triangular shell elements

Figure 1-6: Mapping with first order shell elements (left) and second order shell elements (right).

The shell element mesh on the left was created with first order elements. Notice the imprecise mapping of the mesh to curvilinear geometry. The shell element mesh on the right was created with second order elements, which map correctly to curvilinear geometry.

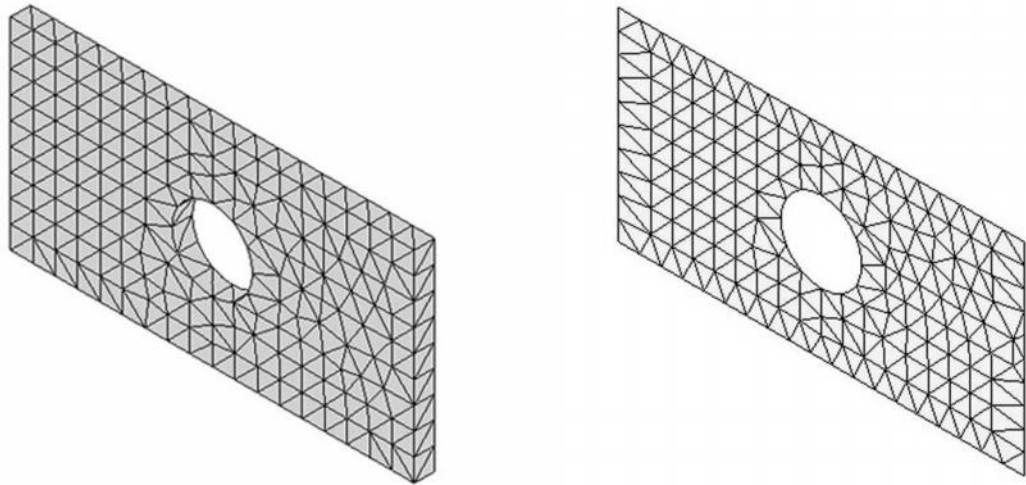
We need to make two important comments about Figure 1-5 and Figure 1-6. First, a mesh should never be that coarse (large size of elements compared to the model). We use a coarse mesh only to show the differences between first and second order elements clearly. Second, notice the “kinks” on the side of the second order elements; they indicate locations of mid side nodes. The second order element does map precisely to second order geometry.

As in the case of solid elements, first order shell elements model linear displacements and constant strain and stress. Second order shell elements model second order (parabolic) displacement and linear strain and stress.

The assumptions of modeling first or second order displacements in shell elements apply only to in-plane directions. The distribution of in-plane stresses across the thickness is assumed to be linear in both first and second order shell elements.

Triangular elements are the only type of shell elements available in **SOLIDWORKS Simulation**.

Certain classes of shapes can be modeled using either solid or shell elements, such as the plate shown in Figure 1-7. Often the nature of the geometry dictates what type of element should be used for meshing. For example, a part produced by casting would be meshed with solid elements, while a sheet metal structure would be best meshed with shell elements.



Second order tetrahedral solid elements

Second order triangular shell elements

Figure 1-7: Plate modeled with solid elements (left) and shell elements (right).

The actual choice between solids and shells depends on the particular requirements of analysis and sometimes on personal preferences.

Beam elements

Beam elements are created by meshing curves (wire frame geometry). They are a natural choice for meshing weldments. Assumptions about the stress distribution in two directions of the beam cross section are made.

A beam element does not have any physical dimensions in the directions normal to its length. It is possible to think of a beam element as a line with assigned beam cross section properties (Figure 1-8).

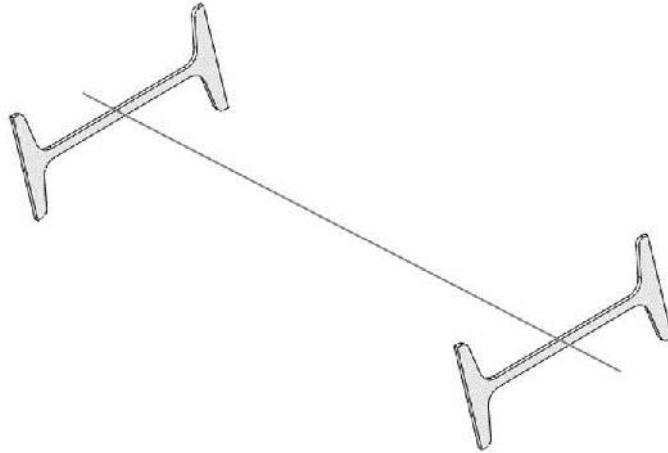


Figure 1-8: Conceptual representation of a beam element.

A beam element is a line with assigned properties of a beam cross section as required by beam theory. This illustration conceptualizes how a curve (here a straight line) defines an I-beam but does not represent actual geometry of beam cross-section.

Before we proceed with the classification of finite elements we need to introduce the concept of nodal degrees of freedom which are of paramount importance in FEA. The degrees of freedom (DOF) of a node in a finite element mesh define the ability of the node to perform translation and rotation. The number of degrees of freedom that a node possesses depends on the element type. In **SOLIDWORKS Simulation**, nodes of solid elements have three degrees of freedom, while nodes of shell elements have six degrees of freedom.

This is because in order to describe the transformation of a solid element from the original to the deformed shape, we only need to know three translational components of nodal displacement. In the case of shell and beam elements, we need to know the translational components of nodal displacements and the rotational displacement components.

Using solid elements we study how a 3D structure deforms under a load. Using shell elements we study how a 3D structure with one dimension “collapsed” deforms under a load. This collapsed dimension is thickness which is not represented explicitly in the

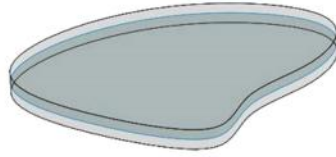
model geometry. Beam elements are intended to study 3D structures with two dimensions removed from the geometry and not represented explicitly by model geometry. It is important to point out that solids, shells and beams are all 3D elements capable of deformation in 3D space.

2D elements

There are also cases where a structure's response to load can be fully described by 2D elements that have only two in-plane degrees of freedom. These are plane stress, plane strain and axi-symmetric elements.

Plane stress elements are intended for the analysis of thin planar structures loaded in-plane, where 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 out-of-plane strain is assumed to be equal zero. Axi-symmetric elements are intended for the analysis of axi-symmetric structures under axi-symmetric load. In all of these special cases, the structure deformation can be fully described using elements with only two degrees of freedom per node. For plane stress and plane strain, these are two components of in-plane translation. For axi-symmetric elements these are radial and axial displacements.

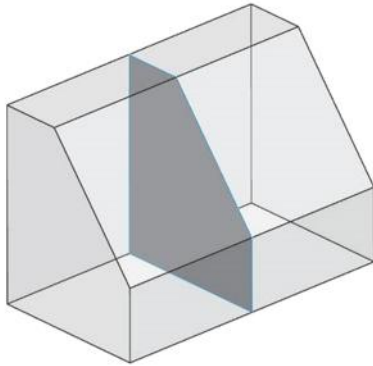
2D elements are summarized in Figure 1-9.



Thin 3D model



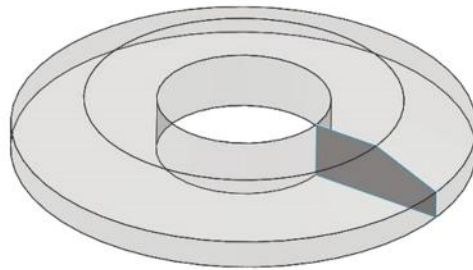
Representative slice ready for meshing with 2D plane stress elements



Thick 3D model



Representative slice ready for meshing with 2D plane strain elements



Axi-symmetric 3D model



Representative slice ready for meshing with 2D axi-symmetric elements

Figure 1-9: Application of 2D elements: plane stress (top), plane strain (middle), and axi-symmetric (bottom).

Just like solids and shells, 2D elements may be of first or second order.

Figure 1-10 presents the basic library of elements in **SOLIDWORKS Simulation**. Solid elements are tetrahedral, shell elements and 2D elements are triangles and beam elements are lines. Elements such as hexahedral solids or quadrilateral shells are not available in **SOLIDWORKS Simulation**.

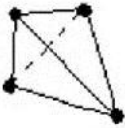
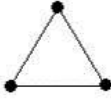
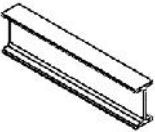
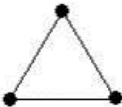
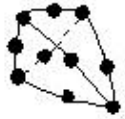
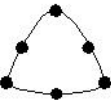
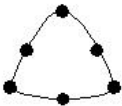
	3D elements			2D elements
	Solid elements	Shell elements	Beam elements	Plate elements
First order element Linear (first order) displacement field Constant stress field				
Second order element Parabolic (second order) displacement field Linear stress field				

Figure 1-10: Basic element library of SOLIDWORKS Simulation.

The majority of analyses use the second order tetrahedral element. Element order is not applicable to a beam element. 2D elements are sometimes referred to as plate elements.

Beam element cross-section is defined by cross section and second moments of inertia of the analyzed structural member. It does not have any physical shape. The above I beam is shown only for the ease of visualization.

What is calculated in FEA?

Each degree of freedom of a node in a finite element mesh constitutes an unknown. In structural analysis, nodal degrees of freedom represent displacement components, while in thermal analysis they represent temperatures. Nodal displacements and nodal temperatures are the primary unknowns for structural analysis and thermal analysis, respectively.

Structural analysis finds displacements, strains and stresses. If solid elements are used, then three displacement components (three translations) per node must be calculated. With shell and beam elements, six displacement components (six translations) must be calculated. 2D elements require calculations of two displacement components. Strains and stresses, are calculated based on the nodal displacement results.

Thermal analysis finds temperatures, temperature gradients, and heat flow. Since temperature is a scalar value (unlike displacements, which are vectors), then regardless of what type of element is used, there is only one unknown (temperature) to be found for each node. All other thermal results such as temperature gradient and heat flux are calculated based on temperature results. The fact that there is only one unknown to be found for each node, rather than three or six, makes thermal analysis less computationally intensive than structural analysis.

How to interpret FEA results

Results of structural FEA are provided in the form of displacements and stresses. But how do we decide if a design “passes” or “fails”? What constitutes a failure?

To answer these questions, we need to establish some criteria to interpret FEA results, which may include maximum acceptable displacements, maximum stress, or the lowest acceptable natural frequency.

While displacement and frequency criteria are quite obvious and easy to establish, stress criteria are not. Let us assume that we need to conduct a stress analysis in order to ensure that stresses are within an acceptable range. To judge stress results, we need to understand the mechanism of potential failure. If a part breaks, what stress measure best describes that failure? **SOLIDWORKS Simulation** can present stress results in any desired form, but it is up to us to decide which stress measures should be used to analyze results.

Discussion of various failure criteria is out of the scope of this book. Any textbook on the Mechanics of Materials provides information on this topic. Here we will limit our discussion to commonly used failure criteria: Von Mises Stress failure criterion and Maximum Normal Stress failure criterion.

Von Mises Stress failure criterion

Von Mises stress, also known as Huber stress, is a stress measure that accounts for all six stress components of a general 3-D state of stress (Figure 1-11).

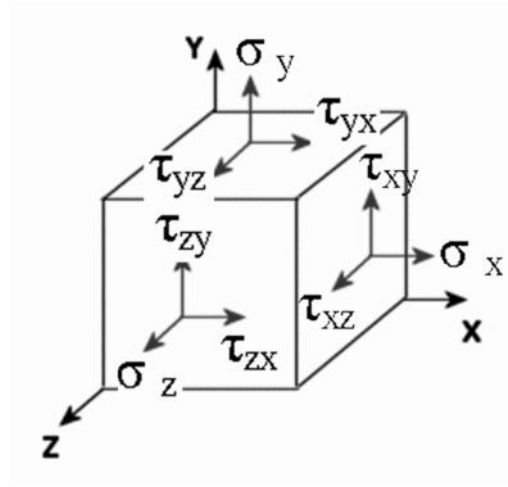


Figure 1-11: General state of stress represented by three normal stresses: σ_x , σ_y , σ_z and six shear stresses.

Two components of shear stress and one component of normal stress act on each side of this elementary cube. Due to symmetry of shear stresses, the general 3D state of stress is characterized by six stress components: σ_x , σ_y , σ_z and $\tau_{xy} = \tau_{yx}$, $\tau_{yz} = \tau_{zy}$, $\tau_{xz} = \tau_{zx}$

Von Mises stress σ_{vm} can be expressed either by six stress components as:

$$\sigma_{vm} = \sqrt{0.5 \times [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3 \times (\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

Von Mises stress σ_{vm} can be also expressed three principal stresses (Figure 1-12) as:

$$\sigma_{vm} = \sqrt{0.5 \times [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}$$

Notice that von Mises stress is a non-negative, scalar stress measure. Von Mises stress is commonly used to present results because the structural safety for many engineering materials showing elasto-plastic properties (for example, steel or aluminum alloy) can be evaluated using von Mises stress.

The maximum von Mises stress failure criterion is based on the von Mises-Hencky theory, also known as the shear-energy theory or the maximum distortion energy theory. The theory states that a ductile material starts to yield at a location when the von Mises stress becomes equal to the stress limit. In most cases, the yield strength is used as the stress limit. According to the von Mises failure criterion, the factor of safety (FOS) is expressed as:

$$FOS = \sigma_{limit} / \sigma_{vm}$$

where σ_{limit} is yield strength.

Maximum Normal Stress failure criterion

By properly adjusting the angular orientation of the stress cube in Figure 1-11, shear stresses disappear and the state of stress is represented only by three principal stresses: σ_1 , σ_2 , σ_3 , as shown in Figure 1-12. In **SOLIDWORKS Simulation**, principal stresses are denoted as P1, P2, P3.

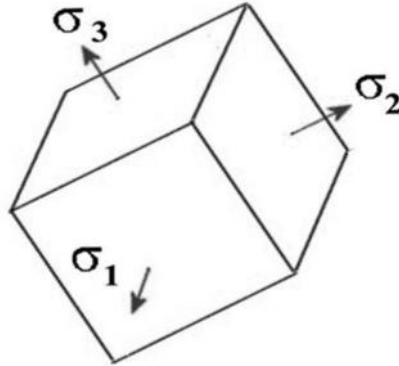


Figure 1-12: The general state of stress represented by three principal stresses: σ_1 , σ_2 , σ_3 .

The Maximum Normal Stress Failure criterion is used for brittle materials. Brittle materials do not have a specific yield point. This criterion assumes that the ultimate tensile strength of the material in tension and compression is the same. This assumption is not valid in all cases. For example, cracks considerably decrease the strength of the material in tension while their effect is not significant in compression because the cracks tend to close.

This criterion predicts failure will occur when σ_1 exceeds the stress limit, usually the ultimate tensile strength. According to the maximum principle stress failure criterion, the factor of safety FOS is expressed as:

$$FOS = \sigma_{limit} / \sigma_1$$

where σ_{limit} is ultimate tensile strength.

Units of measure

Internally, **SOLIDWORKS Simulation** uses the International System of Units (SI). However, for the user's convenience, the unit manager allows data entry in any of three systems of units: SI, Metric, and English. Results can be displayed using any of the three systems. Figure 1-13 summarizes the available systems of units.

	International System (SI)	Metric (MKS)	English (IPS)
Mass	kg	kg	lb
Length	m	cm	in
Time	s	s	s
Force	N	Kgf	lbf
Gravitational acceleration	m/s^2	G	in/s^2
Mass density	kg/m^3	kg/cm^3	lbf/in^3
Temperature	K	°C	°F

Figure 1-13: Unit systems available in SOLIDWORKS Simulation.

SI, Metric, and English systems of units can be interchanged when entering data and analyzing results in SOLIDWORKS Simulation.

As **SOLIDWORKS Simulation** users, we are spared much confusion and trouble with systems of units. However, we may be asked to prepare data or interpret the results of other FEA software where we do not have the convenience of the unit manager. Therefore, we will make some general comments about the use of different systems of units in the preparation of input data for FEA models. We can use any consistent system of units for FEA models, but in practice, the choice of the system of units is dictated by what units are used in the CAD model. The system of units in CAD models is not always consistent; length can be expressed in $[mm]$, while mass density can be expressed in $[kg/m^3]$. Contrary to CAD models, in FEA all units must be consistent. Inconsistencies are easy to overlook, especially when defining mass and mass density and can lead to serious errors.

In the SI system, based on meters [*m*] for length, kilograms [*kg*] for mass, and seconds [*s*] for time, all other units are easily derived from these base units. In mechanical engineering, length is commonly expressed in millimeters [*mm*], force in Newtons [*N*], and time in seconds [*s*]. All other units must then be derived from these basic units: [*mm*], [*N*], and [*s*]. Consequently, the unit of mass is defined as a mass which, when subjected to a unit force equal to 1N, will accelerate with a unit acceleration of 1 mm/s². Therefore, the unit of mass in a system using [*mm*] for length and [*N*] for force, is equivalent to 1000 kg or one metric ton. Therefore, mass density is expressed in metric tonnes [*tonne/mm³*]. This is critically important to remember when defining material properties in FEA software without a unit manager. Review Figure 1-14 and notice that an erroneous definition of mass density in [*kg/m³*] rather than in [*tonne/mm³*] results in mass density being one trillion (10¹²) times higher.

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 x 10 ⁻⁹ tonne/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 x 10 ⁻⁴ slug/12/in ³

Figure 1-14: 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.

Using online help

SOLIDWORKS Simulation features very extensive online Help and Tutorial functions, which can be accessed from the Help menu in the main **SOLIDWORKS** tool bar or from the **Simulation** menu. The Study advisor can be accessed from the Study drop down menu (Figure 1-15).

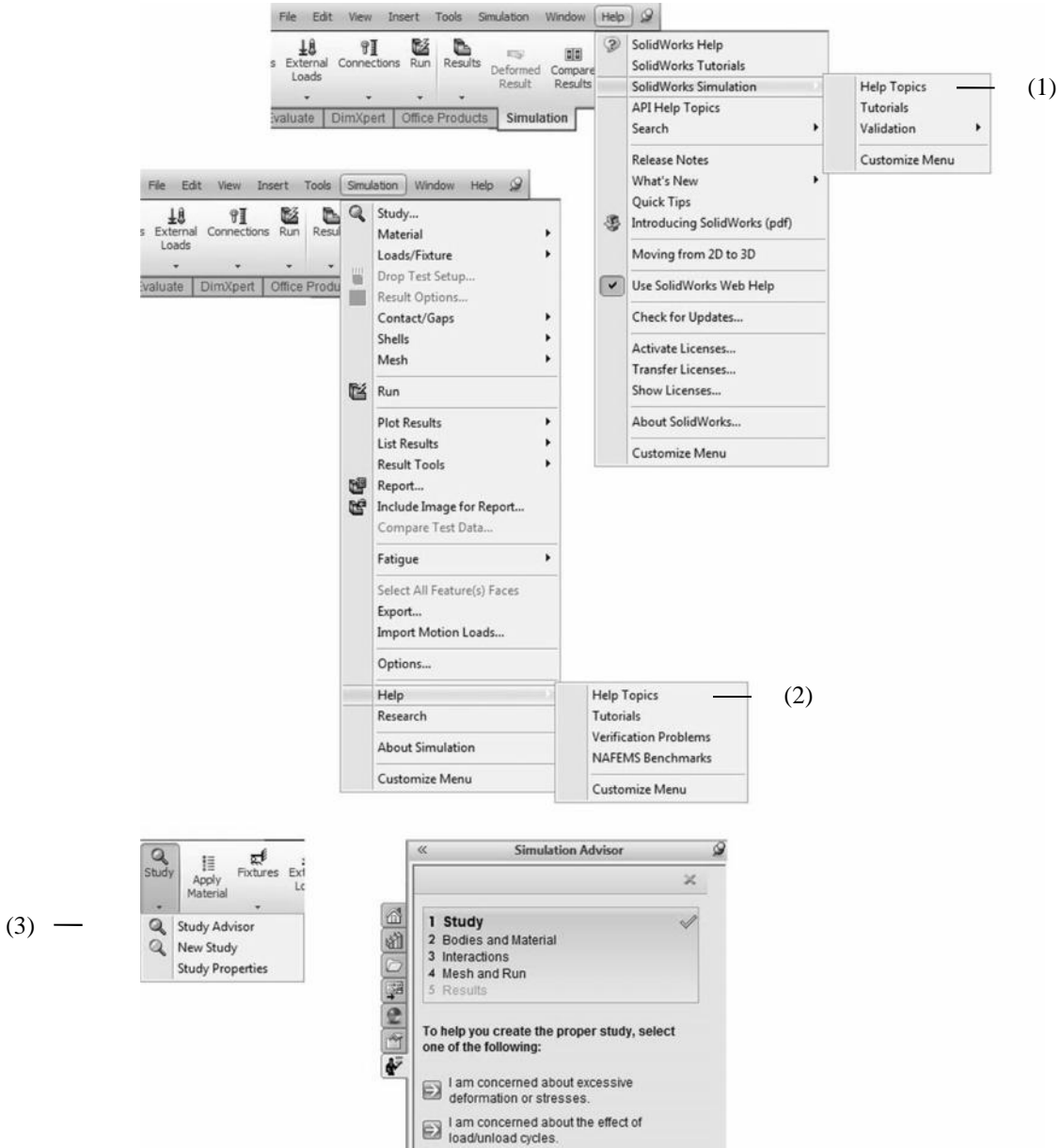


Figure 1-15: Accessing online Help, and Study Advisor.

Online Help, Tutorials, and Validation can be accessed from the main SOLIDWORKS toolbar (1) or from the Simulation menu (2). The Study Advisor can be accessed from the Study drop down menu (3).

Limitations of Static studies

Static study is the only type of study available in some **SOLIDWORKS** packages. Working with **Static** study we need to accept important limitations: material is assumed as linear, and loads are static.

Linear material

Whatever material we assign to the analyzed parts or assemblies, the material is assumed to be linear, meaning that stress is proportional to the strain (Figure 1-16).

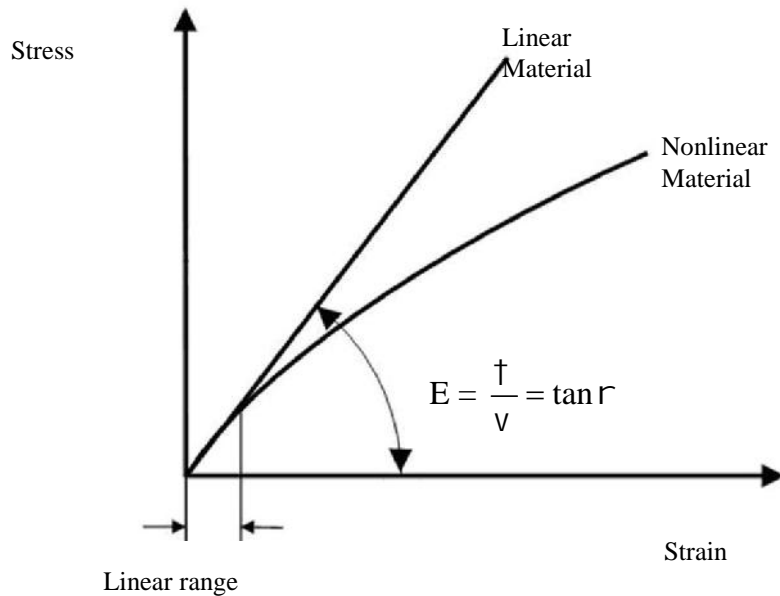


Figure 1-16: The linear material model assumed in SOLIDWORKS Simulation.

With a linear material, stress is linearly proportional to strain. The linear range is where the linear and nonlinear material models are not significantly different.

Using a linear material model, the maximum stress magnitude is not limited to yield or to ultimate stress as it is in reality. Material yielding is not modeled, and whether or not yield may in fact be taking place can only be established based on the stress magnitudes reported in results. Most analyzed structures experience stresses below the yield stress, and the factor of safety is most often related to the yield stress. Therefore, the limitations imposed by linear material seldom impede **SOLIDWORKS Simulation** Professional users.

Static loads

All structural loads and restraints are assumed not to change with time. Dynamic loading conditions cannot be analyzed with **Static** study. This limitation implies that loads are applied slowly enough to ignore inertial effects.

Nonlinear material analysis and dynamic analysis can be performed with **SOLIDWORKS Simulation Premium**.

2: Static analysis of a plate

Topics covered

- ❑ Using the **SOLIDWORKS Simulation** interface
- ❑ Linear static analysis with solid elements
- ❑ Controlling discretization error with the convergence process
- ❑ Finding reaction forces
- ❑ Presenting FEA results in a desired format

Project description

A steel plate is supported and loaded, as shown in Figure 2-1. We assume that the support is rigid (this is also called built-in support, fixed support or fixed restraint) and that a 100000N tensile load is uniformly distributed along the end face, opposite to the supported face.

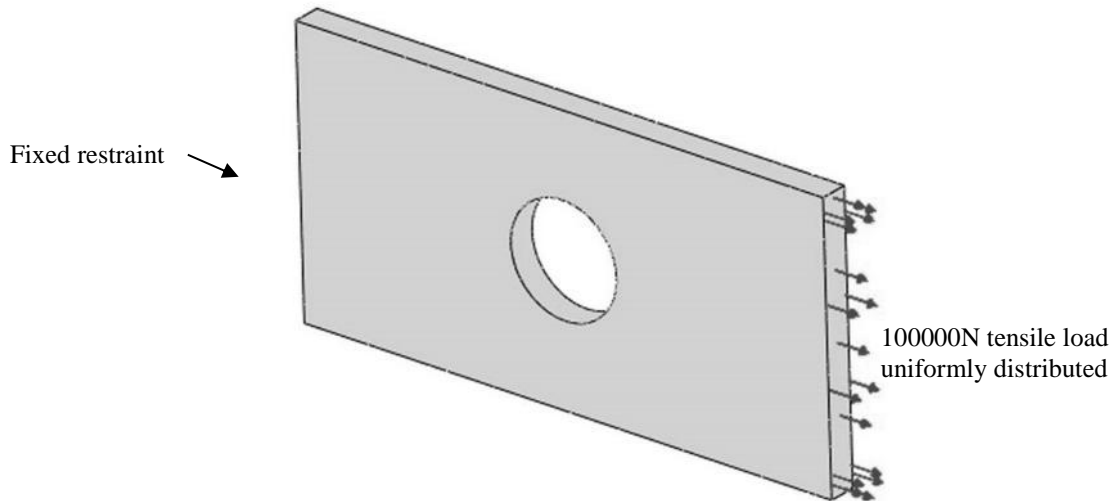


Figure 2-1: SOLIDWORKS model of a rectangular plate with a hole.

We will perform a displacement and stress analysis using meshes with different element sizes. Notice that repetitive analysis with different meshes does not represent standard practice in FEA. However, repetitive analysis with different meshes produces results which are useful in gaining more insight into how FEA works.