Computer Aided Engineering Training

Part I: FEM Simulation (Solid)

Keerati Suluksna, Ph.D.
School of Mechanical Engineering,
Institute of Engineering,
Suranaree University of Technology

Training Agenda

Day 1:

- ☑ Solving Engineering Problems
- ☑ Finite Element Method Overview
- ☑ 1-D FEM Analysis
- ☑ FEM Software Process
- ☑ Preparing 1-D CAD File
- ☑ 1-D Element Case Studies
- ☑ Preparing 2-D CAD File
- ☑ 2-D Element Case Studies

Training Agenda

Day 2:

☑ Preparing 3-D CAD File
☑ 3-D Element Case Studies
☑ Some Special Problem

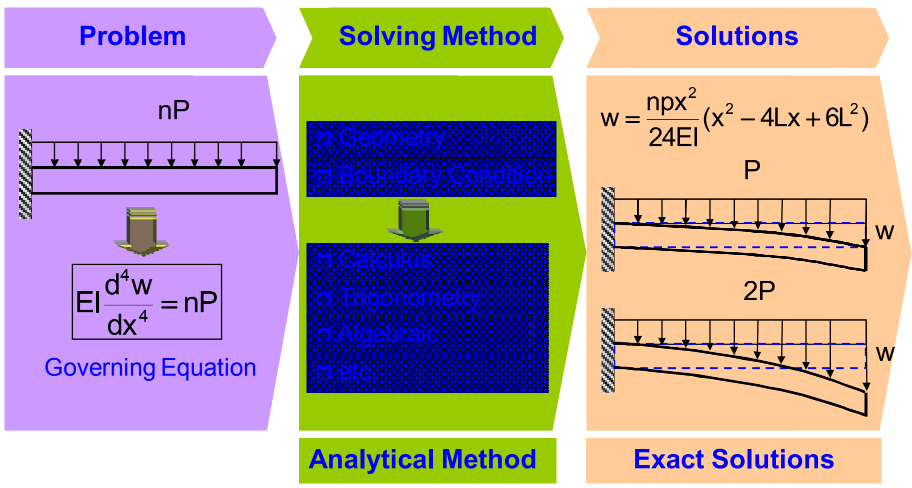
- CFD Overview
- Flow Domain Construction with Gambit
- Grid Generation Techniques
- 2-D Simple Flows Analysis with Fluent

Day 3:

- 2-D Complex Flows Analysis with Fluent
- Unsteady Flow Simulation
- Some Special Problem
- Training Summary

3

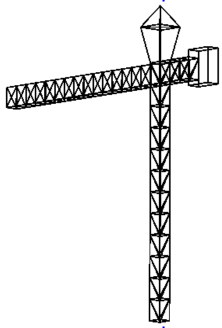
Solving Engineering Problems:
Analytical Method



4

Solving Engineering Problems: Analytical Method

Problem → **Analytical Method** → **Exact Solution**



- Geometry → Complex
- Equation → PDE

$$EI\left(\frac{\partial^4 w}{\partial x^4}+\frac{\partial^4 w}{\partial y^4}+\frac{\partial^4 w}{\partial z^4}\right)=P$$

How to solve this problem?

???

Solving Engineering Problems: Experiment

Problem → **Experiment** → **Experimental Data**

BMW 328i
(5,999,000 Bath)

COST !!

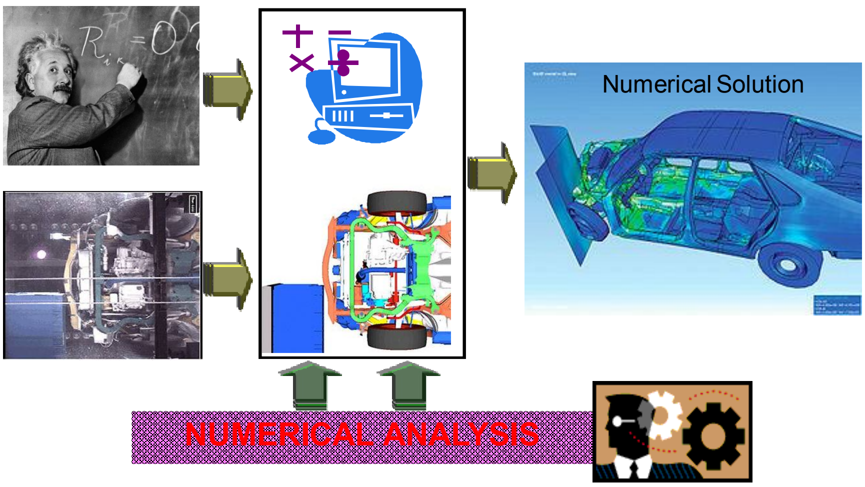
5,999,000x10=!!!

Solving Engineering Problems

FEA of the crash-test between vehicle and lighting columns (Klyavin,2007)

7

Solving Engineering Problems



8

FEM Overview:
A Brief History

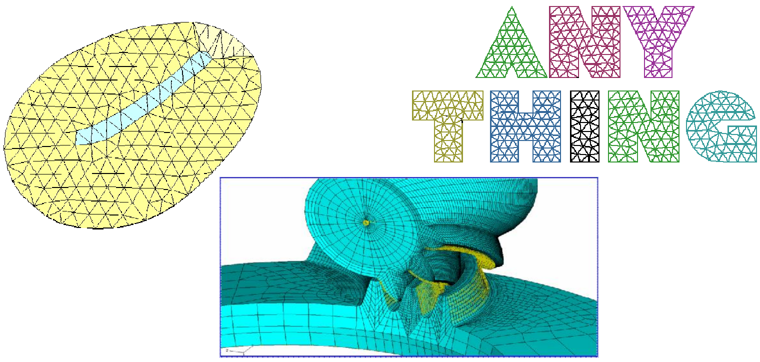
- Finite Element Analysis (FEA) was first developed in 1943 by Richard Courant.
- By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries.
- The rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision.

Richard Courant

9

FEM Overview
How does FEA Work?

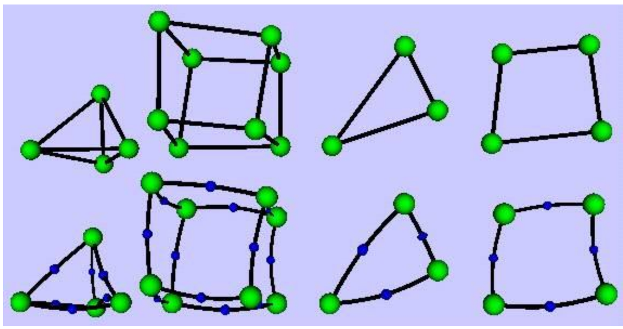
- FEA divides the physical system to be analyzed into a number of discrete elements.
- The complete system may be complex and irregularly shaped, but the individual elements are easy to analyze.



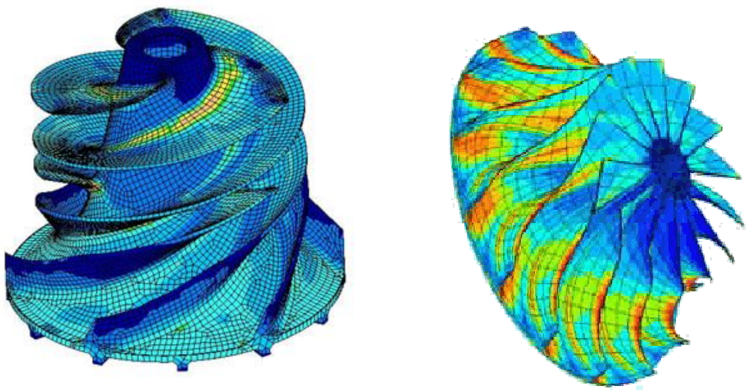
10

FEM Overview
How does FEA Work?

- The elements may be 1-D, 2-D (triangular or quadrilateral), or 3-D (tetrahedral, hexahedral, etc.); and may be linear or higher-order.
- The elements may model mechanics, acoustics, thermal fields, electromagnetic fields, etc., or coupled problems.

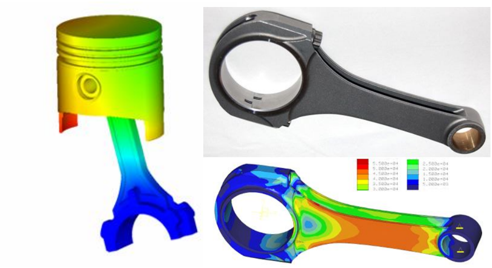


FEM Overview:
Applications



Finite element model of the turbomachinery blades

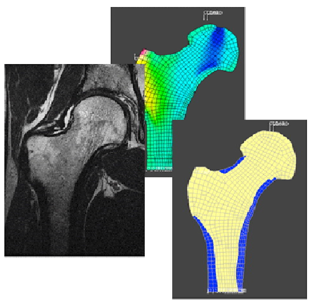
FEM Overview: Applications

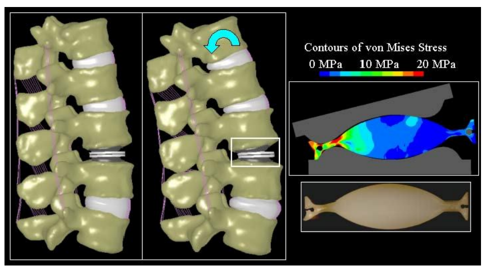


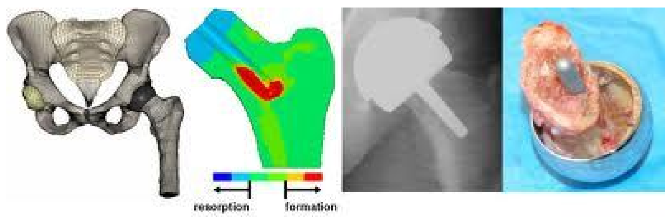
Crank shaft

Car wheel

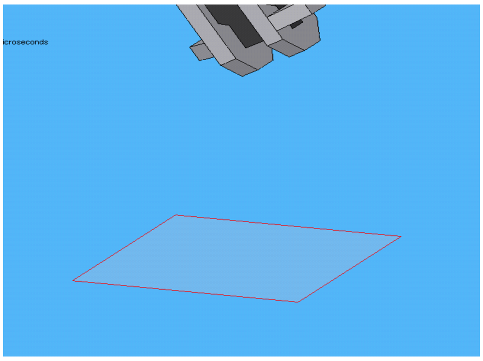
FEM Overview: Applications

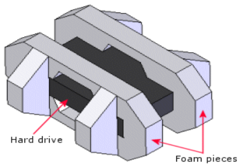


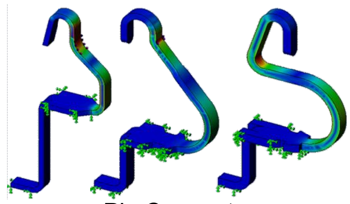




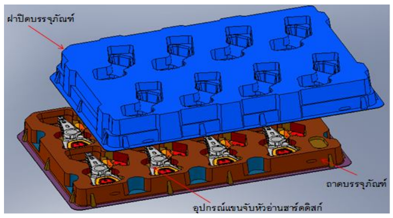
FEM Overview:
Applications







Pin Connectors



Model of a Packaging

FEM Software Process

Redesign

Pre-Processor

Solver

Post-Processor

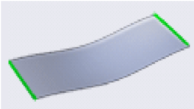
Built

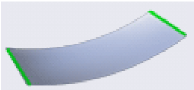
- CAD/ Material Properties
- Conditions/ Load/ Restrain
- Element/Solver Selection
- Yield Criteria
- etc.

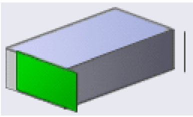
- Mechanics Fields
- Thermal Field
- etc.

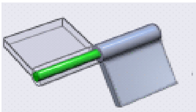
- Visualization/ Display

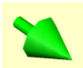
Fixtures/ External Loads

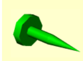

Fixed Geometry

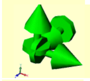

Immovable (No translation)

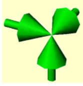

Roller/Slider

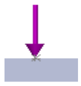

Fixed Hinge

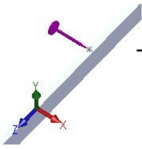

No translation in the direction of the arrow points


No rotation around the direction of the point


Fixed Geometry


Immovable (No translation)


Force/ Pressure


Torque/Moment

17

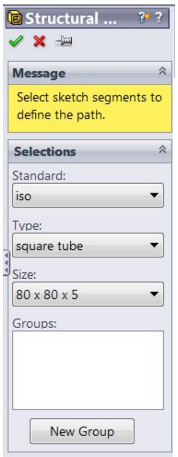
1-D Element Case Studies

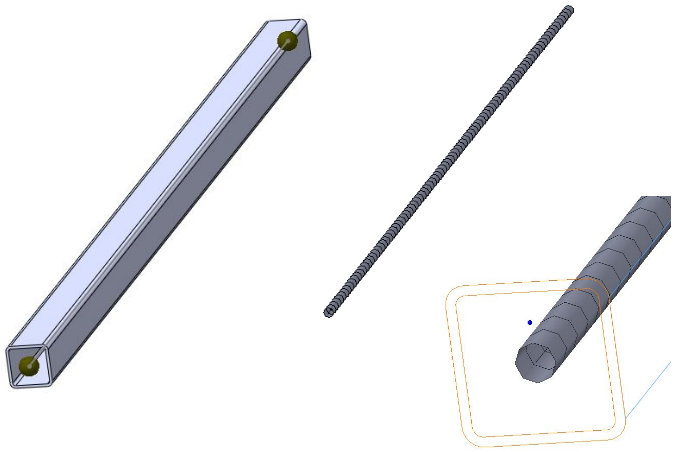
Static Analysis of Trust

18

Preparing 1-D CAD File

Structural Member and Beam Element





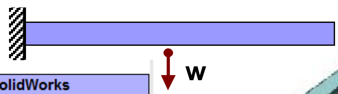
Case D1-1

Deflection of a Cantilever Under Gravity

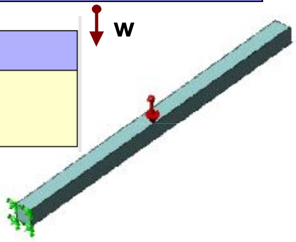
Description

Determine the maximum displacement in Y-direction of a cantilever, fixed at one end, under its own weight. The length of the cantilever is 20" and its section is a square of side 1".

Study Type: Static
Mesh Type: Beam mesh
Material Properties: Modulus of elasticity=3.0e7 psi, Poisson's ratio=0.28, Density=0.2782 lb/in^3



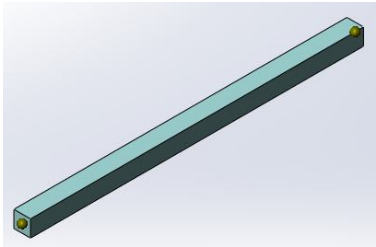
	Theory	SolidWorks Simulation
Maximum Displacement in the Y-direction (UY), inch	2.2256e-3	2.2223e-3



Analytical Solution:
$UY = (3wL^4)/(24EI)$
where: w: Weight per unit length, L: Beam length, E: Modulus of Elasticity, and I: Area moment of inertia

Case D1-1
Opening the Part and Creating a Study

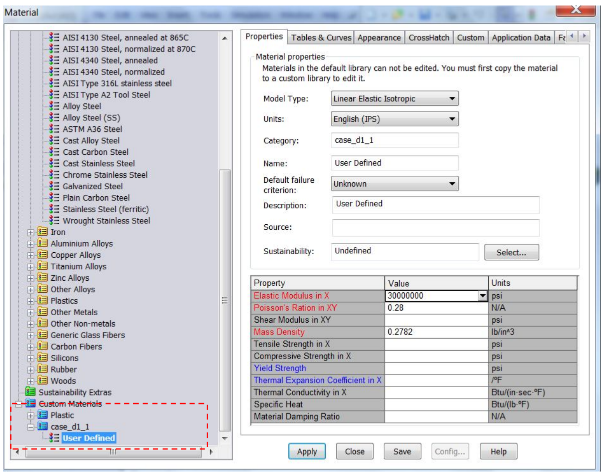
1. Open **Case_D1_1.SLDPRT**.
2. Click **New Study** (Simulation CommandManager).
3. In the PropertyManager:
   a. Type **Case D1-1** for name.
   b. Under **Type,** click **Static.**
4. Click OK.
5. In the Simulation study tree, right-click **Case_D1_1** and select **Treat as Beam.**



21

Case D1-1
Applying Material

1. Right-click the case study **Case_D1_1** and select **Apply/Edit Material…**.
2. Change the **Unit** to **English(IPS).**
3. Select **User Defined** under **Case_D1_1** in the folder **Custom Materials** and then assign the material properties as shown in previous slide.
4. Click **Apply** and then **Close.**



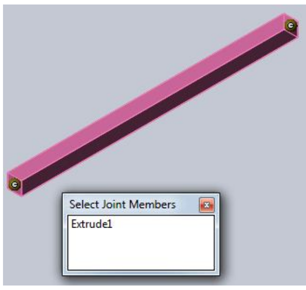
The software creates the study in the Simulation study tree. Note the check mark ✓ on the part in the study tree indicating that you assigned a material.

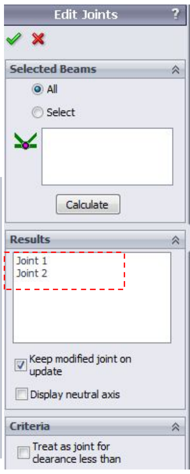
22

Case D1-1
Defining Joints

A joint is identified automatically at a free end of a beam. Fixtures are applied to joints only. To verify joints:

1. Right-click the **Joint group** folder and select **Edit**.
2. In the **Edit Joints PropertyManager** under Selected Beams, click **Calculate.** Two joints appear under **Results**
3. Right-click one of the joints in the graphics area to verify the extruded bodies that make up the joint. The members highlight in the graphics area.
4. Click **OK**.

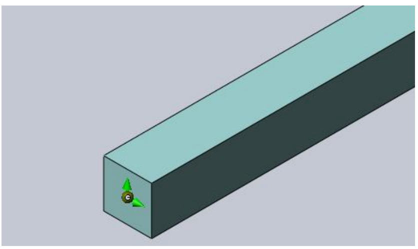


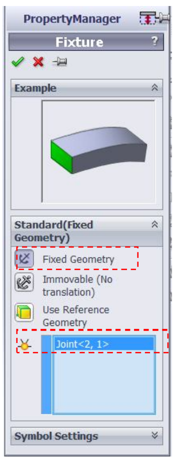


23

Case D1-1
Applying Fixtures

1. Right-click the **Fixtures** folder and select **Fixed Geometry**.
2. In the **PropertyManager**:
   - Select **Fixed Geometry** under **Standard (Fixed Geometry)**.
   - Select the joint shown for **Joints**.
3. Click **OK**.



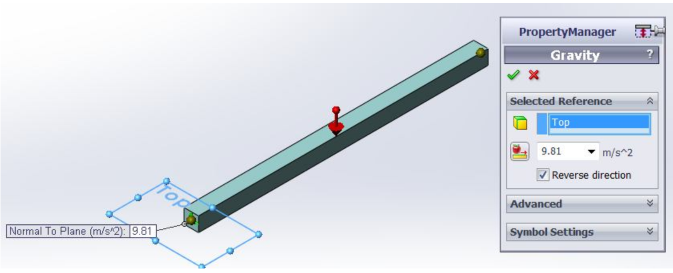


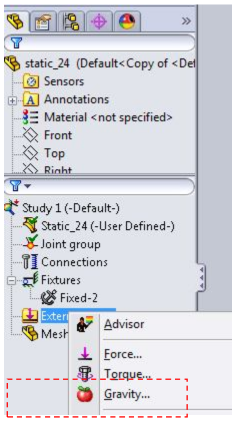
24

Case D1-1
Applying Gravity Load

Apply a gravity force to the part.

1. In the Simulation study tree, right-click **External Loads** and select **Gravity**.
2. In the **Gravity** windows under the **Selected reference**
   - Select **Top plane** for the Face, Edge, Plane for **Direction**.
   - Type 9.81 for the Apply Earth's gravity normal to plane.
3. Click **OK**.





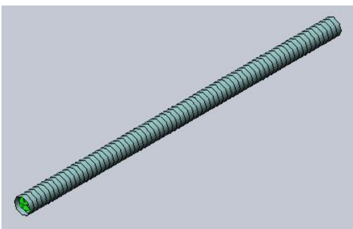
Case D1-1
Meshing the Model and Running the Study

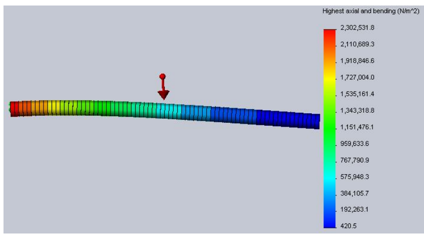
To mesh the model and run the study:

1. In the Simulation study tree, right-click **Mesh** and select **Create Mesh**.

Notice that each beam is divided into a number of beam elements.

2. In the Simulation study tree, right-click the study (**Case_D1_1**) and select **Run**.

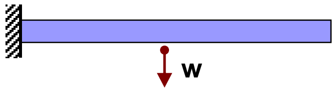


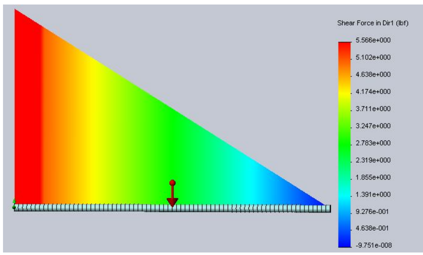


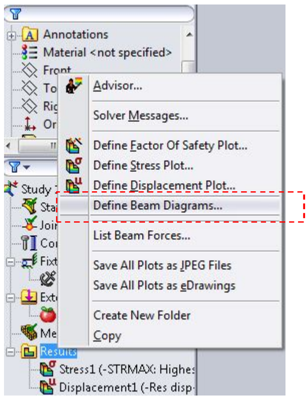
Note: For 1-D simulation (Beam element), the meshing is automatically generated and whether no tools for specify the mesh size.

Case D1-1
Viewing the Shear Diagram

1. Right-click the **Results** folder and select **Define Beam Diagrams**.
2. In the PropertyManager, under **Display**:
   Select **Shear Force in Dir 1** in **Component**.
   Why shear in direction 1?
   Select **lbf** in **Units**.
3. Click **OK**.
4. Change the view orientation to *Left.



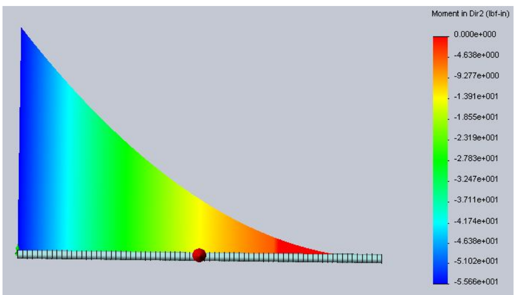


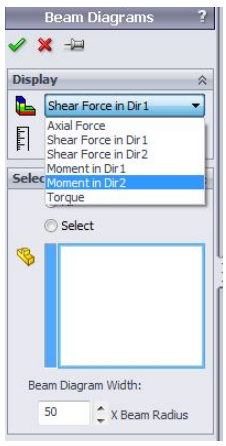


27

Case D1-1
Viewing the Moment Diagram

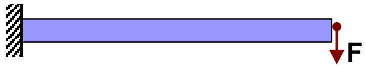
1. Right-click the **Results** folder and select **Define Beam Diagrams**.
2. In the PropertyManager, under **Display**:
   Select **Moment in Dir 2** in **Component** .
   Why moment in direction 2?
   Select **lb-in** for **Units**.
3. Click OK.





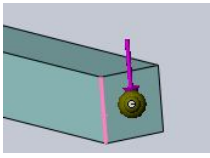
28

Case D1-2
Point Load



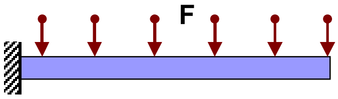
Apply a Point Load to the part.

1. In the Simulation study tree, right-click **External Loads** and select **Force**.
2. In the **Force/Torque** windows under the **Selection**
   - Select **Joint** for apply the force.
   - Select reference **Edge** for the **Face, Edge, Plane for Direction**.
3. In the **Units** windows select **SI** unit.
4. In the **Force** windows select **1 N** force and **Reverse direction**.
3. Click **OK**.



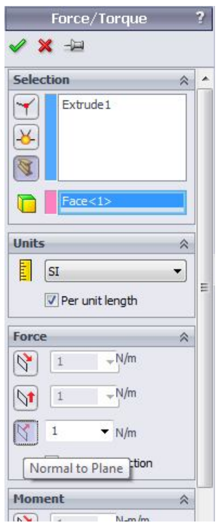
29

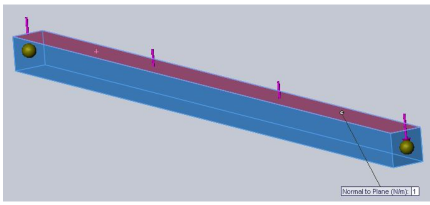
Case D1-3
Distributed Load



Apply a Point Load to the part.

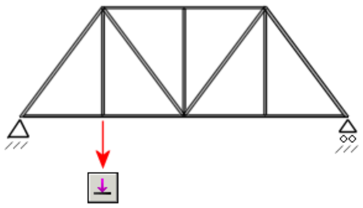
1. In the Simulation study tree, right-click **External Loads** and select **Force**.
2. In the **Force/Torque** windows under the **Selection**
   - Select **Beam** for apply the force.
   - Select reference **Face** for the **Face, Edge, Plane for Direction**.
3. In the **Units** windows select **SI** unit and **Per unit length**.
4. In the **Force** windows select **1 N** force apply on the direction of **Normal to Plane**..
3. Click **OK**.





30

Case D1-4
1-D Truss Analysis

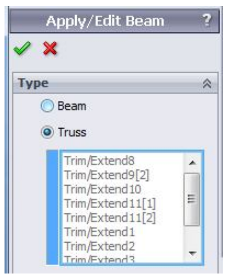


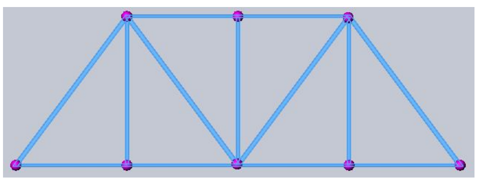
The model is loaded and restrained as shown:

Objectives:
- Creating a study with beam mesh
- Defining trusses and joints
- Adding loads and fixtures to truss joints
- Meshing and running the beam mesh study
- Viewing displacement and axial stress

Case D1-4
Opening the Part and Creating a Study

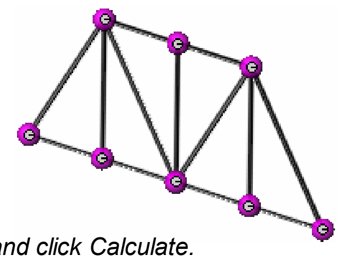
1. Open file **Case_D1_4.SLDPRT.**
2. Click **New Study** (Simulation CommandManager).
3. In the PropertyManager:
   a. Type **Truss**.
   b. Under **Type, click Static.**
4. Click **OK**.
5. In the **Simulation study tree**, under the **Truss** folder, select all nine structural members.
6. Right-click and select **Edit Definition**.
7. In the **PropertyManager**, under Type, select **Truss**.
8. Click **OK**.





Case D1-4
Applying Material/ Defining Joints

1. Right-click the **Truss** folder and select **Apply Material to All Bodies**.
2. Assign a custom material with the following properties:
   - Elastic modulus = 3e7 psi
   - Poisson's ratio = 0.3
   - Mass density = 0.28 lb/in^3
   - Yield strength= 90,000 psi



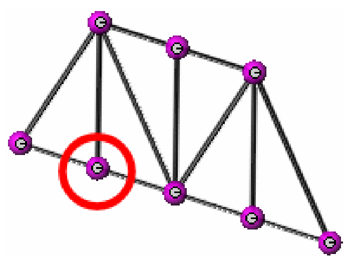
To define Joints:
1. Right-click **Joint Group** and select **Edit**.
2. Eight joints appear under **Results**.

If there are missing or invalid joints, select All and click Calculate.

3. Right-click one of the joints in the graphics area to verify the structural members that make up the joint. The members highlight in the graphics area.
3. Click **OK**.

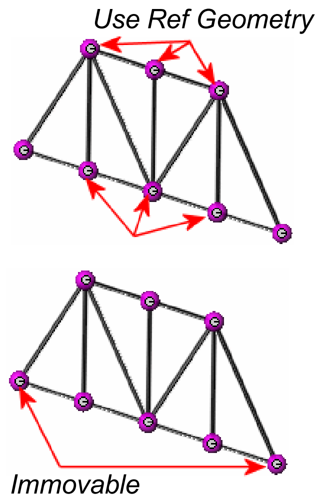
Case D1-4
Adding a Force

1. Right-click the **External Loads** folder and select **Force** .
2. In the PropertyManager, under **Selection:**
   - Click **Joints** .
   - Select the joint shown for **Joints**.
3. Select **Front** from the flyout FeatureManager tree for **Face, Edge, Plane for Direction** .
4. Under **Unit,** select **English (IPS).**
5. Under **Force**:
   - Click Along **Plane Dir 2** .
   - Type **64000**.
   - Select **Reverse direction** so the force points downward.
6. Click **OK**.



Case D1-4
Adding a Fixtures

Add fixtures to prevent out-of-plane motion and to fix the bottom corners of the truss system.
To prevent out-of-plane motion:
1. Right-click the **Fixtures** folder and select **Fixed Geometry**.
2. In the PropertyManager, under **Standard (Fixed Geometry):**
   - Select **Use Reference Geometry** .
   - Select the six joints shown for **Joints** .
   - Select **Front** from the flyout FeatureManager design tree for **Face, Edge, Plane, Axis for Direction** .
3. Under **Translations,** click **Normal to Plane** and ensure the value is zero.
4. Click **OK**.
5. To make the support joints immovable:
Add an **Immovable** (No translation) restraint to the two joints shown.

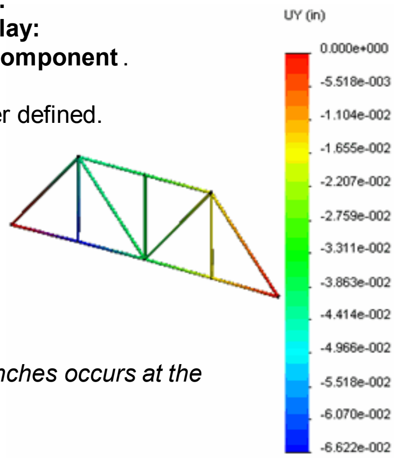


Case D1-4
Meshing the Model and Running the Study

To mesh the model and run the study:
1. In the Simulation study tree, right-click **Mesh** and select **Create Mesh**.
Notice each structural member is represented by a single truss element.
2. In the Simulation study tree, right-click the study and select **Run**.

Case D1-4
Viewing Displacement

1. In the Simulation study tree, right-click the **Results** folder and select **Define Displacement Plot .**
2. In the PropertyManager, under **Display:**
   - Select **UY: Y Displacement in Component** .
   - Select **in** for **Units** .
3. Under **Deformed Shape**, select User defined.
4. Click **OK**.

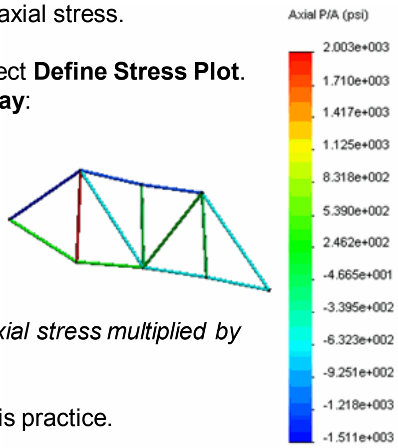


The maximum displacement of 0.066 inches occurs at the highlighted joint.

37

Case D1-4
Viewing Axial Stress

The only stress in a truss is the uniform axial stress.
To plot axial stress:
1. Right-click the **Results** folder and select **Define Stress Plot**.
2. In the Property Manager, under **Display**:
   - Select **psi** in **Units** .
   - Select **Axial** in **Beam stress**.
3. Click **OK**.

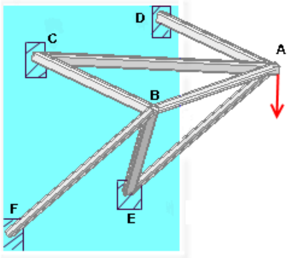


The force in each truss is equal to the axial stress multiplied by its cross-sectional area.

Congratulations! You have completed this practice.

38

Case D1-5
1-D Truss with Vertical Load



Description

A 3D truss consists of two panels ABCD and ABEF, which are attached to a vertical wall at points C, D, E, and F. The panel ABCD is in a horizontal plane. Calculate the axial force of bar AC due to a vertical load of 1,000 lb acting on joint A.

- Study Type: Static.
- Mesh Type: Beam mesh.
- Material Properties: Modulus of elasticity = 30×10^6 psi.
- Cross-Sectional Area: 2.089 in^2 (All structural members have the same cross-sectional area).

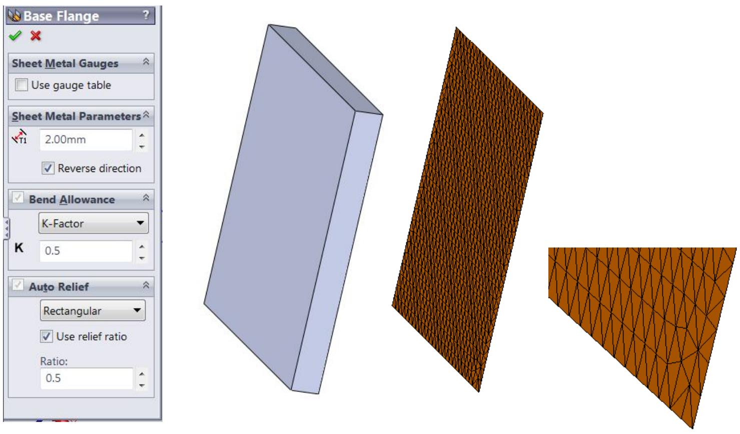
Hint: All the structural members are treated as truss elements.

	Theory	SolidWorks Simulation
Axial force of member AC (lb)	56.0	55.92

Ref: Timoshenko S.P. and Young D. H., "Theory of Structures", McGraw-Hill, New York, 1965, pp. 330-331.

39

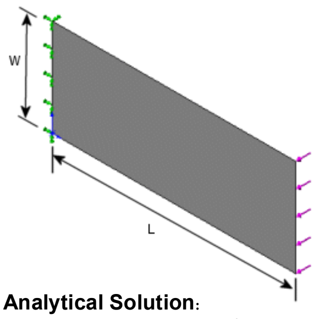
Preparing 2-D CAD File
Sheet Metal



40

Case D2-1
Reactions/Deflections of a Cantilever Beam

Description
- The beam subjected to an 8 lb force acting on the free end.
- The dimensions are: L = 10" and W = 4".



Study Type: Static.
Mesh Type: Shell mesh and Beam mesh.
Shell Parameters: Shell thickness = 1' - Thin formulation.
Meshing Parameters: Use a Global Size of 1' for Shell mesh.
Material Properties: Modulus of elasticity=3.0e7 psi, Poisson's ratio=0.3, and density = 0.28 ib/in3.

	Theory	SolidWorks Simulation Shell Mesh	SolidWorks Simulation Beam Mesh
Maximum deflection at tip, in	2.667e-4	2.574e-4	2.667e-4
Total reaction force, lb	8	8	8

Analytical Solution:
Deflection at tip: UY = $(FL^3)/(3EI)$
where: F: Load, L: Beam length, E: Modulus of Elasticity, and I: Area moment of inertia

41

Case D2-1
Creating a Shell Study/ Assigning Material

1. Open file "**Case_D2_1.SLDPRT**".
2. Click **New Study** (Simulation CommandManager)
3. In the **PropertyManager** under Name, type **Case_D2_1**.
4. Under **Type**, click **Static**.
5. Click **OK**.

The midsurface of the sheet metal part is shown. The software automatically meshes surface bodies with shells.

Assign material properties to the part.
1. In the Simulation study tree, right-click the **sheet** icon and select **Apply/Edit Material** .

The **Material Editor** opens.
2. Under **Select material source**, do the following:
   - Click **From library files** and select **Custom Materials** from the menu.
   - In folder **Case_D2_1** select **User Defined.**
   - Assign the material properties as shown in previous slide (Modulus of elasticity=3.0e7 psi, Poisson's ratio=0.3, and density = 0.28 ib/in3).
3. Click **OK**.

42

Case D2-1
Applying Fixed Restrains

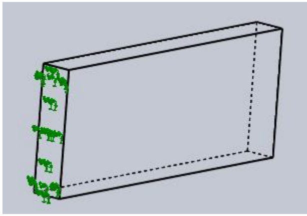
Fix the three holes of the bracket.
1. In the Simulation study study tree, right-click **Fixtures** and select **Fixed Geometry**.

The **Fixture** PropertyManager appears.

2. In the graphics area, click the edges of the three holes as shown in the figure.

Face<1> appear in the **Faces, Edges, Vertices for Fixture** box.

3. Click **OK**.

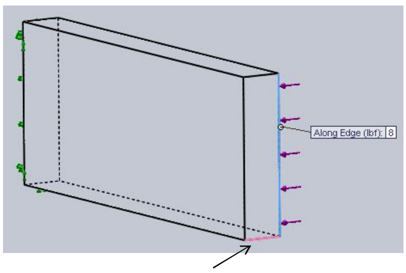


Note: For shell parts, **Fixed Geometry** sets translations and rotations to zero, while **Immovable** sets only translations to zero. **Fixed Geometry** and **Immovable** are similar for studies with solid parts.

43

Case D2-1
Applying Force

Apply a 8.0 lb force to the free end of the part.
1. In the Simulation study tree, right-click **External Loads** and select **Force**.
2. In the **PropertyManager**, under **Type**:
   - Select **Force** and **Select direction.**
   - Select the **Edge** for **Faces and Shell Edges for Forces** and select the reference edge for direction the force
3. Under **Force Value**
   - Set **Units** to **psi**
   - Type **8.0** in the **Force value** box
4. Click **OK**.

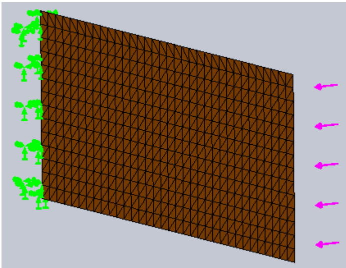


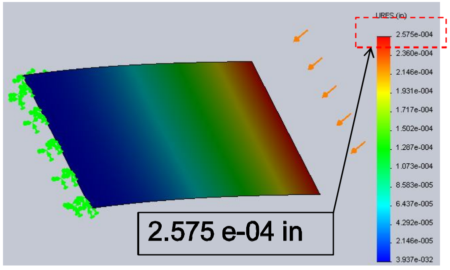
Reference edge for direction the force

44

Case D2-1
Meshing the Part/ Running the Study

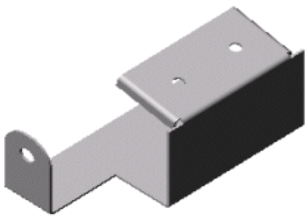
1. In the Simulation study tree, right-click **Mesh** and select **Create Mesh**
2. Click **OK** to accept the values.
3. Rotate the model to verify that all the top faces (grey) are on the same side. The bottom faces (orange) are on the opposite side.
4. In the Simulation study tree, right-click **ShellStudy** and select **Run**.



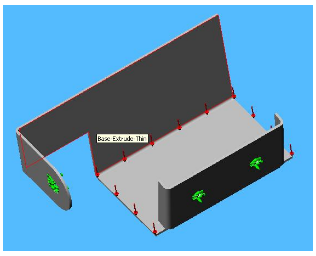


Try to solve the problem with using Beam mesh and then compare with reference result.

Case D2-2
Analysis of a Sheet Metal Part



Objectives:
- Creating a shell study using the midsurfaces option
- Choosing a solver
- Aligning shell faces
- Viewing stress results on shell models



Descriptions:
- English IPS Unit
- 0.15' thickness & alloy steel material
- 1.0 psi of pressure
- Restrain the three holes with the Fixed type

Case D2-2
Creating a Shell Study

To create a shell study:

1. Open file "**Case_D2_2.SLDPRT**".
2. Click **New Study** (Simulation CommandManager)
3. In the **PropertyManager** under Name, type **Case_D2_2**.
4. Under **Type**, click **Static**.
5. Click **OK**.

The midsurface of the sheet metal part is shown. The software automatically meshes surface bodies with shells. To view the solid body, rollback Body-Delete in the **FeatureManager** design tree.

47

Case D2-2
Assigning Material

Assign Alloy Steel from the SolidWorks material library to the part.

1. In the Simulation study tree, right-click the **Case_D2_2** and select **Apply/Edit Material** .

The **Material Editor** opens.

2. Under **Select material source**, do the following:
   - Click **From library files** and select **SolidWorks Materials** from the menu.
   - In the material tree, click the plus sign next to the **Steel** and select **Alloy Steel**.

The name of selected material appears in the **Name** box, and the mechanical properties of Alloy Steel appear in the material properties table.

3. Click **OK**.

48

Case D2-2
Defining Shell Thickness

1. In the Simulation study tree, right-click the **Case_D2_2** and select **Edit Definition**.
2. Set **Unit** to **in** and type **0.15** for **Shell thickness** .
3. Click **OK**.

49

Case D2-2
Applying Fixed Restrains

Fix the three holes of the bracket.

1. In the Simulation study study tree, right-click **Fixtures** and select **Fixed Geometry**.

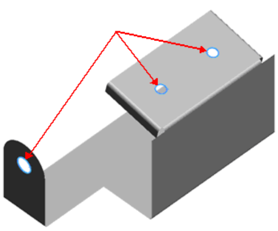
The **Fixture** PropertyManager appears.

2. In the graphics area, click the edges of the three holes as shown in the figure.

Edge<1>, Edge<2>, and Edge<3> appear in the **Faces, Edges, Vertices for Fixture** box.

3. Click **OK**.

Note: For shell parts, **Fixed Geometry** sets translations and rotations to zero, while **Immovable** sets only translations to zero. **Fixed Geometry** and **Immovable** are similar for studies with solid parts.

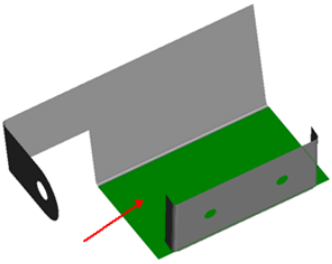


50

Case D2-2
Applying Pressure

Apply a 1.0 psi pressure to the face of the part.

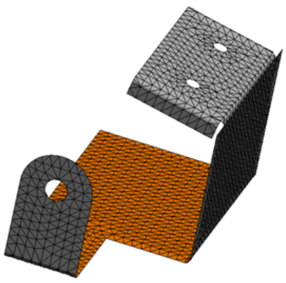
1. In the Simulation study tree, right-click **External Loads** and select **Pressure** .
2. In the **PropertyManager**, under **Type**:
   - Select **Normal to selected face**.
   - Select the face shown for **Faces for Pressure** .
3. Under **Pressure Value**
   - Set **Units** to **psi**
   - Type **1.0** in the **Pressure value** box
4. Click **OK**.



51

Case D2-2
Meshing the Part/ Running the Study

1. In the Simulation study tree, right-click **Mesh** and select **Create Mesh**
2. Click **OK** to accept the values.
3. Rotate the model to verify that all the top faces (grey) are on the same side. The bottom faces (orange) are on the opposite side.
4. In the Simulation study tree, right-click **Case_D2_2** and select **Run**.



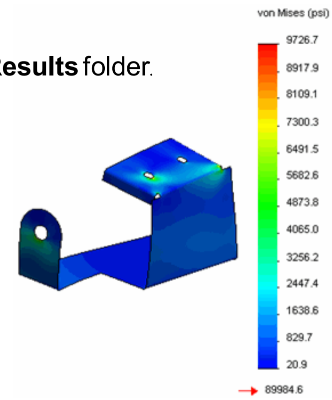
52

Case D2-2
Viewing Stress Result on Top Face

It is important to view stress results on both top and bottom faces of the shell model.

To view equivalent stresses on top faces:

1. In the Simulation study tree, open the **Results** folder.



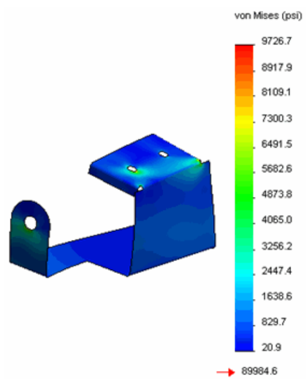
2. Double-click **Stress (-von Mises-)** to display the plot.
3. Rename the plot to **Top von Mises** in the Simulation study tree.

By default, the equivalent stress plot on the top shell faces is shown.

Case D2-2
Viewing Stress Result on Bottom Faces

To view equivalent stresses on bottom faces:

1. In the Simulation study tree, right-click the **Results** folder and select **Define Stress Plot.**



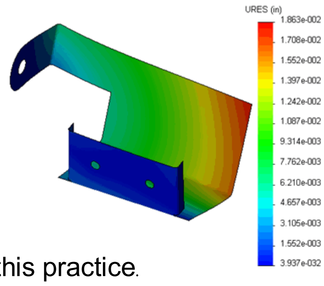
2. In the PropertyManager, under **Display**:
   - 2.1 Set **Units** to **psi**.
   - 2.2 Set **Shell face** to **Bottom**.
3. Click **OK**.
4. Rename this plot to **Bottom von Mises** in the Simulation study tree.
5. Double-click **Bottom von Mises**.

Note that von Mises stresses at the top and bottom faces are slightly different.

Case D2-2
Viewing Resultant Displacement

Displacements for top and bottom faces are assumed equal.
To view resultant displacements:

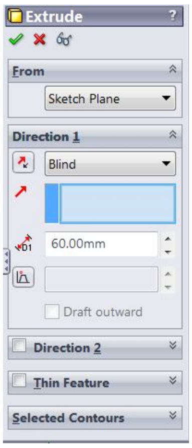
1. In the Simulation study tree, open the **Results** folder.
2. Double-click **Displacement** (-**Res disp**-) to display the plot. If the plot does not exist, create this plot.
3. Rotate the model as shown in the figure.
4. Save the model and close the document.

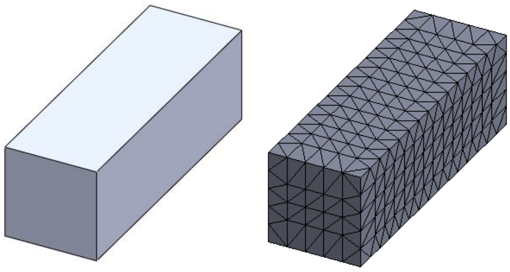


Congratulations! You have completed this practice.

3-D Element Case Studies
Static Analysis of Solid Part

Preparing 3-D CAD File
Solid Model



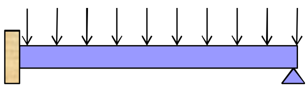


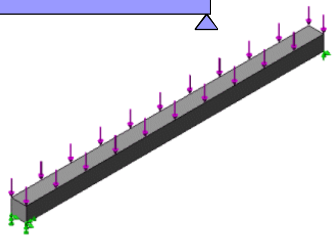
57

Case D3-1
Deformation of a Uniformly Loaded Beam

Description
Determine the maximum displacement in Y-direction of a uniformly loaded beam with:
-Fixed support at one end and a simple support at the other end.
- Length of the beam is 20"
- Beam section is a square of side 1".
- Distributed load of 125 lbf.

Study Type: Static.
Mesh Type: Solid mesh and Beam mesh.
Material Properties: Modulus of elasticity = 3.0e7 psi, Poisson's ratio=0.28, and mass density=0.28 lb/in3





	Theory	SolidWorks Simulation Solid Mesh	SolidWorks Simulation Beam Mesh
Maximum Displacement in the Y-direction (UY), inch	1.733e-3	1.755e-3	1.733e-3

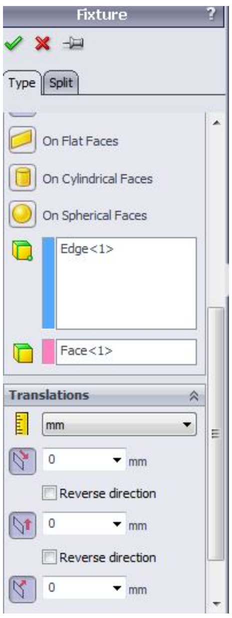
Analytical Solution:
$UY_{max} = WL^4/(184.6EI)$
where: W: Uniform load, L: Beam length, E: Modulus of elasticity, and I: Moment of inertia

58

Case D3-1
Creating a Part Study

Open the file **"Case_D3_1.SLDPRT.**
To assign Alloy Steel from the SolidWorks material library:
1.Right-click the case study **Case_D1_1** and select **Apply/Edit Material….**
2. Change the **Unit** to **English(IPS).**
3. Select **User Defined** under **Case_D1_1** in the folder **Custom Materials** and then assign the material properties as shown in previous slide.
4. Click **Apply** and **Close**.

Case D3-1
Applying Fixed Restrains



1. Click the down arrow on **Fixtures** and select **Fixed Geometry**, or right-click **Fixtures** in the study tree and select **Fixed Geometry**.
2. In the graphics area, click the face shown in the fig. Face<1> appear in the **Faces, Edges, Vertices for Fixture** box.
3. Click **OK**.
4. Apply the simple condition to the other end of the beam.
   - Select **Advanced Fixtures** and then **Use Reference Geometry**
   - Select **Edge<1>** for **Face, Edge, Vertices for Fixtures**
   - Select **Face<1>** for **Direction**
   - For **Translation,** turn on all **Along Direction Dir**

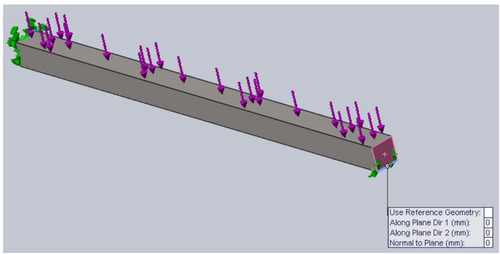
The software fixes the faces of the selected face and creates an icon named Fixture-1 in the Fixtures folder of the Simulation study tree.

Case D3-1
Applying Force

1. Right-click **External Loads** in the Simulation study tree and select **Force**.
2. In the PropertyManager, on the **Type** tab under **Type**, click **Selected direction**.
3. In the graphics area, select the face as shown for **Faces for Pressure**
4. Select **Edge<1>** for reference the force direction
4. Under **Force Value**, select **lb** in **Units** then type 125 for **Force value**.

If you change the units after typing a value, the software converts the value to the new units.

5. Click **OK**.

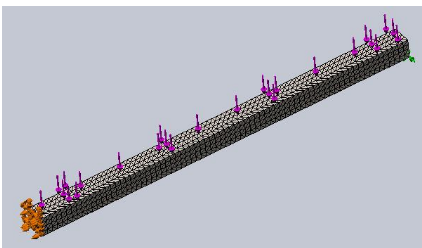


Case D3-1
Meshing/Running the Analysis

1. In the Simulation study tree, right-click **Mesh** and select **Create Mesh**, or click the down arrow on **Run** (Simulation CommandManager) and select **Create Mesh.**
2. Click **OK** to accept the values.

Meshing starts and the **Mesh Progress** window appears. After meshing is completed, the meshed model appears in the graphics area.

3. Click **Run** (Simulation CommandManager).



To display the mesh information:

1. In the Simulation study tree, right-click **Mesh** and select **Details**.
2. Close the **Mesh Details** list box.

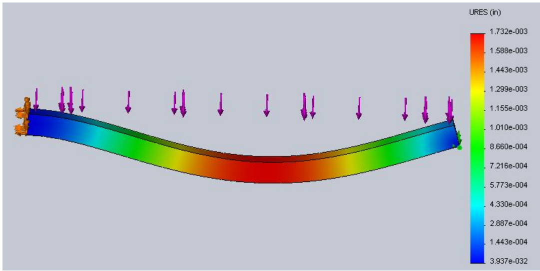
To hide or show the mesh:

Click **Show/Hide Mesh** on the Simulation toolbar.

Case D3-1
Viewing Resultant Displacement

In the Simulation study tree, open the **Results** folder.
Double-click **Displacement** (-**Res disp**-) to display the plot.
To animate the resultant displacement plot:

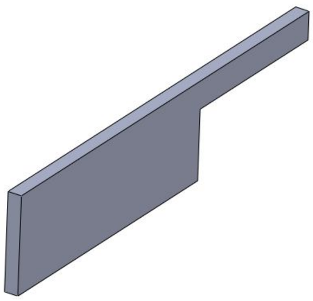
1. Click **Plot Tools** and select **Animate**.
2. Click **Stop** to stop the animation.
3. Click **Loop** then click **Animate** to start the animation.



The animation plays in a continuous looping pattern. It will play from start to end, then start to end again, and continue to repeat.

63

Case D3-2
Analysis of a Simple 3-D Part-1



Objectives:
- Assigning material to the part using SolidWorks Materials Editor
- Creating a static analysis study
- Applying restraints and force
- Setting meshing options and meshing the part
- Running the study
- Viewing basic results of static analysis

64

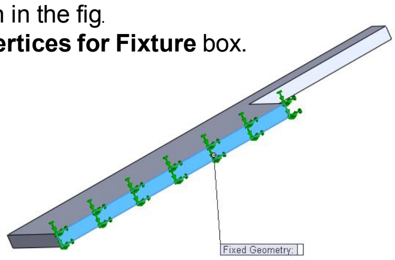
Case D3-2
Creating a Study/ Applying Material Properties

1. Open the file **"Case_D3_2.SLDPRT.**
2. Click **New Study** (Simulation CommandManager).
3. In the PropertyManager, under **Name** type **Case_D3_2**.
4. Under **Type**, click **Static**.
5. Click **OK**.

To assign material properties:
1.Click **Edit Material** (SolidWorks Standard toolbar).
2.In the left pane, click the plus sign next to **SolidWorks Materials**, then click the plus sign next to **Steel** and select **Alloy Steel**.
Mechanical properties of Alloy Steel appear in the **Properties** tab.
3.Click **Apply** and **Close**.
The name of the assigned material appears in the FeatureManager tree.

Case D3-2
Applying Fixed Restrains

For static analysis, we must apply adequate fixed restraints to stabilize the model. In this example, a lower face of the part are fixed.
1. Click the down arrow on **Fixtures** and select **Fixed Geometry**, or right-click **Fixtures** in the study tree and select **Fixed Geometry**.
The **Fixture** PropertyManager appears.
2. In the graphics area, click the face shown in the fig.
Face<1> appear in the **Faces, Edges, Vertices for Fixture** box.
3. Click **OK**.



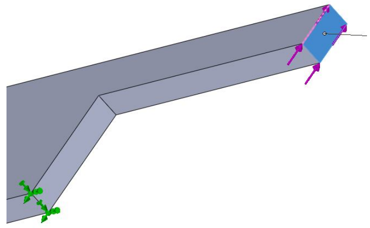
The software fixes the faces of the selected face and creates an icon named Fixture-1 in the Fixtures folder of the Simulation study tree.

Case D3-2
Applying Force

1. Right-click **External Loads** in the Simulation study tree and select **Force**.
2. In the PropertyManager, on the **Type** tab under **Type**, click **Selected direction**.
3. In the graphics area, select the face as shown for **Faces for Pressure**
4. Select **Edge<1>** for reference the force direction
4. Under **Force Value**, select **N** in **Units** then type **200** for **Force value**.

If you change the units after typing a value, the software converts the value to the new units.

5. Click **OK**.



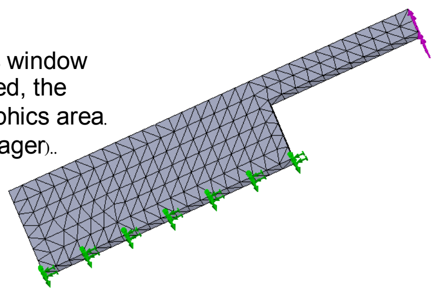
The software applies 200 N pressure and creates an icon named **Force-1** in the **External Loads** folder of the Simulation study tree.

Case D3-2
Meshing/Running the Analysis

1. In the Simulation study tree, right-click **Mesh** and select **Create Mesh**, or click the down arrow on **Run** (Simulation CommandManager) and select **Create Mesh.**
2. Click **OK** to accept the values.

Meshing starts and the **Mesh Progress** window appears. After meshing is completed, the meshed model appears in the graphics area.

3. Click **Run** (Simulation CommandManager).



To display the mesh information:
1. In the Simulation study tree, right-click **Mesh** and select **Details**.
2. Close the **Mesh Details** list box.

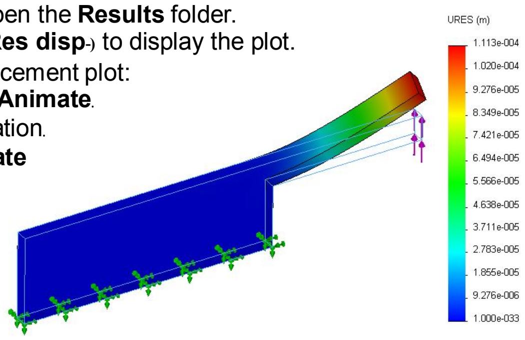
To hide or show the mesh:
Click **Show/Hide Mesh** on the Simulation toolbar.

Case D3-2
Viewing Resultant Displacement

In the Simulation study tree, open the **Results** folder.
Double-click **Displacement** (-**Res disp**-) to display the plot.

To animate the resultant displacement plot:

1. Click **Plot Tools** and select **Animate**.
2. Click **Stop** to stop the animation.
3. Click **Loop** then click **Animate** to start the animation.

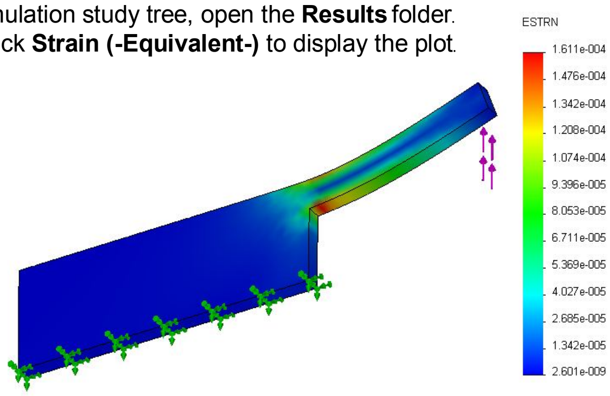


The animation plays in a continuous looping pattern. It will play from start to end, then start to end again, and continue to repeat.

Case D3-2
Viewing Element Strains

To plot the equivalent element strains:

1. In the Simulation study tree, open the **Results** folder.
2. Double-click **Strain (-Equivalent-)** to display the plot.

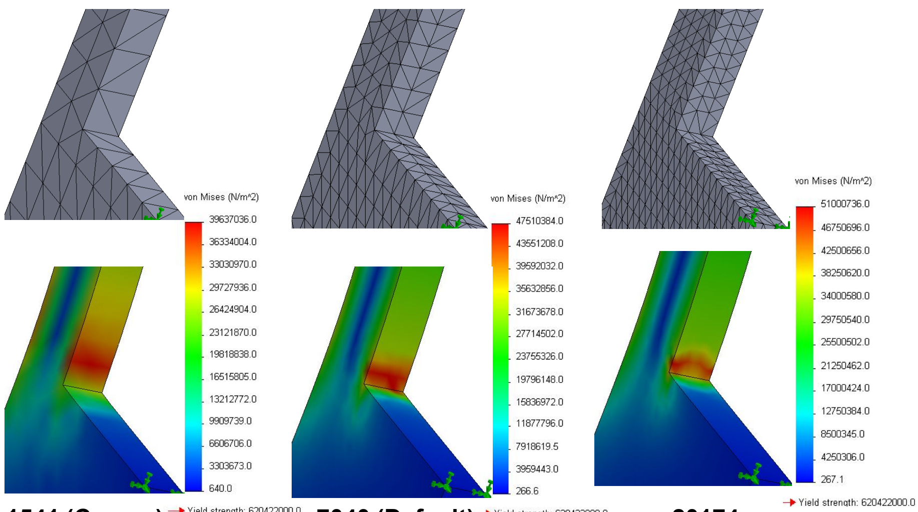


Case D3-2
Viewing von Mises Stresses

To plot von Mises stresses:
1. In the Simulation study tree, open the **Results** folder.
2. Double-click **Stress (-von Mises-)** to display the plot.

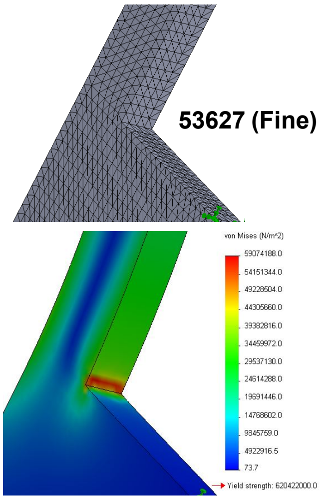
To view the stress plot in a different unit system, right-click the plot icon and click **Edit Definition**. Under **Display**, set **Units** to the desired system of units and click .

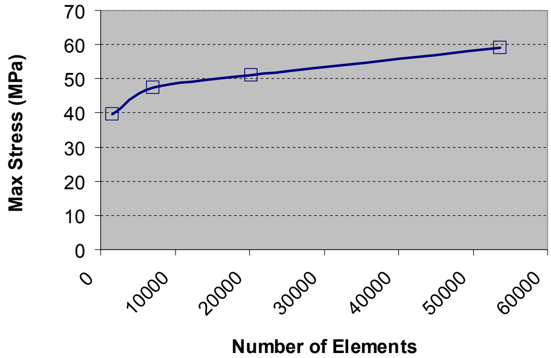
Case D3-2
Effect of Mesh Density



1541 (Coarse) **7040 (Default)** **20174**

Case D3-2
Effect of Mesh Density

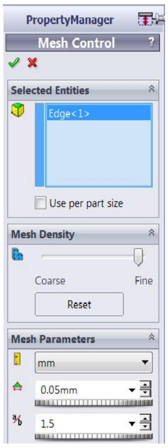


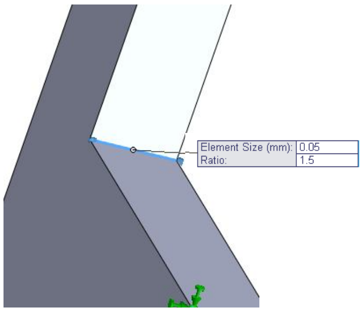


Total elements	1541	7040	20174	53627
Maximum stress (Mpa)	39.6	47.5	51.0	59.1

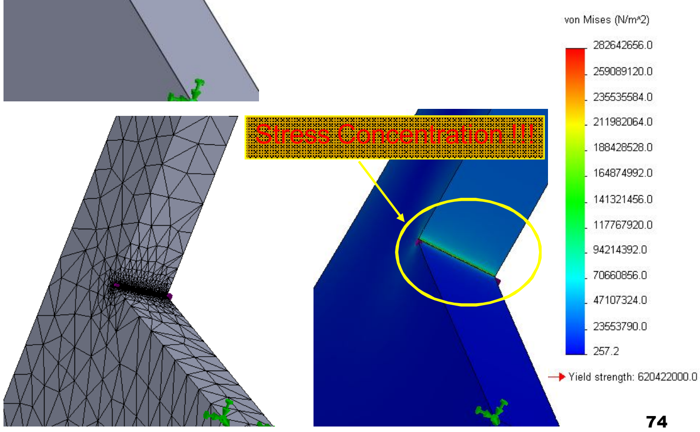
73

Case D3-2
Mesh Controlling



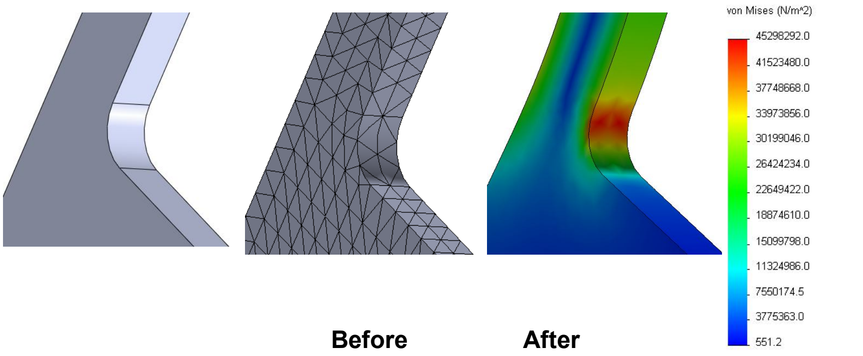


Total elements 20169
Max Stress 282.6 MPa !!



74

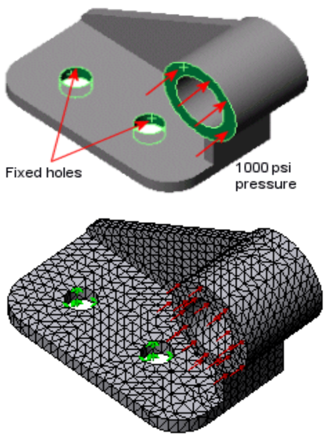
Case D3-2
Modifying the Part



	Before	After
Total Elements	**7040**	**7628**
Max Stress (MPa)	**47.5**	$\boxed{\textbf{45.3}}$

More safe !!

Case D3-3
Analysis of a Simple 3-D



Objectives:
- Assigning material to the part using SolidWorks Materials Editor
- Creating a static analysis study
- Applying restraints and pressure load
- Setting meshing options and meshing the part
- Running the study
- Viewing basic results of static analysis

Case D3-3
Creating a Part Study

Open the file **"Case_D3_3.SLDPRT.**
To assign Alloy Steel from the SolidWorks material library:
1.Click **Edit Material** (SolidWorks Standard toolbar).
The **Material Editor** appears.
2.In the left pane, click the plus sign next to **SolidWorks Materials**, then click the plus sign next to **Steel** and select **Alloy Steel**.
Mechanical properties of Alloy Steel appear in the **Properties** tab.
3.Click **Apply** and **Close**.
The name of the assigned material appears in the FeatureManager tree .

77

Case D3-3
Creating a Static Analysis

To create a static study:
1. Click **New Study** (Simulation CommandManager).
2. In the PropertyManager, under **Name** type **Static-1**.
3. Under **Type**, click **Static** .
4. Click **OK**.

The software creates the study in the Simulation study tree. Note the check mark ✓ on the part in the study tree indicating that you assigned a material.

78

Case D3-3
Applying Fixed Restrains

For static analysis, we must apply adequate fixed restraints to stabilize the model. In this example, two holes at the base of the part are fixed.

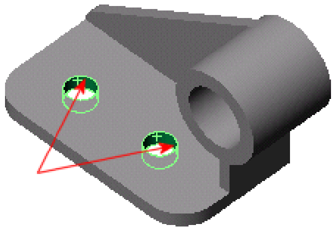
To fix the two holes:

1. Click the down arrow on **Fixtures** and select **Fixed Geometry**, or right-click **Fixtures** in the study tree and select **Fixed Geometry**.

The **Fixture** PropertyManager appears.

2. In the graphics area, click the faces of the two holes shown in the fig.
   Face<1> and Face<2> appear in the **Faces, Edges, Vertices for Fixture** box.
3. Click **OK**.

The software fixes the faces of the two holes and creates an icon named Fixture-1 in the Fixtures folder of the Simulation study tree.



79

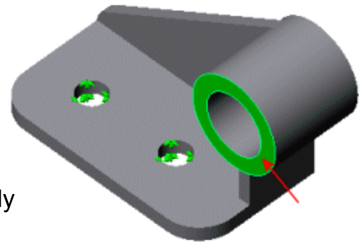
Case D3-3
Applying Pressure

1. Right-click **External Loads** in the Simulation study tree and select **Pressure**.
2. In the PropertyManager, on the **Type** tab under **Type**, click **Normal to selected face**.
3. In the graphics area, select the face as shown for **Faces for Pressure**
4. Under **Pressure Value**, select **psi** in **Units** then type **1000** for **Pressure value**

If you change the units after typing a value, the software converts the value to the new units.

5. Click **OK**.

The software applies 1000 psi pressure and creates an icon named **Pressure-1** in the **External Loads** folder of the Simulation study tree.



80

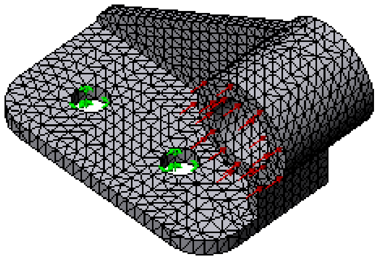
Case D3-3
Meshing/Running the Analysis

To mesh the part and run static analysis:

1. In the Simulation study tree, right-click **Mesh** and select **Create Mesh**, or click the down arrow on **Run** (Simulation CommandManager) and select **Create Mesh.**
2. Click **OK**.

Meshing starts and the **Mesh Progress** window appears. After meshing is completed, the meshed model

3. Click **Run** (Simulation CommandM



Note: The analysis runs and the **Results** folder appears in the Simulation study tree

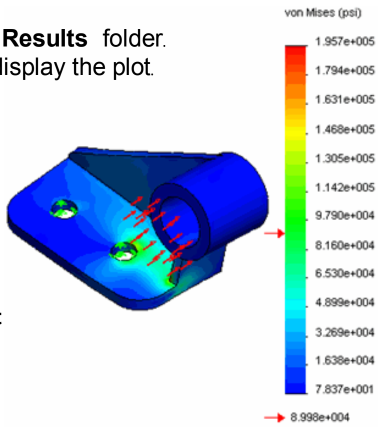
Case D3-3
Viewing von Mises Stresses

To plot von Mises stresses:

1. In the Simulation study tree, open the **Results** folder.
2. Double-click **Stress (-von Mises-)** to display the plot.

The stress plot is generated on the deformed shape. To illustrate the deformed shape, the software scales the maximum deformation to 10% of the diagonal of the bounding box of the model. In this case, the deformation scale is approximately 12.

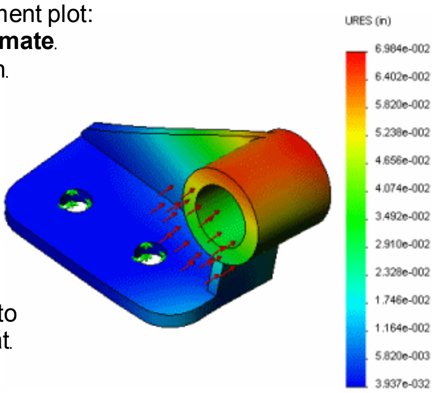
To view the stress plot in a different unit system, right-click the plot icon and click **Edit Definition**. Under **Display**, set **Units** to the desired system of units and click .



Case D3-3
Viewing Resultant Displacement

In the Simulation study tree, open the **Results** folder.
Double-click **Displacement** (-**Res disp**-) to display the plot.
To animate the resultant displacement plot:

1. Click **Plot Tools** and select **Animate**.
2. Click **Stop** to stop the animation.
3. Click **Loop** then click **Animate** to start the animation.

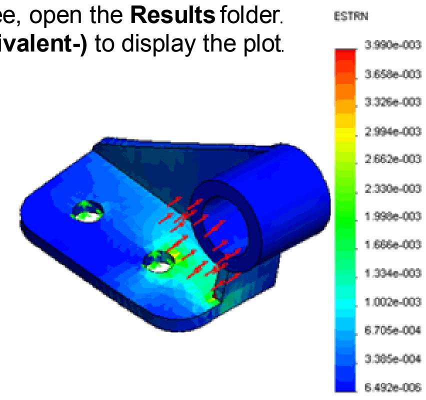


The animation plays in a continuous looping pattern. It will play from start to end, then start to end again, and continue to repeat.

Case D3-3
Viewing Element Strains

To plot the equivalent element strains:

1. In the Simulation study tree, open the **Results** folder.
2. Double-click **Strain (-Equivalent-)** to display the plot.



Case D3-3
Assessing the Safety of the Design

The Factor of Safety wizard helps you assess the safety of your design.
To view the Factor of Safety (FOS) distribution in the model:
1. In the Simulation study tree, right-click the **Results** folder and select **Define Factor of Safety Plot**.

The **Factor of Safety** PropertyManager appears.

2. In the PropertyManager, under **Step 1 of 3**, select **Max von Mises stress** in **Criterion** .
3. Click **Next** .
4. Under **Step 2 of 3**, select **Yield strength**.

Note that the elastic properties of the material of the part and the maximum von Mises stress are listed.

5. Click **Next**
6. Under **Step 3 of 3**, select **Factor of safety distribution**.
7. Click **OK**.

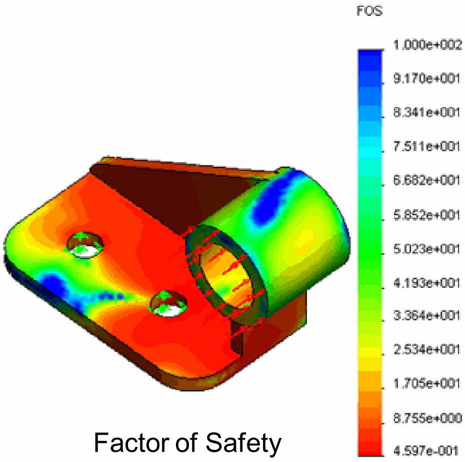
85

Case D3-3
Assessing the Safety of the Design

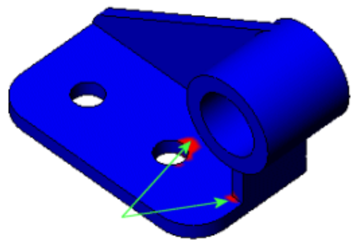
To plot critical regions of the part:
1. Click the down arrow on **Results** and select **New Plot**, **Factor of Safety.**
2. In the PropertyManager, under **Step 1 of 3**, select **Max von Mises stress** in **Criterion** .
3. Click **Next** .
4. Under **Step 2 of 3**, select **Yield strength**.
5. Click **Next**
6. Under **Step 3 of 3**:
   - Select **Areas below factor of safety**.
   - Type 1 for **Factor of safety**.
7. Click **OK**.

86

Case D3-3
Assessing the Safety of the Design



Factor of Safety



Critical Region

87

Case D3-3
Generating the Study Report

To generate a study report:

1. Click **Report** (Simulation CommandManager).
2. In the dialog box, select **Connector Definitions** in **Included sections** and click to move this to **Available sections**.

This excludes this section from the report.

3. Repeat step 2 for **Design Scenario Results** and **Sensor Results** to remove some more unused sections from the report.
4. Select **Description** in **Included sections**.
5. Under **Section properties**:
   - Select **Comments**.
   - Type **My first report** in the box.
6. Under **Document settings**:
   - Type **First Report** for **Report name**.
   - Select **Show report on publish**.
   - Select **HTML** for **Publish as**.

88

Case D3-3
Generating the Study Report

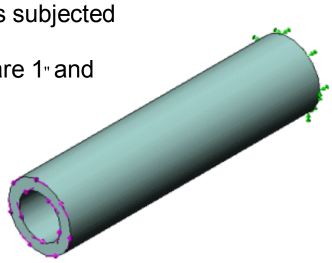
7. Click **Publish**.
The report displays in your default web browser. You can navigate through different sections of your report by clicking on the links at the top.
8. To close the report window, click .
To save the model and the analysis information in the part document:
Click **File**, **Save**.

Congratulations! You have completed your first analysis lesson.

Practices

Practices
Shear Stress in a Hollow Cylinder

Description:
- One end has fixed and the other end has subjected to a torque of 10 lb-in.
- The inner and outer radii of the cylinder are 1" and 1.5" respectively
- The length of the cylinder is 12".



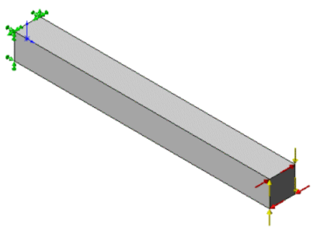
	Theory	SolidWorks Simulation
Maximum Shear Stress (TYZ), psi	2.351	2.362

91

Practices
Torsion of a Square Box Beam

Description
- Find the shear stress and the angle of twist for the square box beam shown in the figure.
- The free end is subjected to a 300 lb-in torque.
- The beam has a length of 1500".
- The beam cross section is a square with a side length of 150".

Study Type: Static.
Mesh Type: Shell mesh.
Shell Parameters: Shell thickness = 3'
- Thin formulation.
Meshing Parameters: Use a Global Size of 75'.
Material Properties: Modulus of elasticity=7.5 psi, Poisson's ratio=0.3.



	Theory	SolidWorks Simulation
Shear stress (τ xz), psi	0.0021365	0.002132
Rotation (θ), radians	0.01541	0.01536

92

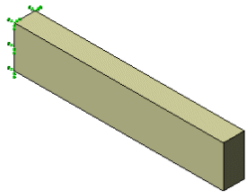
Practices
Bending of a Solid Beam

Description

- A 10" long cantilever beam has a rectangular cross section of 1" width and 2" height.
- Find the deflection of the free end under the effect of the following loads: an end moment of 2000 in-lb, and a shear force of 300 lbs.

Study Type: Static.
Mesh Type: Solid mesh and Beam mesh in separate studies.
Meshing Parameters: Use a Global Size of 2 in for Solid mesh.
Material Properties: Modulus of elasticity = 3.0e7 psi, Poisson's ratio = 0.
Results

Y-displacement at the free end (UY), in	Theory	SolidWorks Simulation Solid Mesh	SolidWorks Simulation Beam Mesh
End moment (Moment Study)	-0.005	-0.005006	-0.005
Shear force (Force Study)	0.005	0.005093	0.005



Analytical Solution:
$w_y = (2PL^3)/(6EI)$ (Force Study); $w_y = (ML^2)/(2EI)$ (Moment Study)
where: P: Shear force, M: End momemt, L: Beam length, E: Modulus of elasticity, I: Area moment of inertia

93