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An Overview of Computer Aided Design and Finite Element Analysis

by

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An Overview of Computer Aided Design and Finite Element Analysis

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1.0 Course Overview

NOTE: This is a course in Computer Aided Design (CAD) for professional engineers, not a course in the related subject of Computer Aided Drafting.

The course will introduce the concepts of computer aided design (CAD) and finite element analysis (FEA). Practical applications will be discussed to show how to use CAD to improve the design process and how to use FEA to better predict design function.

Illustrations throughout this course were developed using two of the most common solid modeling software packages: Solidworks and Autodesk Inventor. I am not trying to indicate that those are superior programs, and I am not in any way trying to promote those two packages. Those two programs were selected simply based on their popularity in industry. The majority of the concepts presented in this course can be used in any solid modeling software.

Concepts of FEA are discussed with the intent to develop an understanding of how to use FEA for different applications. This course will not provide a step-by-step tutorial on using any particular FEA software package. Specific tools or functions are discussed, along with proper applications, to further your understanding of utilizing FEA to its full potential.

2.0 General Concepts

2.1 What is Computer Aided Design

The technology for creating engineering drawings of parts and assemblies keeps advancing. Days of using slide rules and manual drafting techniques, as illustrated in Figure 1, are a thing of the distant past. These manual methods for design were slow and tedious, which made iterative design procedures very difficult and expensive. Most engineers of today could not imagine trying to design anything too complex working within the limitations of manual drafting and hand calculations.

Computer programs of many different types are powerful tools commonly used by engineers to become more efficient and reduce errors. **Computer aided design** (CAD), in the most basic definition, is the use of computer software to assist the design engineer in the overall creation and analysis of parts and assemblies. It should be noted that the acronym CAD also stands for



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Computer Aided Drafting, but for this course we will focus on aspects of computer aided design with only some general discussion of computer aided drafting.



Figure 1 Manual drafting and design procedures

Technically, computer aided design could refer to the use of any computer program to aid in design. MATLAB, for example, is a very powerful engineering design tool that is frequently used by engineers. However, for this course we will be focusing mainly on the use of solid modeling software and its use in computer aided design and finite element analysis. Section 4.4 will briefly discuss the use of EXCEL but focusing only on its use in conjunction with solid modeling software.

2.1.1 2D verses 3D Technology

Many engineers of today only learned how to use two-dimensional (2D) software such as AutoCAD (first released in 1982). Though 2D software is still utilized and is great for general



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drafting, it is very limited on applications of true computer aided design and they have limited 3D capabilities. Three-dimensional (3D) solid modeling software allows for a more complete analysis and greater flexibility. Common 3D solid modeling software includes Solidworks, CREO, and Autodesk Inventor. This course will provide screenshot examples from both Solidworks and Autodesk Inventor because they are two very popular programs.

One advantage that 3D software has over 2D software is that it is generally easier to visualize complex parts in 3D. As an example, a part is shown in Figure 2 (a) represented as the standard two-dimensional orthographic views (front, top, and right-side views). The same part is shown in Figure 2 (b) represented in a simple three-dimensional isometric view. Though this part is still fairly basic, it can be much easier to visualize the part when it is represented as a three-dimensional isometric view. Solid modeling software has the added benefit of being able to rotate the part in 3D space to get a more complete visualization.

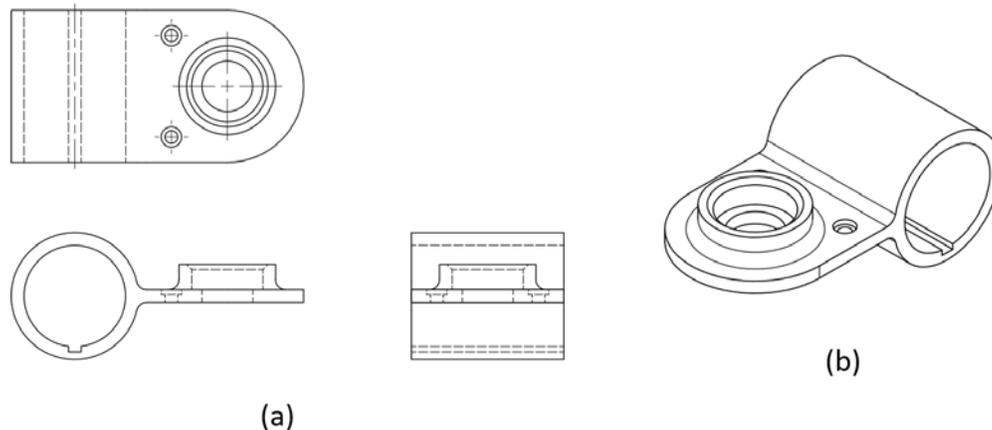


Figure 2 Comparison of (a) 2D views of a part and (b) 3D isometric representation of a part

Three-dimensional software allows the user to combine individual part files into an assembly. Assembly files, such as the example provided in Figure 3, are useful to see how parts will work together and will check for interferences between parts. Assemblies can be done in 2D software, but interferences are much easier to detect in 3D software.



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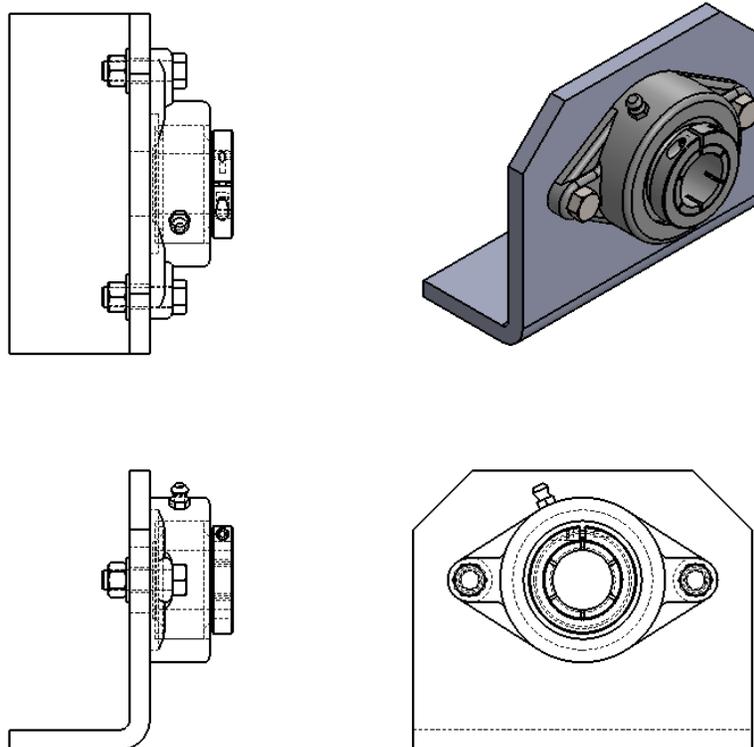


Figure 3 Example of an assembly [generated in Solidworks]

Three-dimensional software has added benefits that help automate development of drawing sets. Some examples include automation of calling out hole size and features, bolt information, and bill of material tables. The assembly shown in Figure 3, for example, could automatically generate a bill of material and know how many bearings, mounting plates, bolts, washers, and nuts are in the assembly. Leaders can be developed that automatically tie to the bill of material. Changes, such as changing the bolt diameter, would automatically be reflected in the bill of



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material. These automation features expedite drawing development and reduce errors commonly associated with design changes.

2.1.2 Computer Aided Manufacturing

Though not within the true scope of this course, it is important to briefly describe the benefits of computer aided manufacturing (CAM) and its relation to computer aided design. A true power of three-dimensional solid modeling software is its ability to integrate with computer aided manufacturing tools such as CNC equipment and 3D printers. CAM is also the foundation to the historical development of current CAD systems.

Patrick Hanratty is known as the father of CAD/CAM. In his early career (1950's) he developed the Program for Numerical Tooling Operations (PRONTO) while at GE. PRONTO was the first commercial numerical control system. In the 1970's he started his own software company called Integrated Computer Systems, where he developed a complete CAD/CAM system called INTERAPT (INTERactive Automatically Programed Tools). Another important contributor to CAD was Ivan Sutherland, an electrical engineer that developed sketchpad. The work of Hanratty and Sutherland is the foundation of all CAD systems of today.

Today, three-dimensional CAD systems can directly output to numerous CAM software. It is not uncommon to develop rapid prototypes using 3D printers to determine design functionality. Code can be automatically developed to determine machine paths for CNC equipment for final production. CAD systems are, therefore, a tool used through all stages of design and fabrication.

2.2 What is Finite Element Analysis

Finite element analysis (FEA) is a numerical method used to determine approximate solutions of physical problems. FEA has been around since the 1940's and was originally developed for problems relating to solid mechanics. The use of FEA did not become practical until the advent of high-speed computers. The current applications, however, extend past solid mechanics to areas such as thermal analysis and dynamics.

The method of finite element analysis consists of modeling a part by dividing (discretizing) the part into an equivalent system consisting of many smaller elements. The smaller elements are



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interconnected at nodes. Elements and nodes for a basic bar are shown in Figure 4. The number of equations that must be solved during FEA is related to the number of nodes and elements in the model.

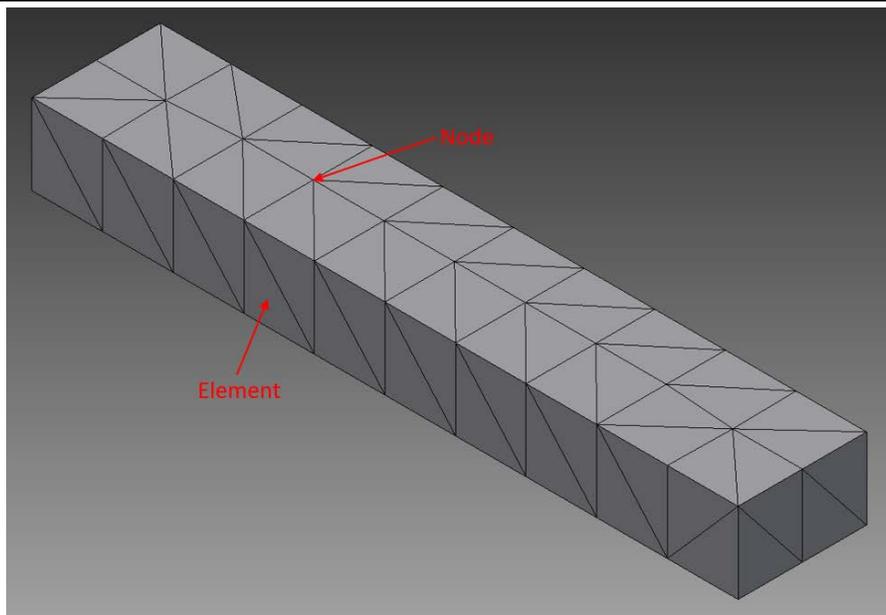


Figure 4 Elements and nodes for a bar

The finite element method of analysis can be used in situations when an analytical solution does not exist. The method is a numerical technique that can be used to approximate an answer and predict behavior of components. Examples include very complex geometry and complex restraints (such as indeterminate structures) or complicated material properties. The method can determine a system's response to combinations of complex loading types (point loads, thermal loads, pressures, etc.) and time dependent loading conditions. FEA can be used to determine mode shapes and natural frequencies and can perform analysis for large deflections.

It is important to validate results obtained from FEA. In some cases, you may need to perform analysis on a problem where test results can be easily obtained. Data obtained from testing can be used to validate FEA models. In other cases, test results may be impossible to obtain or



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simply impractical or expensive. If test results are not possible it becomes very critical to have reliable analysis procedures to validate the model and check convergence.

A major disadvantage of FEA revolves around user mistakes. FEA software will generate results based on user input, and if that input has errors the output can be useless. Simple, but disastrous, errors could include inconsistencies in units. Using the wrong element type (see Section 8.4) or having distorted elements can generate poor results. It is very important that engineers and designers using FEA understand the general theory of FEA to limit errors. The essential concepts of the basic theory of FEA will be discussed in Section 7.0.

2.3 Integration of CAD and FEA

Design is most often an iterative process. Design changes are common as the project team works to optimize the design. Solid modeling software allows for easy integration of computer aided design and finite element analysis. The integration allows engineers to perform FEA during different phases of the project and can greatly expedite optimization.

3.0 Basics of Solid Modeling

3.1 Introduction

As stated in Section 2.1, this course is not a computer aided drafting course. Therefore, the overall intent of this course is not to teach the basic concepts of solid modeling software. However, this section will discuss some of the common features and applications in solid modeling software to help the reader understand the numerous applications of solid modeling software and to better understand the common terminology. If you want additional information on how to generate parts, numerous books are available to teach the fundamentals of different solid modeling software.

3.2 Sketches



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In solid modeling software, a 3D solid volume is created by first creating a 2D sketch. The sketch must be a closed region in order to develop a 3D solid. The process will be illustrated here using Solidworks, though the process is nearly identical in other software packages.

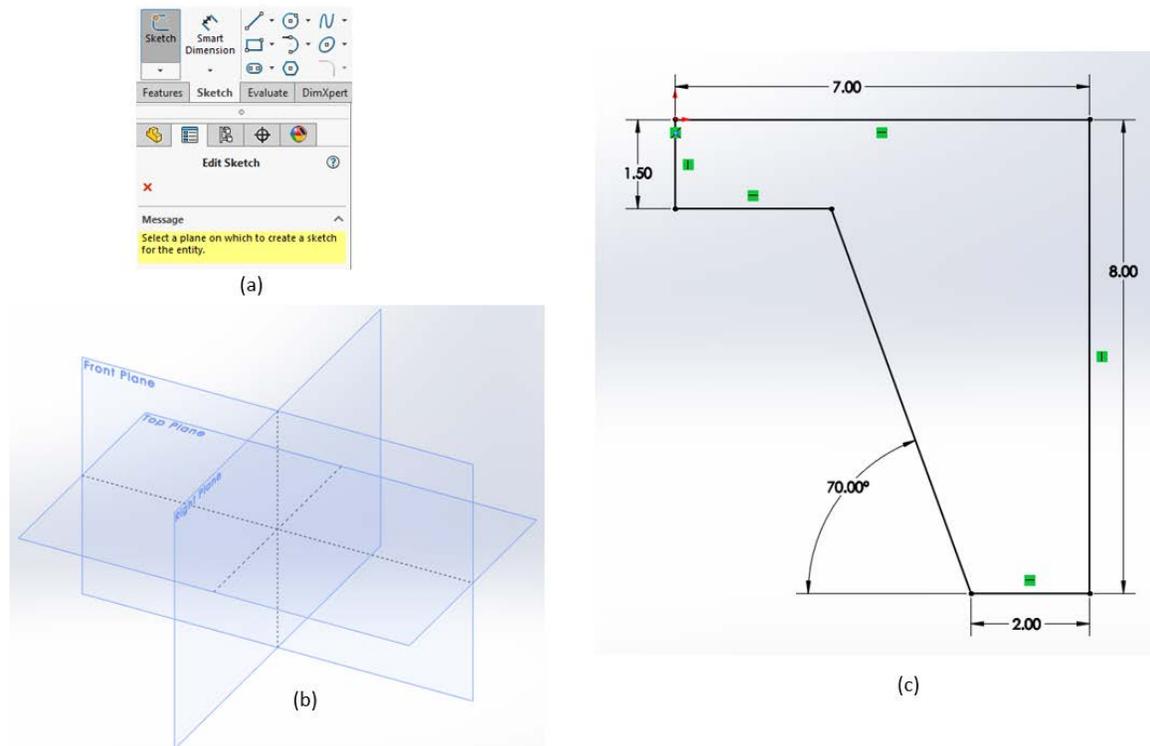


Figure 5 (a) Sketch toolbar (b) Possible sketch planes (c) Sketch developed
[generated in Solidworks]

To start a sketch, the user selects the sketch button as shown in Figure 5 (a). The message will appear asking the user to select a plane for the sketch. For a new part, the user will need to select one of the global planes. The options of front plane, top plane, and right plane are shown in Figure 5 (b). Some programs will use the names of XY plane, XZ plane, and YZ plane. Once a plane is selected the user can draw and dimension a sketch as shown in Figure 5 (c).



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Once the sketch is complete, it can be used to generate a solid feature. Figure 6 shows a solid feature generated from the sketch shown in Figure 5 (c). Any changes made to the sketch will automatically adjust the solid to fit the new sketch.

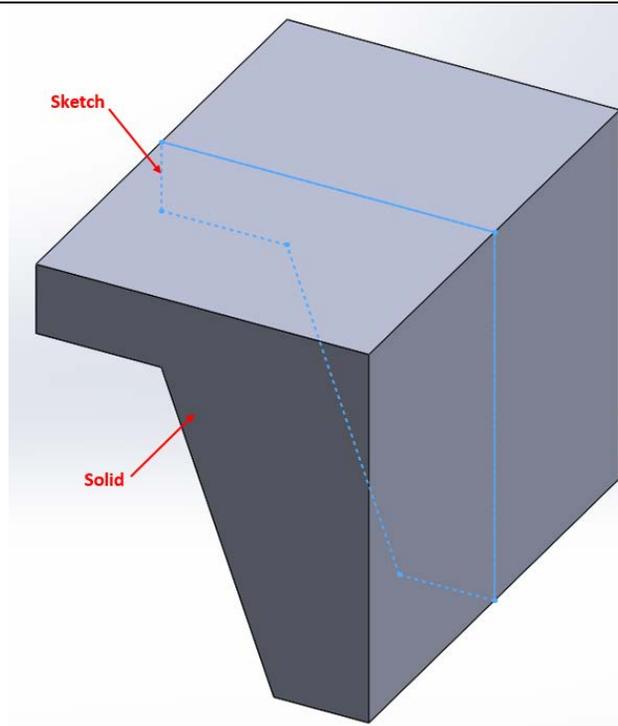


Figure 6 Solid feature generated from a sketch [generated in Solidworks]

Once a solid feature exists, any plane on that solid may be selected for sketch planes. Figure 7 (a) shows the solid shown in Figure 6 with a new sketch developed on the back side of the part. A new solid can be generated from the new sketch as shown in Figure 7 (b).



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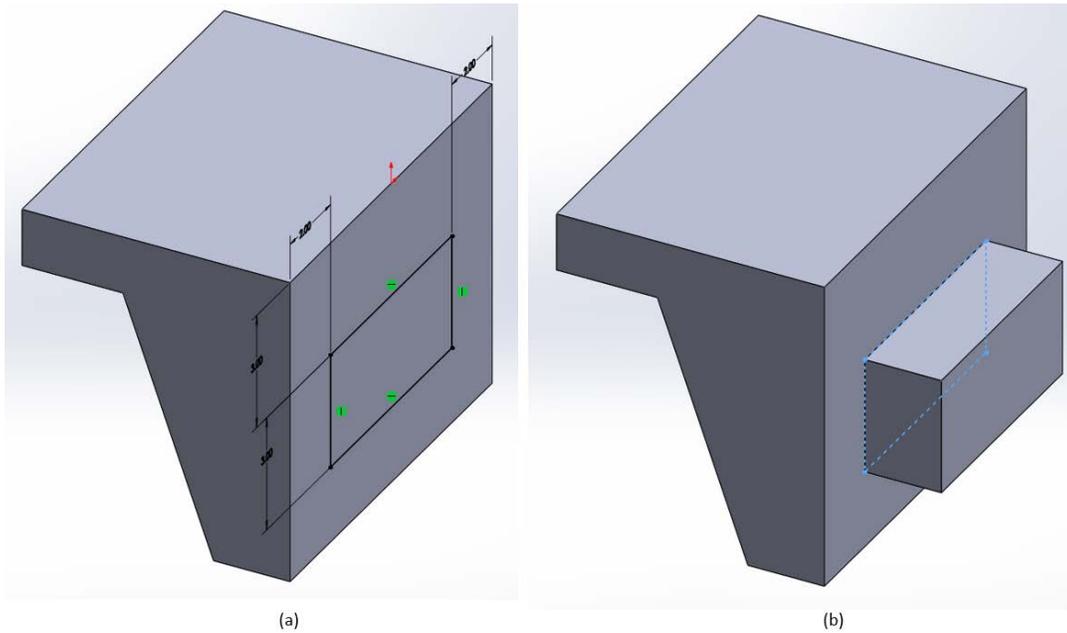


Figure 7 (a) Sketch created on a feature plane (b) Solid feature generated from the new sketch [generated in Solidworks]

3.3 Extrude and Revolve

Once a sketch is defined, the two most common applications to use that sketch to make a solid would be extrude and revolve. Extrude is a linear process, as shown in Figure 8 (a). Revolve is an angular extrusion around an axis, as illustrated in Figure 8 (b). Extrude is also the process that was used to generate the solid shown in Figure 6.



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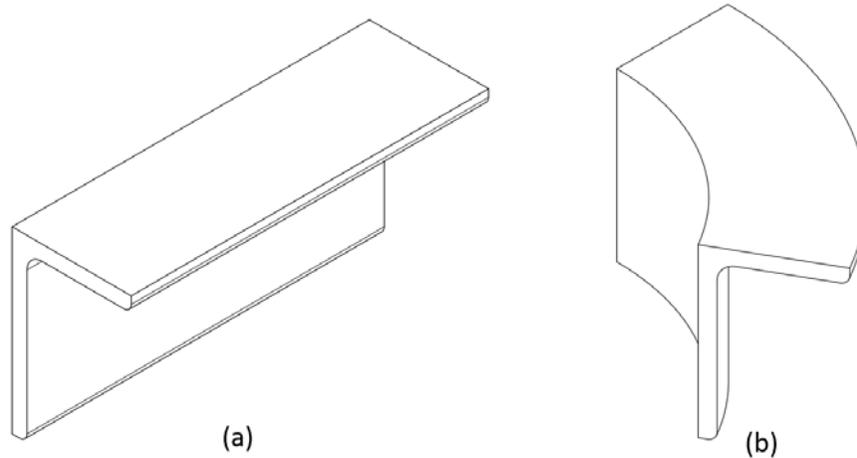


Figure 8 (a) Extrude (b) Revolve

4.0 Parametric Modeling for Computer Aided Design

4.1 Introduction

Now that we have some basic concepts of how solid modeling software can be used to generate simple parts, we can move into topics that deal more with computer aided design. Parametric modeling is an extremely useful feature of solid modeling programs that simplifies design changes. In parametric modeling, a model's geometry is defined by dimensions or parameters. Changing the parameter automatically updates the geometry. For a model that is well thought out with parameters, a simple change of one parameter can change several dimensions. This can be accomplished by defining dimensions in terms of that single parameter with equations.

4.2 Geometric Constraints

Certain aspects of a part's geometry can be constrained in a sketch, such as stating that two lines must remain perpendicular. Figure 9 (a) shows the constraints toolbar from Autodesk Inventor.



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The first three options for constraints are labeled for reference as coincident, collinear, and concentric. The other options (in order) are fixed, parallel, perpendicular, horizontal, vertical, tangent, smooth, symmetric, and equal. Figure 9 (b) is a sketch that illustrates the use of some of these constraints. For example, the length of the upper line and the left side line are constrained to be equal length. The two vertical lines are constrained to be parallel, and so are the two horizontal lines. The upper line is constrained to be horizontal. The upper and right-side line are constrained to be perpendicular. If the user makes changes to the sketch, the set constraints will remain.

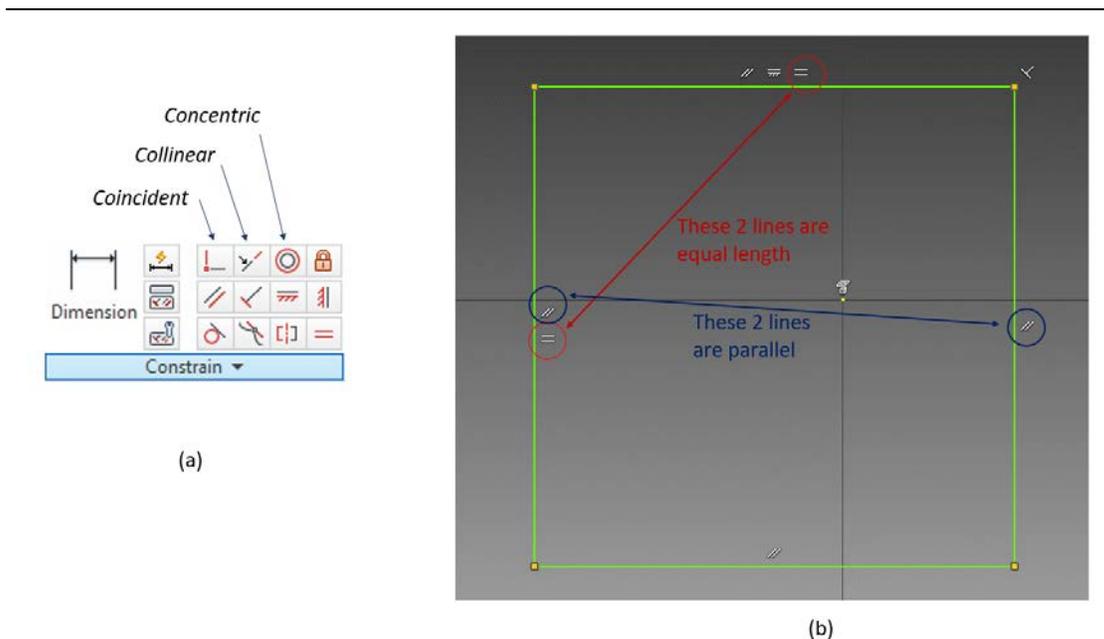


Figure 9 (a) Constraints toolbar (b) Sketch illustrating constraints (created in Autodesk Inventor)

4.3 Equations and Global Variables

Section 3.2 introduced the concept of using sketches, which are used to generate 3D solids. Dimensions are used in sketches to define the size of the solid. Users can input dimensions as



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numbers, and every dimension added is stored in the software as a parameter. These parameters can be used in very complex equations to define other dimensions or parameters.

Global variables can be used when multiple dimensions use a common dimension. Figure 10 shows the dialog box in Solidworks for defining global variables and using equations. In the figure a global variable 'Length1' is defined and given a value of 7. Equations, which are also shown in Figure 10, can use global variables to define dimensions in sketches. The part dimension stored as D1 is set to equal the global variable. So, changing the global variable will automatically change the dimension value. The dimension D2 is set to equal D1, which is the simple use of an equation.

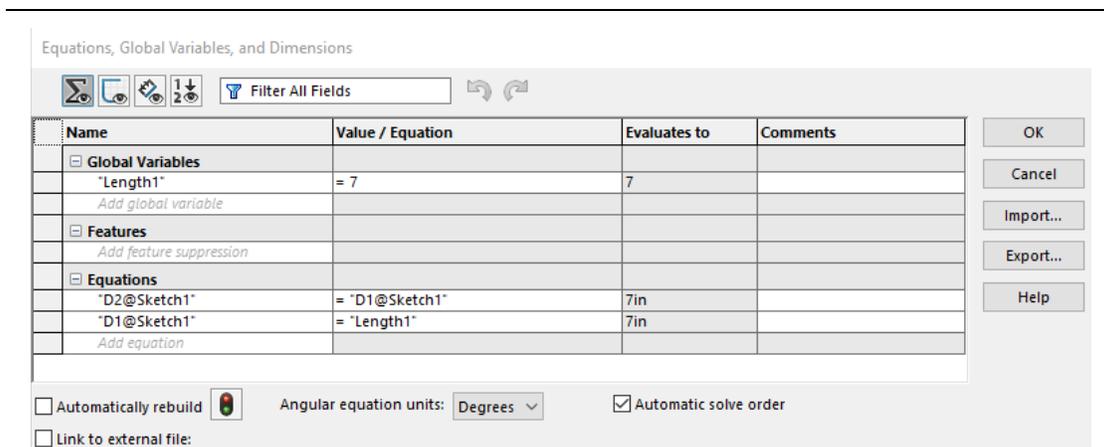


Figure 10 Equations and global variables in Solidworks

4.4 Linking EXCEL to Solid Modeling Software

Another very powerful design aspect of solid modeling software is the ability to link EXCEL to the solid modeling process. EXCEL is a very useful program for performing engineering computations. Data in EXCEL can be linked to solid modeling software to define things such as global variables. Equations that would be too difficult to do directly in the solid modeling software, such as those to set dimensions, can be done in a linked EXCEL file. For example, a



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table can exist in an EXCEL file that controls bend radius of sheet metal based on material and thickness. Dimensions from a solid model can be linked in EXCEL, where a VLOOKUP command can be used to pull data from a table and create a bend radius dimension. That bend radius dimension can then be exported back to the solid modelling software to define that dimension.

5.0 Determining Physical Properties

5.1 Defining Materials

Solid modeling software will allow the user to define the material used for parts. The user can select the material from a built-in library, as shown in Figure 11. Solid modeling software will generally have a very complete list of materials including different forms of steel, aluminum alloys, copper alloys, zinc alloys, plastics, woods, and carbon fibers. The material's properties such as elastic modulus, shear modulus, density, and thermal conductivity are all predefined in the software. In Figure 11 the selection of AISI 1020 steel is made, and the properties are all listed for that material.



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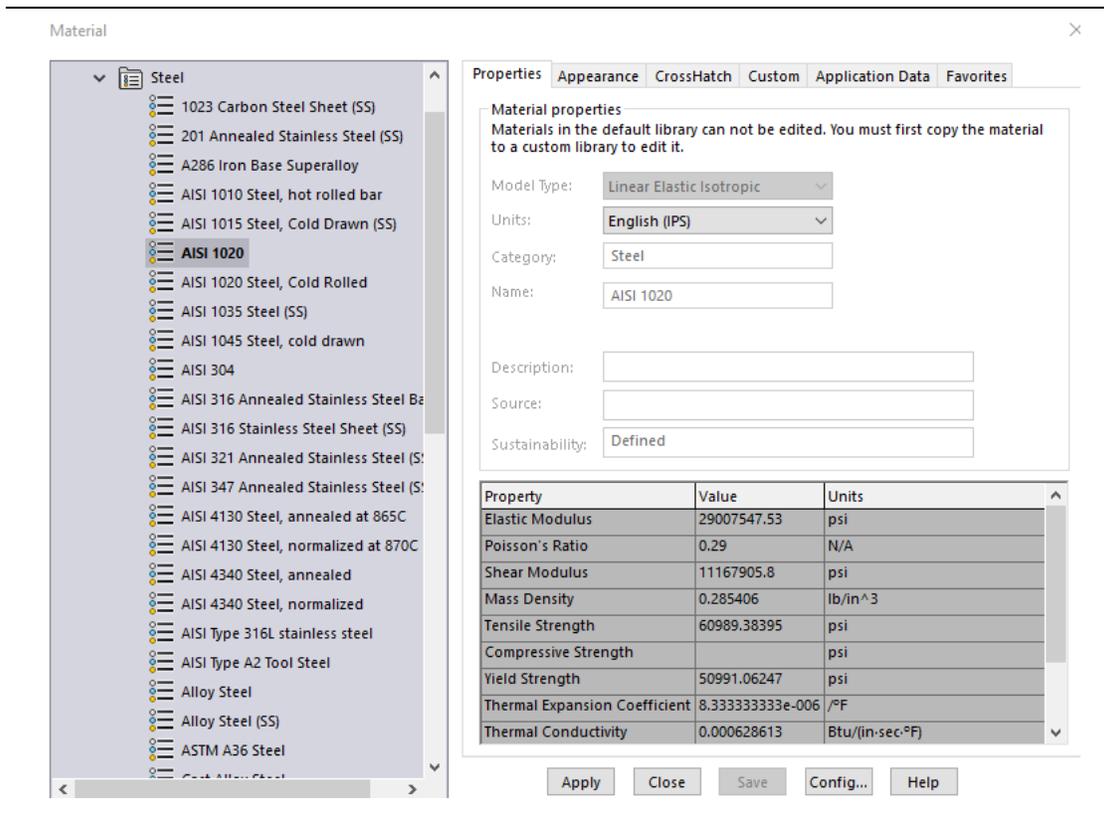


Figure 11 Defining material (generated using Solidworks)

5.2 Mass Properties

Once the part geometry and material properties are defined for a part, solid modeling software can calculate mass properties for the part. Because a part generated in solid modeling is a volumetric shape, the program can easily calculate surface area and volume of the part. The material density is also defined once a material is picked for the part. Therefore, solid modeling programs can give properties for the overall mass of a part.



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Mass properties of Part1
Configuration: Default
Coordinate system: -- default --

Density = 0.29 pounds per cubic inch

Mass = 69.92 pounds

Volume = 245.00 cubic inches

Surface area = 238.00 square inches

Center of mass: ( inches )
X = 1.27
Y = -1.44
Z = 2.50

Principal axes of inertia and principal moments of inertia: ( pounds * square inches )
Taken at the center of mass.
lx = ( 1.00, 0.00, 0.00)   Px = 431.20
ly = ( 0.00, 1.00, 0.00)   Py = 431.20
lz = ( 0.00, 0.00, 1.00)   Pz = 571.05

Moments of inertia: ( pounds * square inches )
Taken at the center of mass and aligned with the output coordinate system.
Lxx = 431.20           Lxy = 0.00           Lxz = 0.00
Lyx = 0.00            Lyy = 431.20        Lyz = 0.00
Lzx = 0.00            Lzy = 0.00         Lzz = 571.05

Moments of inertia: ( pounds * square inches )
Taken at the output coordinate system.
lxx = 1012.90         lxy = -127.42       lxz = 221.47
lyx = -127.42        lyy = 980.46        lyz = -251.45
lzx = 221.47         lzy = -251.45       lzz = 827.95

```

Figure 12 Physical properties of a part (generated using Solidworks)

Figure 12 gives an example of a generic part file. The output gives information about the volume, surface area, and mass of the part. Other properties are also calculated for the part. The coordinate position of the part's center of mass is shown, along with moments of inertia. These properties can be used in design calculations and can generally be exported back into EXCEL for further calculations. The properties are also used in built-in design tools and simulation tools in solid modeling software.



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6.0 Built-in Design Tools

6.1 Introduction

Many solid modeling software packages include some built-in design tools. Figure 13 shows the design tab from Autodesk Inventor. The design tab includes tools for designing bolted connections, frames, shafts, cams, springs, and more.

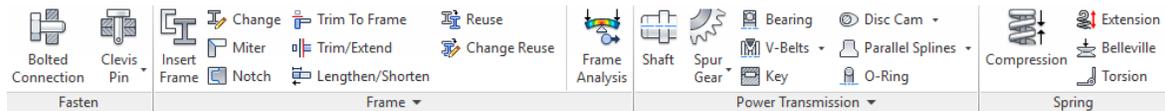


Figure 13 Autodesk Inventor design tab

6.2 Shaft Design Tool

Figure 14 illustrates the use of the shaft design tool in Autodesk Inventor. The design tool allows the user to quickly define the geometry of the different sections of the shaft. Chamfered or rounded edges can be added quickly, and you can also add keyways. The model automatically updates to reflect any changes.



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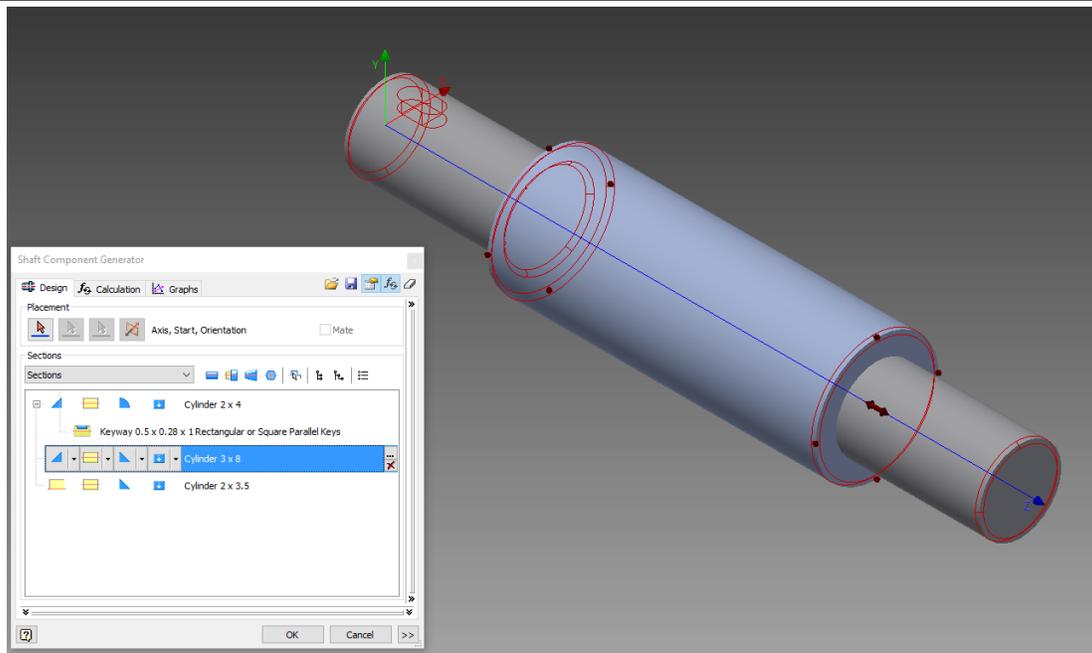
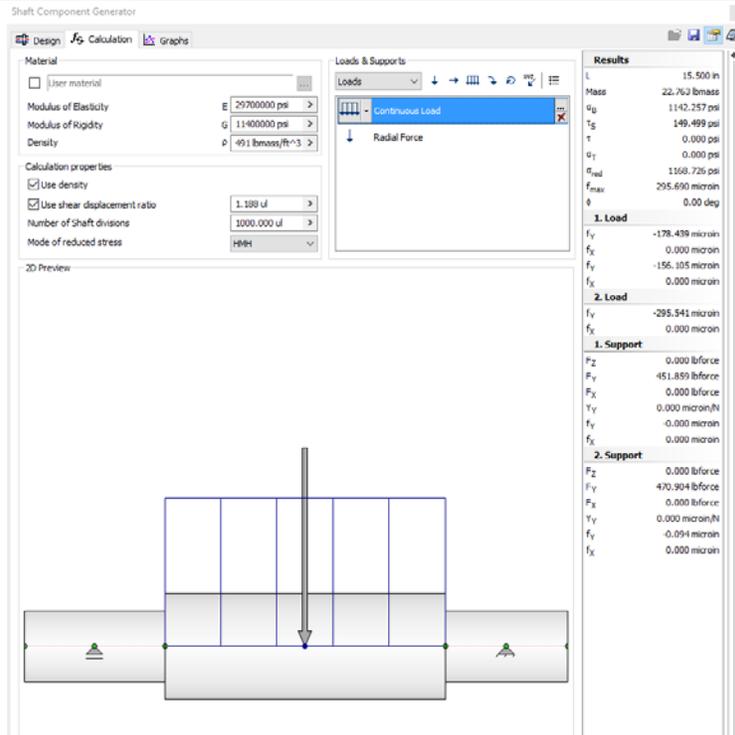


Figure 14 Autodesk Inventor shaft design

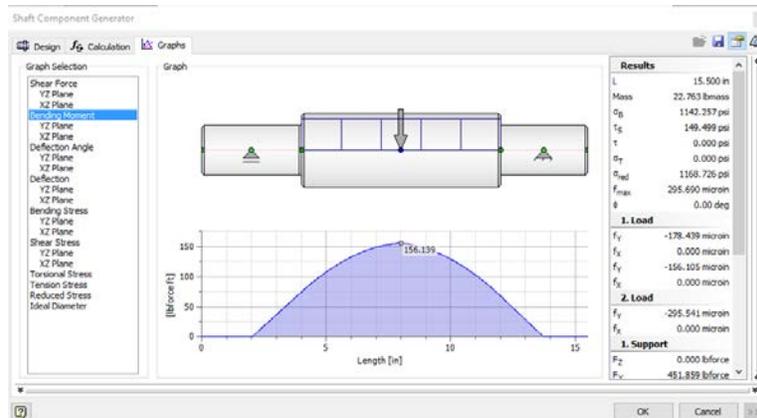
Once the shaft geometry is defined, loading can be added as shown in Figure 15 (a). The applied loading is used in conjunction with the defined geometry to determine stresses and deflections. Graphs of shear, moment, and other conditions can be generated as shown in Figure 15 (b).



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(a)



(b)

Figure 15 Shaft analysis tool



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6.3 Spring Design Tool

As another general example of built-in design tools, Autodesk Inventor also has a spring design module. Figure 16 shows the inputs for an extension spring design. The user inputs values for the overall geometry of the desired spring. Calculations are then done in Inventor to determine properties such as spring constant.

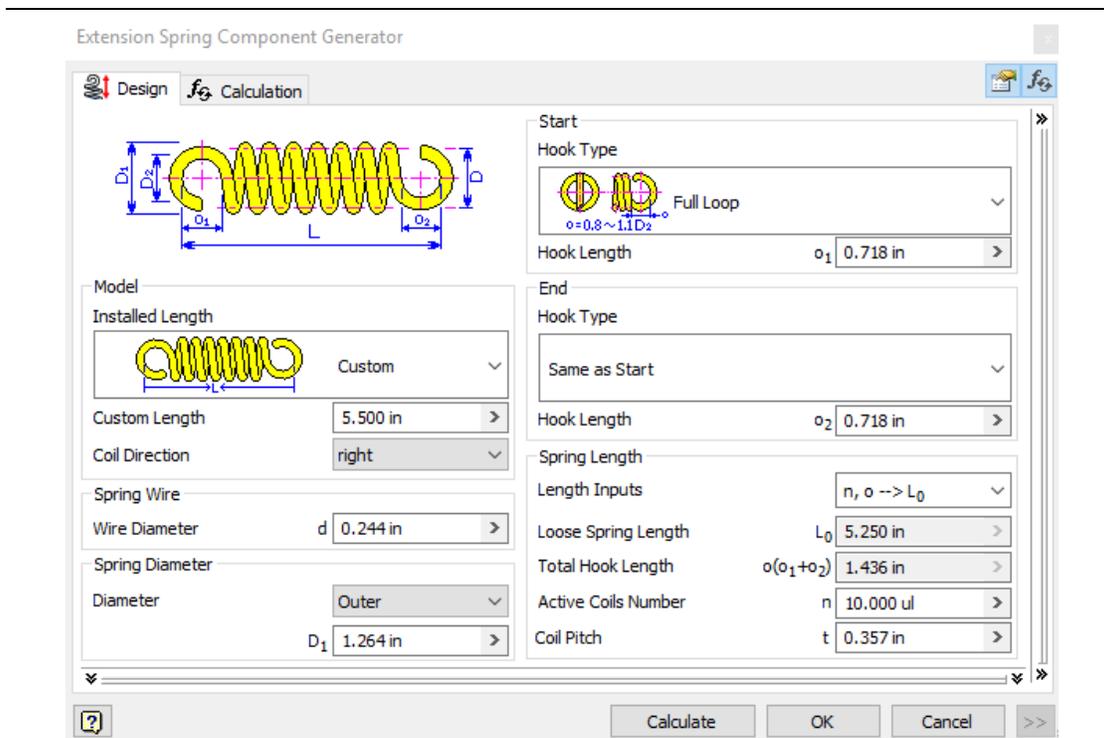


Figure 16 Extension spring design tool (Autodesk Inventor)

Once the design is complete, Inventor will generate the solid model of the spring as shown in Figure 17. Changes can be made within the design module, and the changes will automatically be reflected in the generated solid model.



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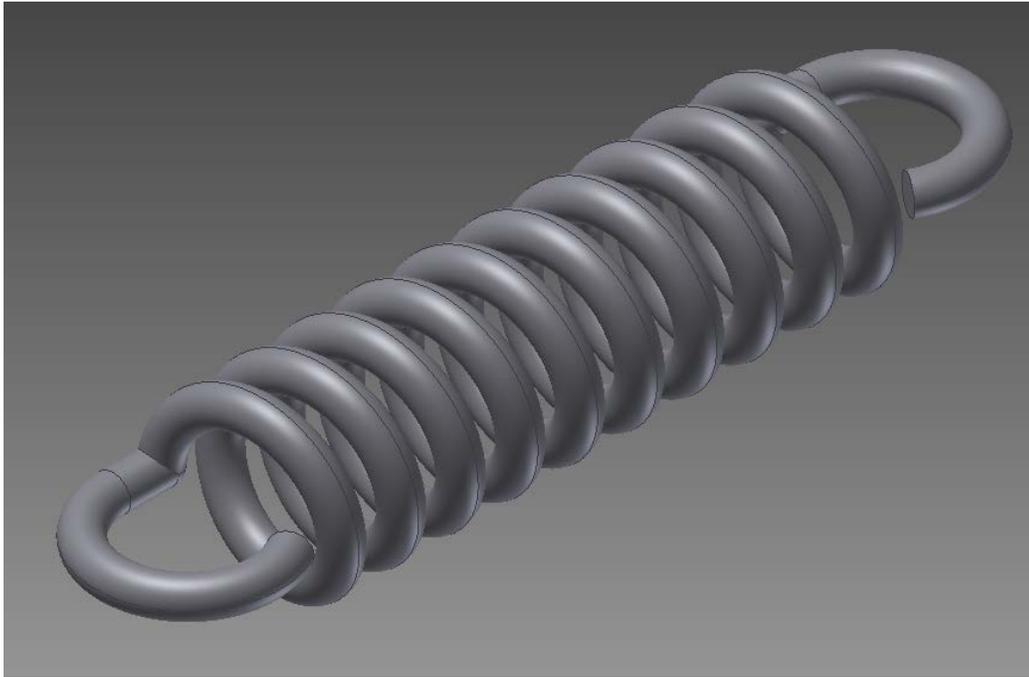


Figure 17 Extension spring generated from the design tool in Figure 16

7.0 Basic Theory of FEA

7.1 Introduction

A full coverage of the theory of finite element analysis is beyond the scope of this course. However, it is important to have a general understanding of the theory to better understand how to perform FEA and to better interpret the results. This section will not get into too much detail of the math involved with FEA, but it will give a simplified overview of how the method works.

To understand how FEA models work, you must first understand the behavior of a very basic spring. If a force is applied to a spring, the spring will deflect. For a linear spring, the deflection is proportional to the applied force. The proportionality constant is known as the spring constant k . Therefore, the force F applied to a spring will equal the spring constant k times the change in length (or deflection) x to give the Equation 1. The equation is known as Hooke's Law.



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Hooke's Law

$$F = kx$$

Equation 1

Many materials, such as steel, follow the same relationship for small deflections. Essentially, a steel rod in tension acts as a stiff spring. Therefore, structural elements can be modeled as springs. Consider the cantilever beam shown in Figure 18, which has a length L and an applied concentrated load P at the free end.

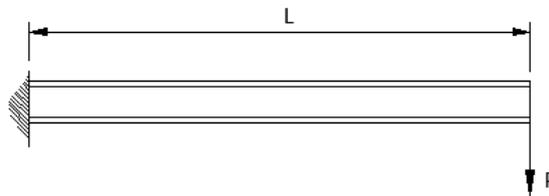


Figure 18 Cantilever beam

From Mechanics of Materials, the deflection at the free end is determined from

$$\delta = \frac{PL^3}{3EI}$$

where E is the modulus of elasticity and I is the moment of inertia. The equivalent 'spring' stiffness can be determined using $F = kx$ (or $P = k\delta$) and the deflection equation. The spring constant for the cantilever beam can then be determined to be

$$k = \frac{P}{\delta} = \frac{3EI}{L^3}$$

For this very basic example, the cantilever beam could be modeled as a single spring with the above stiffness. The behavior of the beam could be determined using Hooke's law, and the deflection can be determined from the applied loading.



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7.2 One-Dimensional FEA Theory

More complex FEA requires the use of many springs arranged as a system. Consider the multiple degree of freedom (DOF) spring mass system shown in Figure 19, which could represent a one-dimensional discretized system.

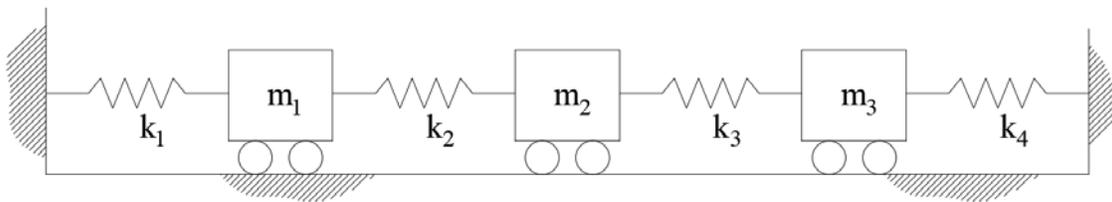


Figure 19 Multi-DOF spring mass system

A force F_1 is now applied to mass 1. As shown in Figure 20, the applied force will cause mass 1 to move a distance x_1 . The other two masses are held in their original positions.

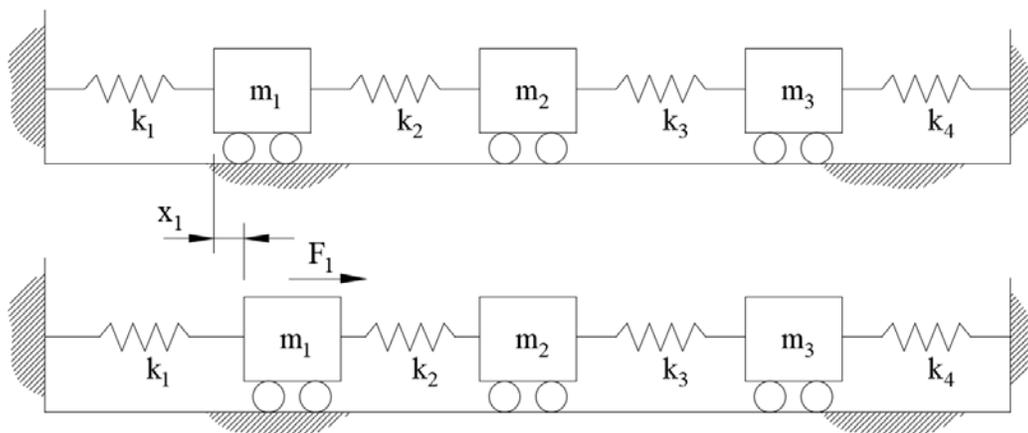


Figure 20 Force applied to mass 1



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Free body diagrams can now be drawn for the masses. Note that mass three will not have any forces because mass 2 is not allowed to move. The free body diagrams for mass 1 and mass 2 are shown in Figure 21.

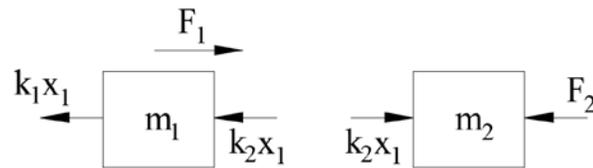


Figure 21 Free body diagrams for F_1

Mass 1 will have the applied load F_1 . The springs attached to mass 1 will apply a restoring force resulting in forces k_1x_1 and k_2x_1 . The force k_2x_1 will transfer a force to mass 2. Because mass 2 is not allowed to move, a force F_2 will resist motion. The free body diagrams give Equation 2 and Equation 3. Note that the equation for F_2 is negative because it is acting to the left. For consistency, all forces are considered positive when acting to the right to agree with the applied force directions (F_2 is shown positive to the right in Figure 22 as an applied force).

$$F_1 = (k_1 + k_2)x_1 \quad \text{Equation 2}$$

$$F_2 = -k_2x_1 \quad \text{Equation 3}$$

Similarly, a force F_2 can be applied to mass 2. The applied force will cause mass 2 to move a distance x_2 , while all other masses are held stationary. The resulting system is shown in Figure 22.



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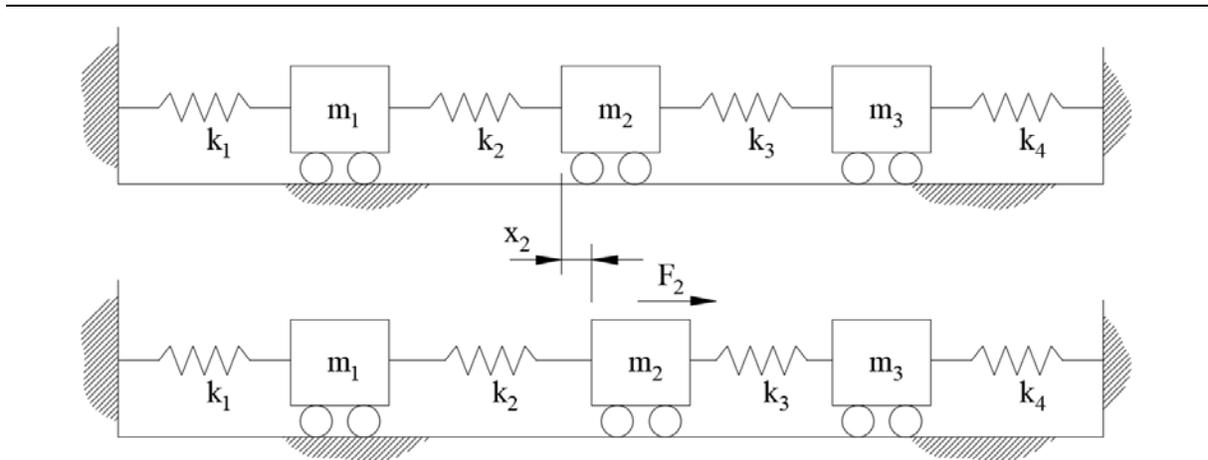


Figure 22 Force applied to mass 2

Free body diagrams are developed for all three masses with the results shown in Figure 23.

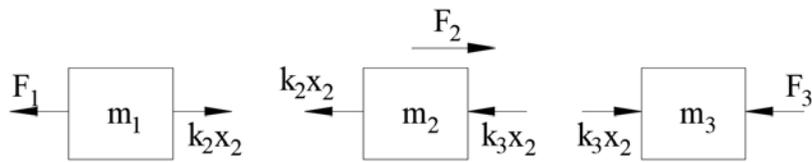


Figure 23 Free body diagrams for F_2

Mass 2 will have the applied force F_2 . Due to its motion, mass 2 will have restoring forces k_2x_2 and k_3x_2 . The force k_2x_2 will be transmitted to mass 1, resulting in a force F_1 to hold the mass stationary. The spring force k_3x_2 will be transmitted to mass 3, which is held stationary by a force F_3 . The resulting equations are shown Equation 4, Equation 5, and Equation 6.



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$$F_1 = -k_2 x_2 \quad \text{Equation 4}$$

$$F_2 = (k_2 + k_3) x_2 \quad \text{Equation 5}$$

$$F_3 = -k_3 x_2 \quad \text{Equation 6}$$

Lastly, mass 3 will have an applied force of F_3 causing it to move a distance x_3 . The system is shown in Figure 24.

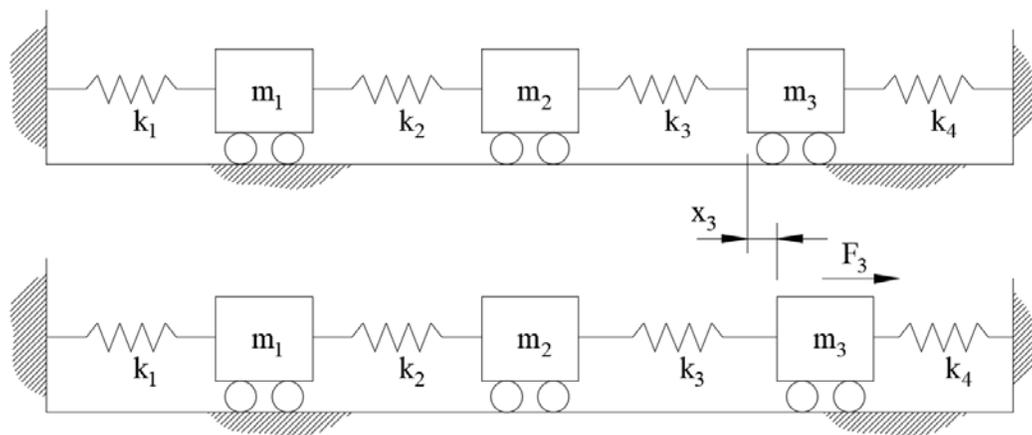


Figure 24 Force applied to mass 3

The free body diagrams are developed using the same procedures described above, with the results shown in Figure 25. Note that mass 1 will not have any forces because mass 2 is held stationary. Mass 3 will have an applied force and two restoring forces. Mass 2 will have a transmitted restoring force, which must be counteracted by a force to hold the mass stationary.



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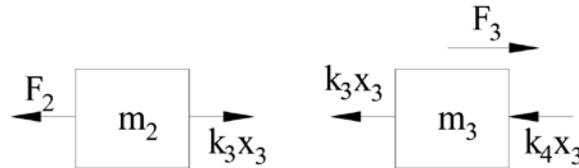


Figure 25 Free body diagrams for F_3

The equations from the free body diagrams are given in Equation 7 and Equation 8.

$$F_2 = -k_3x_3 \quad \text{Equation 7}$$

$$F_3 = (k_3 + k_4)x_3 \quad \text{Equation 8}$$

All the equations developed from Equation 2 through Equation 8 can now be expressed in matrix form, which would result in Equation 9.

$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix} = \begin{bmatrix} k_1 + k_2 & -k_2 & 0 \\ -k_2 & k_2 + k_3 & -k_3 \\ 0 & -k_3 & k_3 + k_4 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} \quad \text{Equation 9}$$

Equation 9, which is Hooke's law (Equation 1) in matrix form, is a set of simultaneous equations that can be solved to give the displacements of all three masses based on any set of applied forces.

The matrix in Equation 9 with the spring stiffness values is known as the stiffness matrix, which defines the model characteristics and the resistance of change when subjected to loading. The stiffness matrix will be symmetric and will have a size based on the degrees of freedom (3x3 for this case). Applied forces would be known, and the displacements could be calculated using



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Equation 9 based on the stiffness matrix. Once deflections are known, the derived variables such as stress, can be calculated.

7.3 Two and Three-Dimensional FEA Theory

Once we understand that we can model simple one-dimensional systems of springs, we can now connect springs in specific patterns to model two or three-dimensional solids. Full coverage of this topic is beyond the scope of this course, but a quick illustration will be discussed. Modeling a two-dimensional flat plate element, for example, could be done using a set of springs shown in Figure 26.

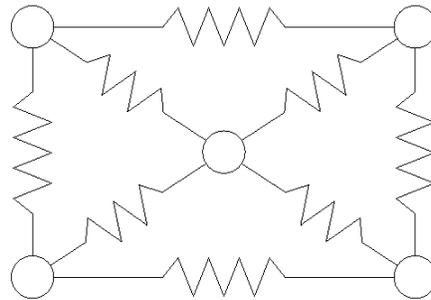


Figure 26 Spring model for 2D flat plate element

Based on the arrangement of springs, a stiffness matrix can be determined.

Boundary conditions would be defined along with applied loading. Displacement of each calculation point could then be determined from the matrix form of Hooke's law, similar to that shown in Equation 9. Once displacements are computed, derived variables can be determined. The spring model in Figure 26 could be expanded into three-dimensional shapes to model 3D solids.

The basic theory defined so far focused on developing a matrix form of Hooke's law. Recall that Hooke's law is only valid for small deformations (within the elastic range of the material). For



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large deformations the analysis becomes more complex. A full discussion of non-linear analysis is beyond the scope of this course.

7.4 FEA for Heat Transfer

The theory discussed so far has focused on the idea of deformation and stress analysis. However, FEA has other applications such as heat transfer. Again, full development of the equations is beyond the scope of this course. However, the basic idea will be the same for heat transfer. We can still consider the equations like Hooke's law with force being replaced with heat flux and displacement being replaced with temperature. The stiffness matrix will now represent a resistance to heat flow through the material rather than its resistance to deformation.

8.0 Example Applications of FEA

8.1 Introduction

Many solid modeling software packages include a module for finite element analysis. Solid models can also be exported to be used in a separate FEA software such as ANSYS. One major advantage to using the same software for generating the model and performing FEA is the ability to perform fast analysis and design changes. This rapid repetition of analysis and design change makes iterative design very efficient.

The finite element capabilities in solid modeling software can be used for one-dimensional problems such as trusses. However, other software is commonly available for analyzing trusses and beams. Therefore, this section will focus on FEA of three-dimensional volumes (or two-dimensional representations of three-dimensional volumes if one dimension is small).



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8.2 Steps in the FEA Process

8.2.1 Preprocessing

This step defines the geometry of the problem. Preprocessing involves creating and importing a 3D solid model, mesh generation, applying loads and supports, and defining the relationship of stress to strain (linear model or non-linear model). Element types are defined in this step along with the material properties. The boundary conditions are also defined in this step.

8.2.2 Computation

The computation process, or solution process, occurs within the FEA software. The stiffness matrices are generated, and the unknown values of deflection are calculated in this step. From the deflection values, derived variables like stress and reaction forces are calculated.

8.2.3 Post Processing

The post processing step also occurs within the FEA software. Post processing is used to print and plot results.

8.3 Type of Analysis

8.3.1 Static vs. Dynamic

When determining if the analysis type is static or dynamic, you need to determine if the loads are applied gradually or quickly. Static analysis is used when loads are applied gradually and will assume that inertial effects are negligible. Transient dynamic analysis can be used to determine the response to general time-dependent loads such as impact events. Another common dynamic analysis would be modal analysis, which is used to identify natural frequencies.

8.3.2 Linear vs. Nonlinear

In a linear analysis, there will be a linear relationship between applied loading and displacements. For example, a 100 lb force causes 10 times the deformation of a 10 lb force. Analysis times for linear analysis are much faster compared to non-linear, so linear analysis is often a first estimate to more complex non-linear problems.



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When there is a nonlinear relationship between applied loading and deformations, the problem is a nonlinear analysis. Nonlinearities can be geometric or material. Geometric nonlinearities would include large deformation or stress stiffening. In large deformation the stress may not stay in the elastic region. Material nonlinearities could include creep or plasticity. It could be important to know how the part was manufactured. Cast parts can have variations of material properties. Sheet metal parts that have been formed can have residual stresses at the bend locations. Another area of nonlinear analysis would be contact. Cases could include situations where parts can separate from each other, such as gears in contact.

When picking FEA software, it is important to understand types of analysis you plan on using. Many basic FEA software packages may be limited to linear static analysis, or they may have limited functionality in dynamics such as basic modal analysis.

8.4 Element Types and Mesh Generation

Finite element analysis uses elements to create a model. Different element types exist and selecting the proper type of element is critical to properly predict the behavior of the solid. One-dimensional beam elements are best used for cases with high aspect ratios (dimension is significantly larger in one direction compared to the other two). Structural trusses would be an example of a case for beam elements. Thin wall structures are good cases to use two-dimensional shell elements. Thicker solid components should be modeled using three-dimensional elements.

Once a solid model is created you must generate a mesh used to discretize the part for finite element analysis. Most CAD software will automatically generate a mesh to be used in finite element analysis. Figure 27 shows an example of a mesh generated using Autodesk Inventor. The user has some control over how the mesh will be generated. Overall mesh size, for example, can be controlled to generate a course mesh or fine mesh. Course mesh size will drastically reduce the time required to perform the analysis, but the results may not be as accurate.



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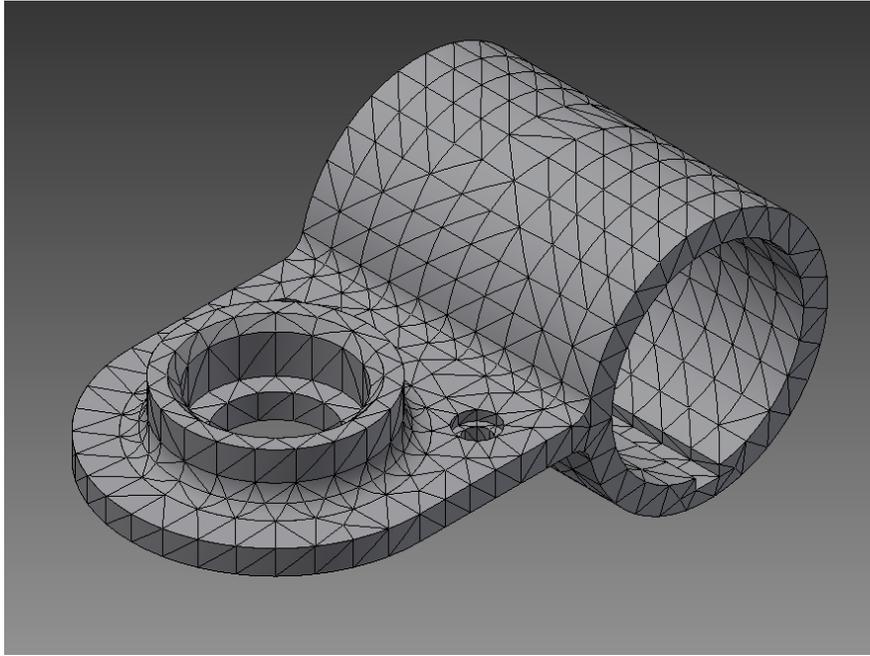


Figure 27 Automatic mesh generation (created in Autodesk Inventor)

The accuracy of the model can be determined by its convergence error. Convergence error is the percent difference between consecutive iterations after making changes to the mesh. There are two methods used for the changes to check convergence error: the H-element method and the P-element method. The H-element method reduces the size of the elements, and the P-element method increases the order of the polynomials. Basically, the P-element method uses a constant mesh.

The finite element method is typically iterative to allow for convergence studies. The first step is to build the model; which includes defining material properties, applying loads and constraints, and applying the mesh. The second step is to solve the model. The last step is to refine the mesh and resolve, making sure to record solution values. A basic result of this procedure can be seen in Figure 28. Stress at a specific point of interest is taken from each analysis. As the mesh is refined (number of nodes increased), the stress at the same point is recorded and compared to previous values. Though the actual true value is unknown, convergence is based on a stress value being similar to the previous value. If a small change (maybe around 3%) in the results



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occur with additional refinement, then no further refinement is needed, and convergence has occurred.

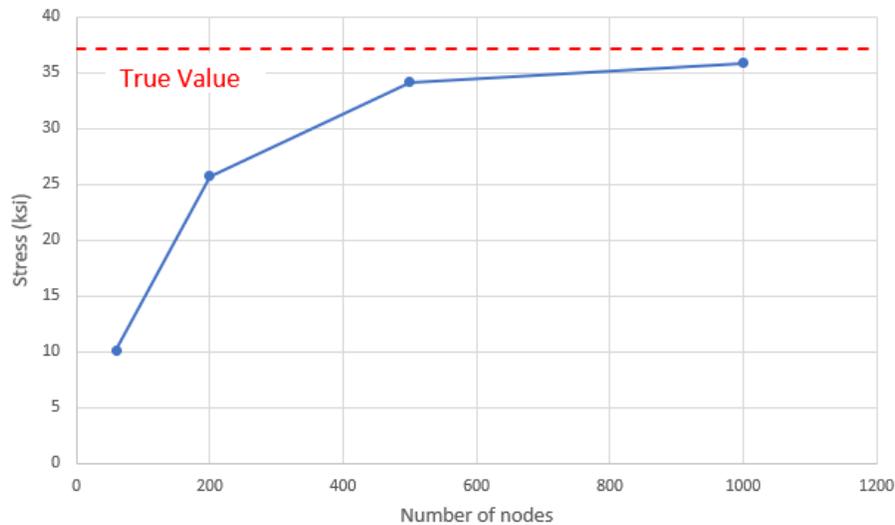


Figure 28 Convergence plot

8.5 2D FEA Applications

We now consider a case utilizing two-dimensional analysis. The FEA model can be simplified in cases where the member to be analyzed is thin and the loads act only in the plane. The simplified model will use two-dimensional solid finite elements. Two-dimensional surface analysis will use shell elements, which are elements that have small thickness compared to other dimensions.

Consider the plate shown in Figure 29. The plate is 1/4" thick and made from AISI 120 steel. The dimensions shown are in inches. The plate will be loaded in the long axis direction. This example is a common application looking at stress concentrations.



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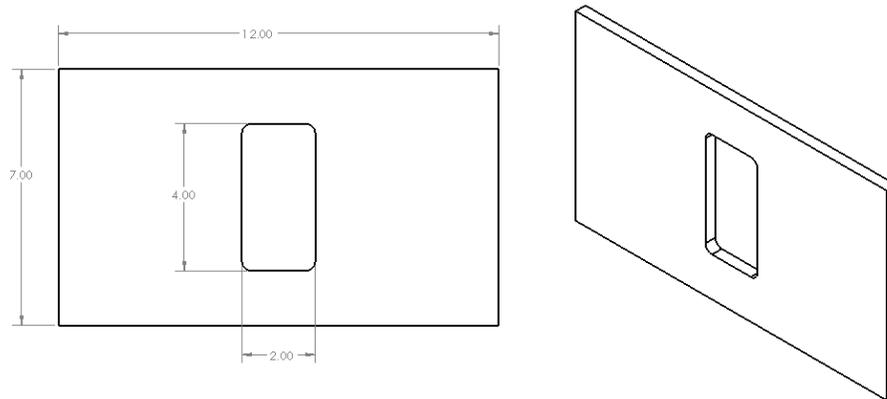


Figure 29 Thin plate example

This example problem will use the advantage of the part's symmetry. Only one quadrant of the plate will be used, as shown in Figure 30.

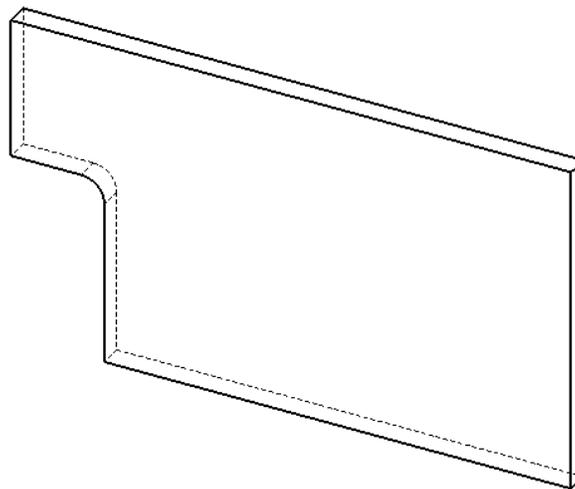


Figure 30 Plate quadrant to be modelled



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The boundary conditions can now be applied as illustrated in Figure 31. The bottom edge is constrained to have zero translational displacement in the normal direction (thickness direction) and the vertical direction (width direction). The side edge is constrained to have zero translational displacement in the normal direction (thickness direction) and the horizontal direction (length direction). Both edges have rotational constraints to cause zero rotation about the vertical and horizontal axis directions.

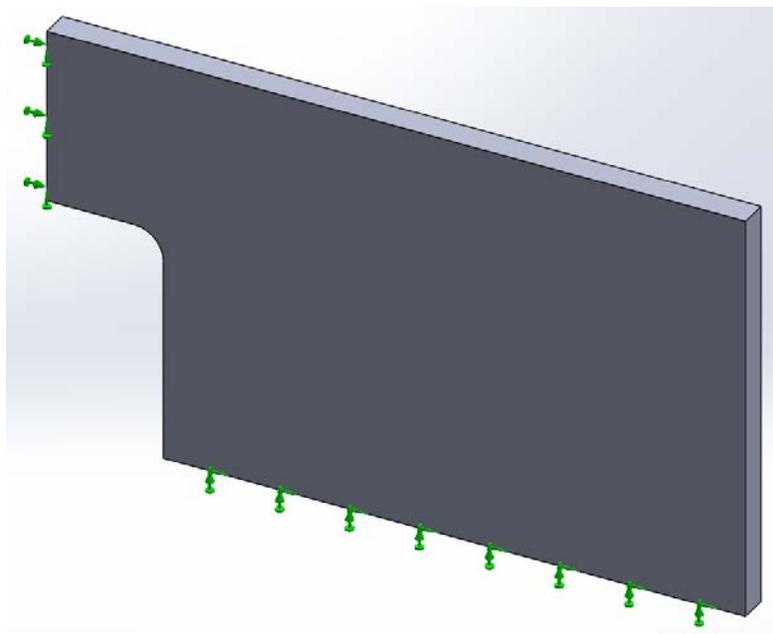


Figure 31 Boundary conditions (generated in Solidworks)

The desired loading can then be applied to the edge, as shown in Figure 32.



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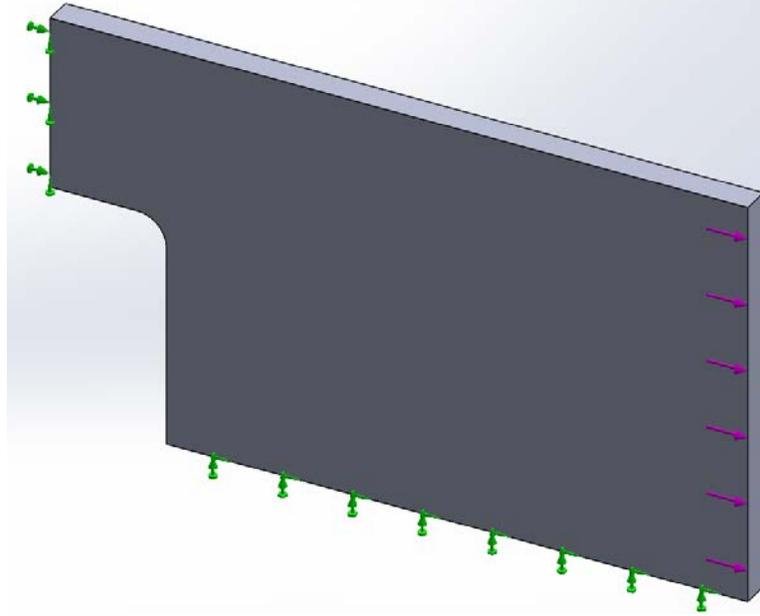


Figure 32 Model with applied loading

The next step is to generate the mesh. A relatively course mesh can be used initially, as shown in Figure 33. The figure shows a mesh with a global element size of 0.375.



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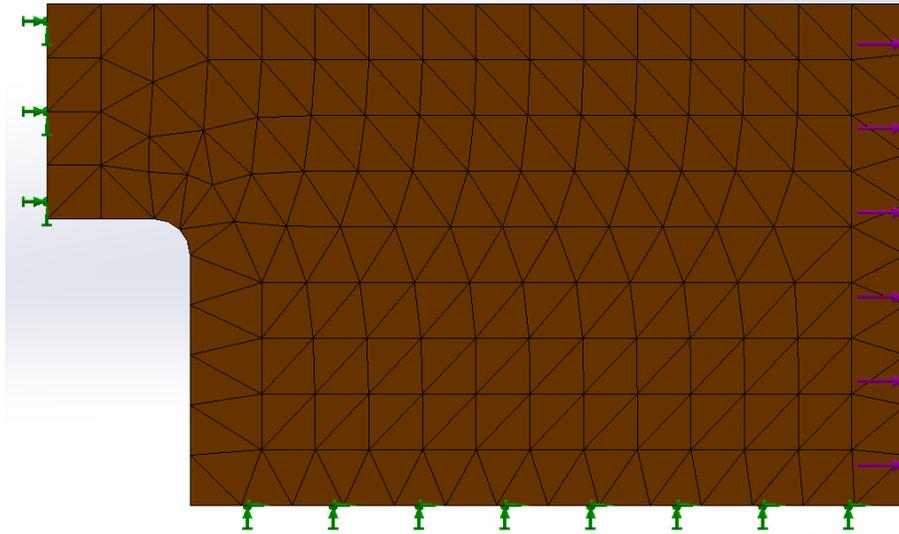


Figure 33 Initial mesh (created by Solidworks)

Now that the mesh is created, the solver can be run to generate results. The principal stress results are shown in Figure 34. The symmetry constraints forced the bottom edge to remain horizontal and the left edge to remain vertical. The mesh can be refined to check for convergence [the process of convergence checking will be discussed for the 3D application in the next section].



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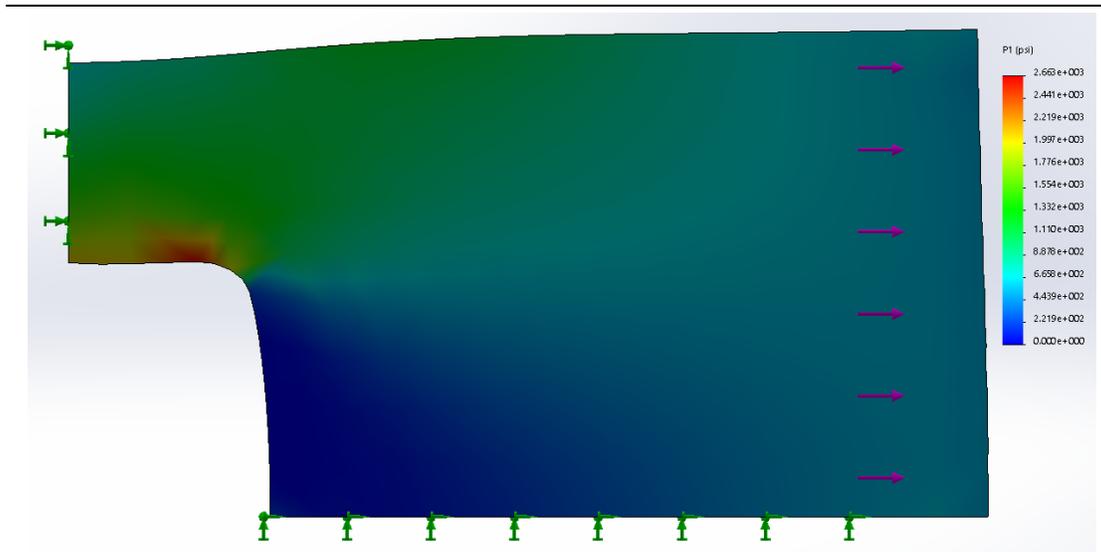


Figure 34 Principal stress results for initial mesh (generated in Solidworks)

8.6 3D FEA Applications

The real power of FEA is in the analysis of three-dimensional complex forms. To illustrate the procedure, consider the generic part shown in Figure 35. We will assume that the base faces of the semicircular disk will be fixed, and the outer surface will carry a pressure loading.



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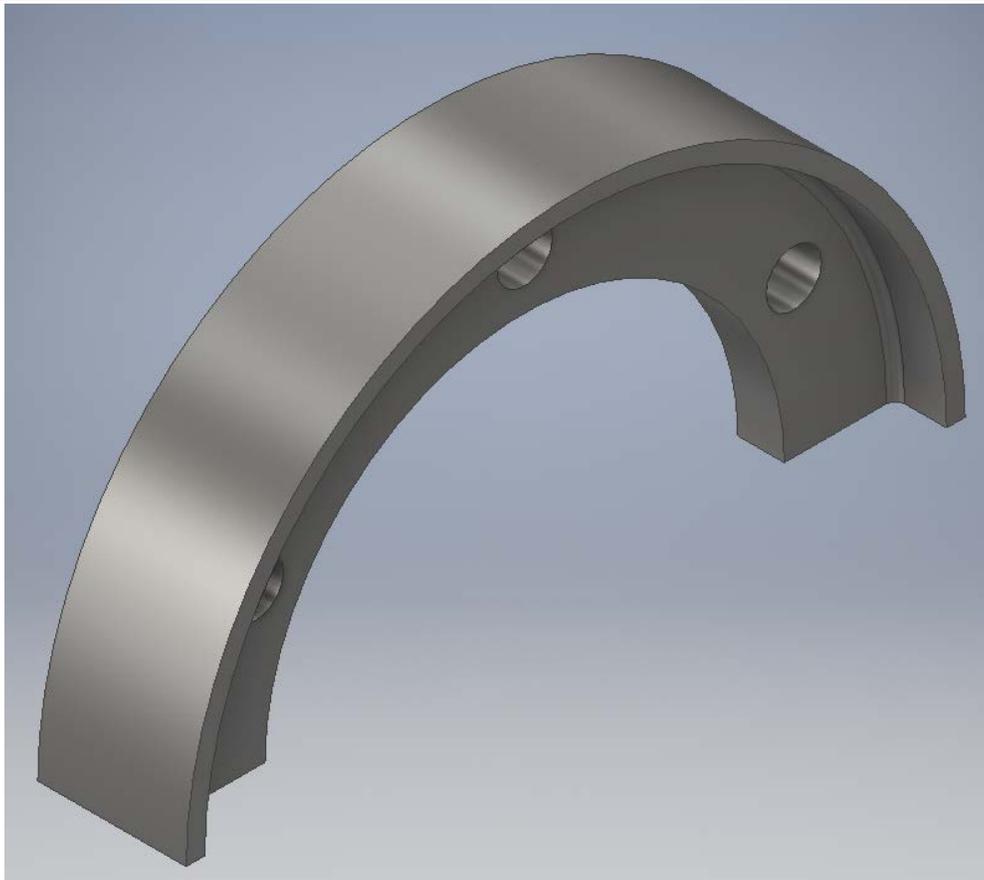


Figure 35 3D part (generated in Inventor)

Figure 36 shows the constraints added to the bottom faces fixing those surfaces in all directions. An outer pressure loading is applied. Inventor generates the initial mesh for the solid.



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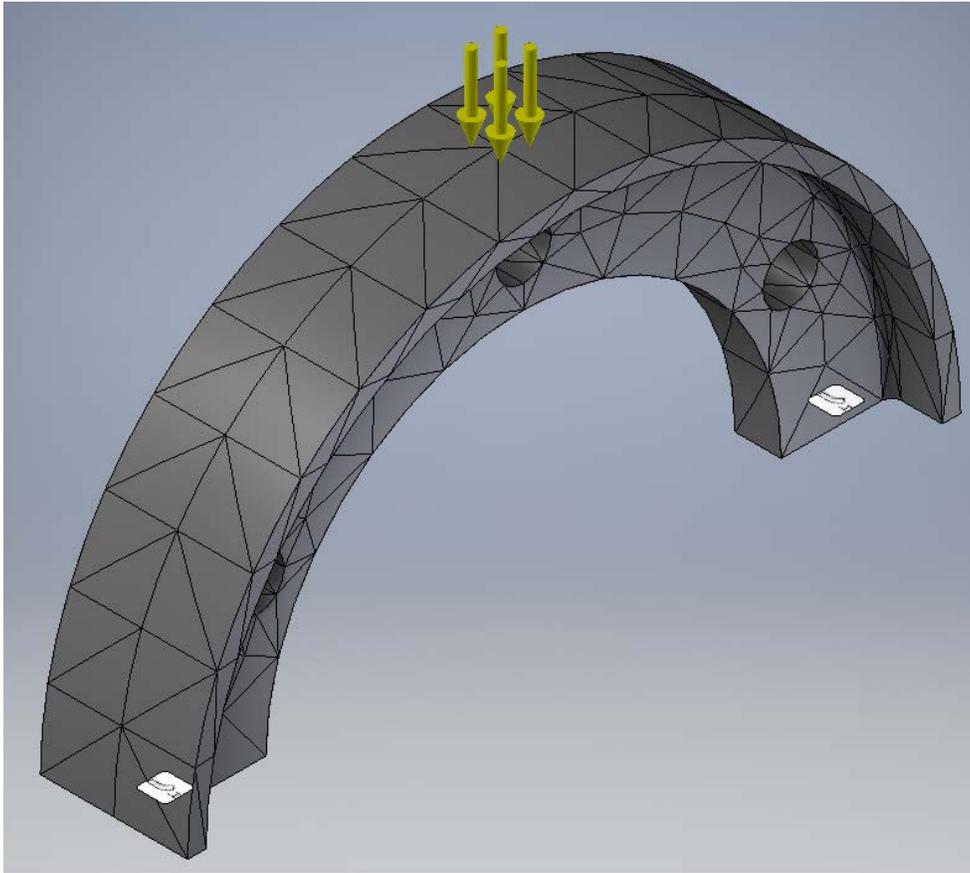


Figure 36 3D part with constraints, loading, and initial mesh (generated in Inventor)

For the analysis we will use Inventor's built in convergence tool. Figure 37 shows the dialog box for the convergence settings. The maximum number of H refinements has been set as 5, so the analysis will stop after a maximum of 5 refinements. The stop criteria setting of 3% indicates the analysis will stop if results between refinements has less than a 3% change in value.



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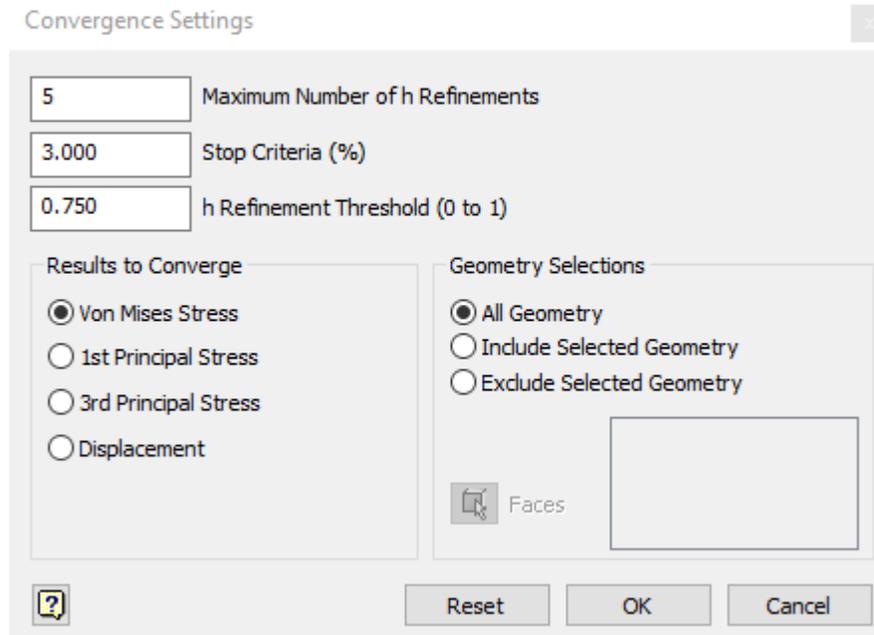


Figure 37 Inventor convergence settings

The H refinement threshold has been set as 0.75. A zero setting would include all the elements as candidates for refinement. A value of 1 would exclude all elements from refinement resulting in no refinement. The value of 0.75 would result in the elements with equivalent errors at the top 25% being subject to refinements. The setting indicates that Von Mises stress will be used for refinement and that all geometry will be included. The final result for Von Mises stress is shown in Figure 38.



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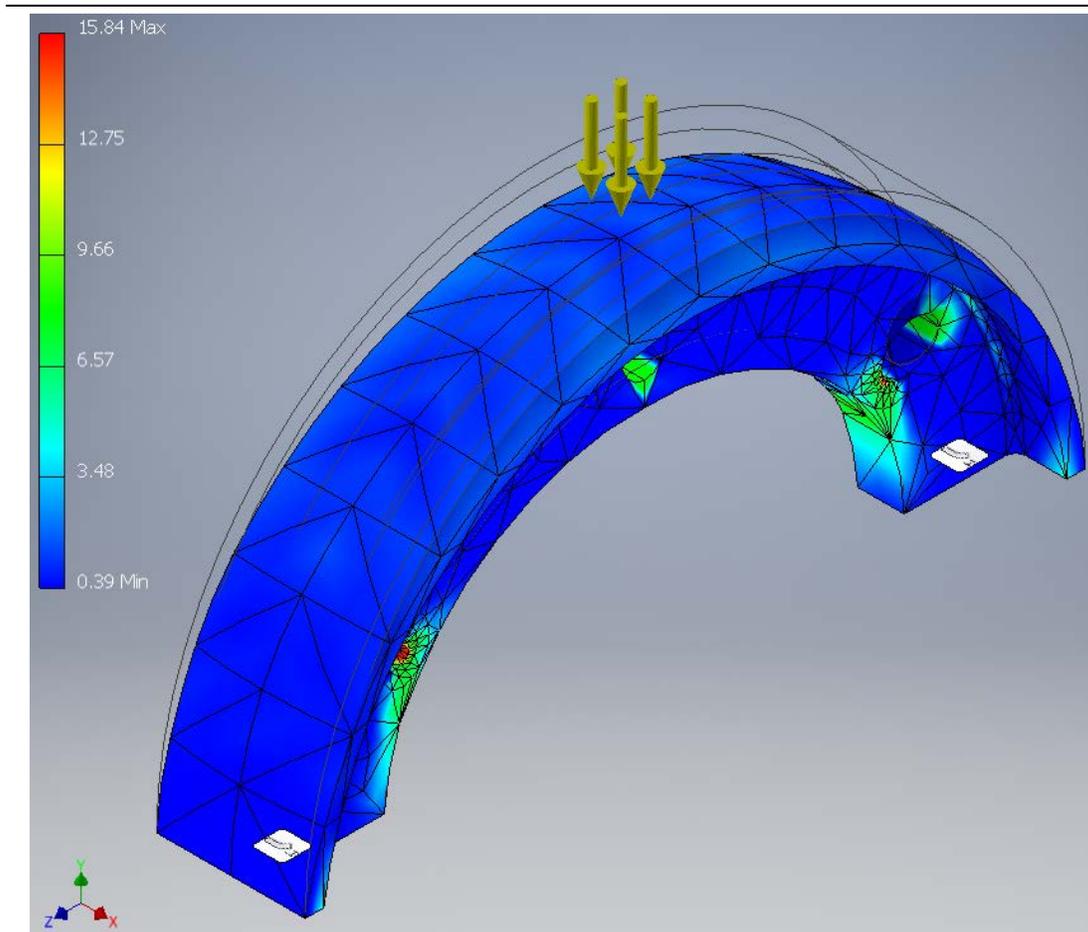


Figure 38 Stress results

Inventor will track the results from each refinement to give a convergence plot. The plot for this analysis is shown in Figure 39. The maximum stress is converging to a value of 15.84 ksi.



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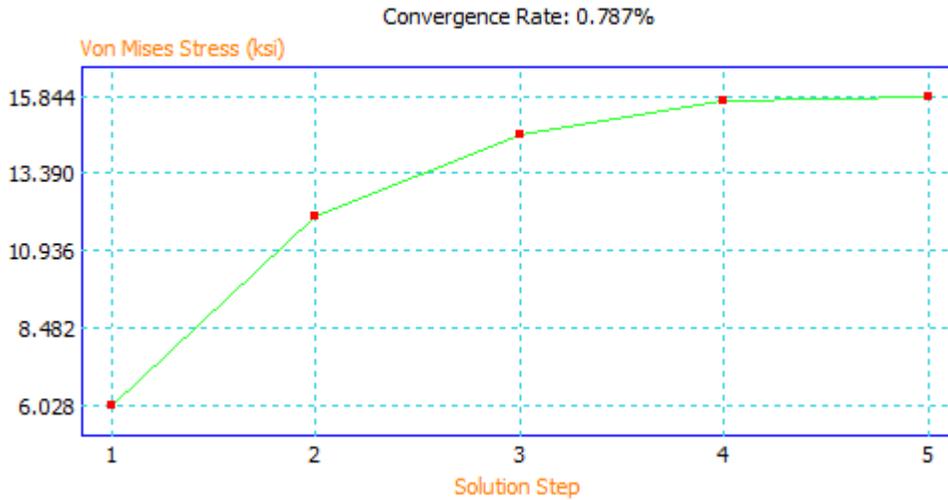


Figure 39 Convergence plot

The h refinement threshold was set as 0.75, which indicated that only the elements with equivalent errors at the top 25% are subject to refinements. This concept is known as localized refinement. It can be seen in Figure 40 that the elements in the high stress areas around the hole are refined. The outer surface, for example, did not need refinement and is left as a coarse mesh.



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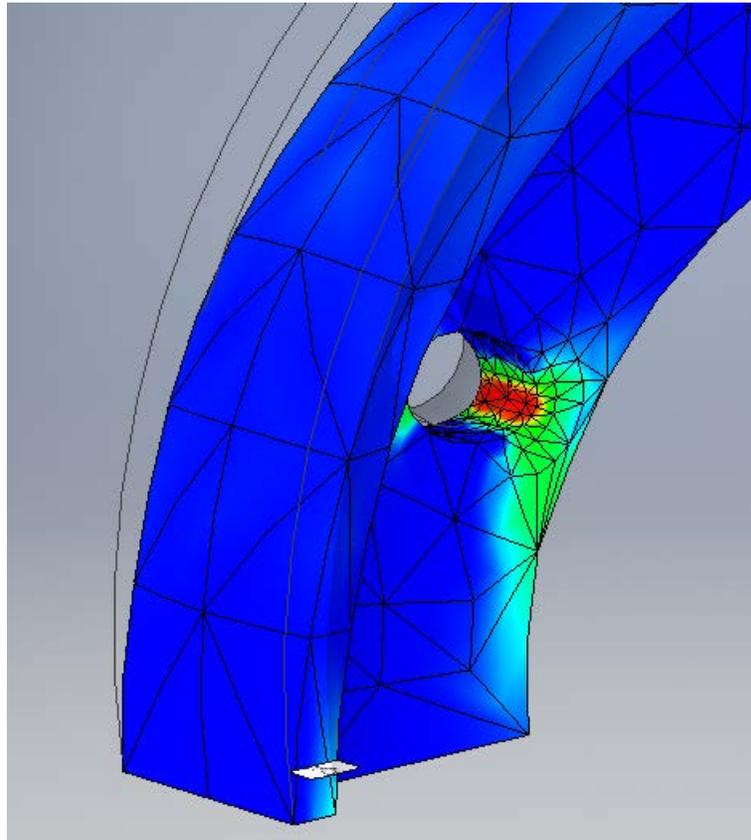


Figure 40 Localized refinement

8.7 Errors in FEA Results

It is very important to remember that FEA is a tool used by engineers. Hand calculations or experiments should be used to provide rough estimates of the expected results. If the results obtained from FEA do not match those expected, it is important to check the model for errors. This last section will briefly discuss common errors.

There will be three primary sources of error: user error, modeling error, and discretization error. User errors involve setting up the model incorrectly and generally stem from lack of experience. Modeling errors involve incorrect representations of the situation, such as using incorrect



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constraints. The last category of discretization error will be caused by an insufficient mesh to accurately capture the results. Two areas of discretization error, mesh size and element distortion, will be discussed.

8.7.1 Mesh Size

One common source of errors would be a mesh that is too coarse. As the mesh is refined, the solution should become more accurate. The concept of convergence was discussed in Section 8.4 and should be used to determine accuracy. Running more than one test can identify areas within the model that change a lot when the mesh is refined, and other areas may not change much at all from refinement. An efficient way to refine a mesh, therefore, is to focus on refining the areas of the model that change the most while leaving the other areas a coarser mesh. This concept was illustrated in Figure 40.

8.7.2 Element Distortion

Distorting the element shape will tend to reduce accuracy and will make the element stiffer. Because FEA is commonly used on complex shapes, distorting cannot be completely eliminated. You do, however, want to keep element shapes as compact and regular as possible. For example, an ideal triangle would be equilateral, and an ideal hexahedron will be a cube. An ideal quadrilateral, as shown in Figure 41, would be a square. The figure also shows two methods of distorting that shape. You want to avoid large aspect ratios; a length to height ratio of a rectangular shape should generally be less than 3. A skew angle should generally be less than 30 degrees.

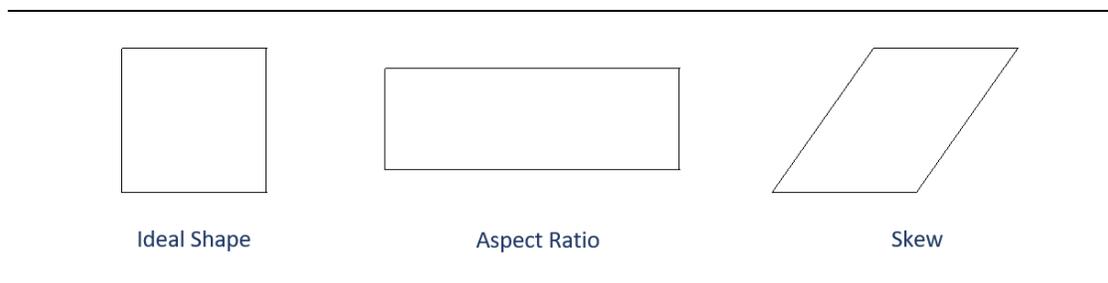


Figure 41 Element distortion