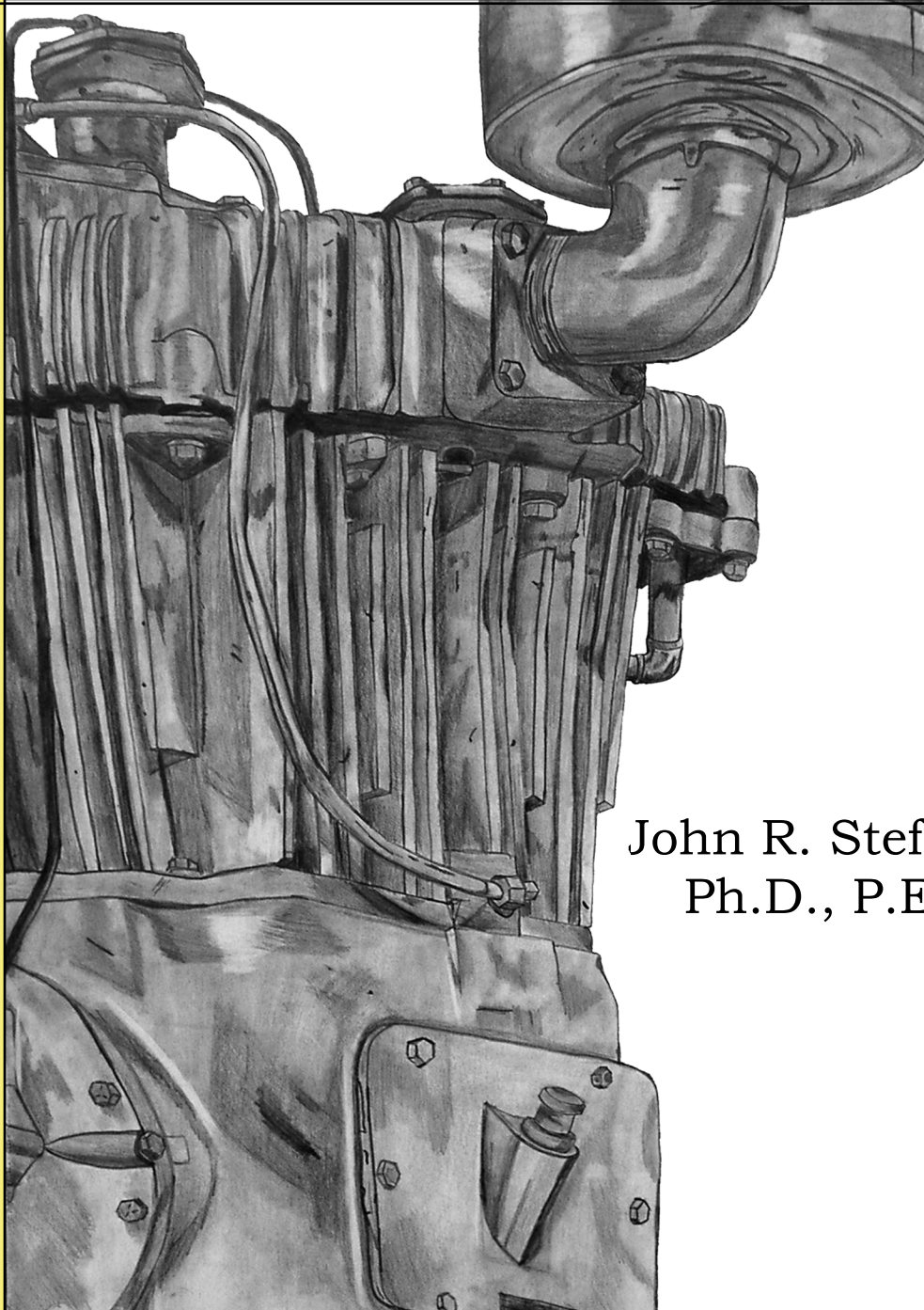


Analysis of Machine Elements

using SolidWorks Simulation 2009



John R. Steffen,
Ph.D., P.E.

SDC
PUBLICATIONS

Schroff Development Corporation
www.schroff.com

Better Textbooks. Lower Prices.

CHAPTER #2

CURVED BEAM ANALYSIS

This example, unlike the first, will lead you quickly through those aspects of creating a finite element Study with which you already have experience. However, where new information or procedures are introduced, additional details are included. For consistency throughout this text, a common approach is used for the solution of all problems.

Learning Objectives

In addition to software capabilities studied in the previous chapter, upon completion of this example, users should be able to:

- Use SolidWorks Simulation *icons* in addition to menu selections.
- Apply a *split line* to divide a selected face into one or more separate faces.
- Simulate *pin loading* inside a hole.
- Use **Design Checks** to determine the *safety factor* or lack thereof.
- Determine *reaction* forces acting on a finite element model.

Problem Statement

A dimensioned model of a curved beam is shown in Fig. 1. Assume the beam material is 2014 Aluminum alloy, and it is subject to a downward vertical force, $F_y = 3800$ lb, applied through a cylindrical pin (not shown) in a hole near its free end. The bottom of the curved beam is considered “fixed.” In this context, the *actual* fixed end-condition is analogous to that at the end of a cantilever beam where translations in the X, Y, Z directions and rotations about the X, Y, Z axes are considered to be zero. However, recall from Chapter 1 that

Fixture types within SolidWorks Simulation also depend on the type of element to which they are applied. Therefore, because solid tetrahedral elements are used to model this curved beam, **Immovable** restraints are used.

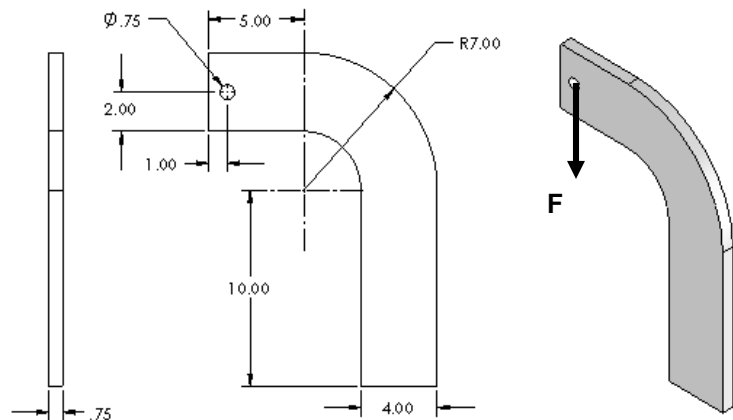


Figure 1 – Three dimensional model of curved beam.

Design Insight

Numerous mechanical elements occur in the shape of initially curved beams. Examples include: C-clamps, punch-press frames, crane hooks, and bicycle caliper brakes, to name a few. This example examines the stress at section A-A shown in Fig. 2. Section A-A is chosen because it is the furthest distance from the applied force F thereby creating reaction force $R = F$ and the maximum bending moment $M = F \cdot L$ at that location. Accordingly, classical equations for stress in a curved beam predict maximum stress at section A-A. In this example, the validity of this common assumption is investigated while exploring the additional capabilities of SolidWorks Simulation software listed above.

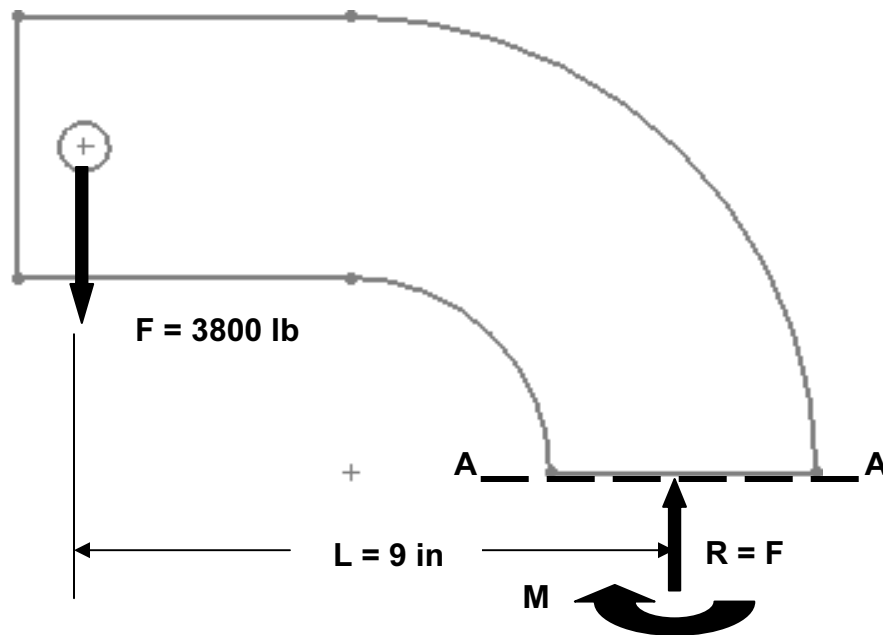


Figure 2 – Traditional free-body diagram of the upper portion of a curved beam model showing applied force F acting at a hole, and reactions $R = F$, and moment M acting on section A-A.

Creating a Static Analysis (Study)

1. Open SolidWorks by making the following selections. (Note: A “/” is used below to separate successive menu selection). Alternatively, double-click the SolidWorks icon on your desktop.

Start/All Programs/SolidWorks 2009 (or) Click the **SolidWorks** icon on your screen.

2. In the SolidWorks main menu, select **File / Open...** Then browse to the location where SolidWorks Simulations files are stored and select the file named “**Curved Beam**” then click [**Open**].

If a pop-up window appears and states: **The following documents will be converted when saved “Curved Beam.SLDPRT”**, click [OK] to close the window.

Reminder:

If you do not see **Simulation** listed in the main menu of the SolidWorks screen, click **Tools/Add-Ins...**, then in the **Add-Ins** window check ☒ **SolidWorks Simulation** in both the **Active Add-Ins** and the **Start Up** columns, then click **[OK]**. This action adds **Simulation** to the main menu.

Because one goal of this chapter is to introduce use of SolidWorks Simulation icons, three methods of displaying these icons are outlined below. If Simulation icons already appear on your screen, perform the following steps to investigate all display options available. Proceed as follows.

Display method #1

3. Right-click anywhere on the toolbar at top of the screen. This action opens the SolidWorks Simulation Command Manager menu shown in Fig. 3.
4. Within the Command Manager menu, select **Simulation**, shown at the arrow in Fig. 3. The Simulation toolbar opens; a partial view of it appears in Fig. 4. This toolbar is typically located beneath the main menu. Until a Study is initiated, all icons but one are grayed out (inactive).

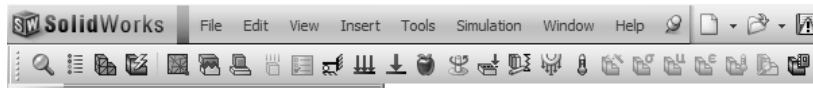


Figure 4 – SolidWorks Simulation icons displayed by selecting **Simulation**.

Additional toolbars may be displayed on your screen. However, the main menu and Simulation toolbars are the focus of Fig. 4, therefore, other toolbars have been removed.

Toolbars can be moved to different locations on the screen. Reposition toolbars by clicking-and-dragging the “handles” located at the left or upper end of each toolbar.

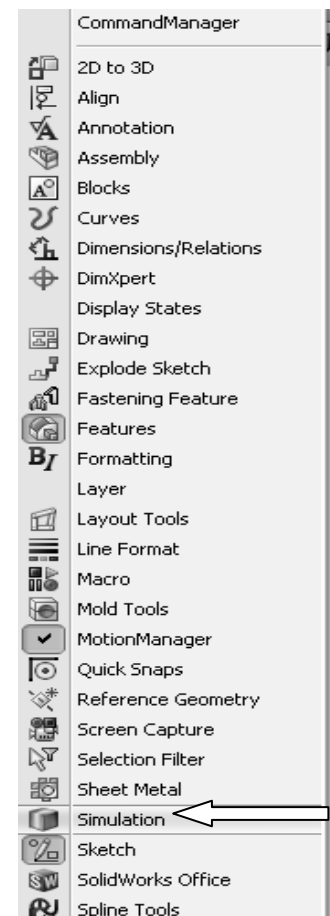


Figure 3 – **Command Manager** pop-up menu.

Display method #2

5. Right-click anywhere on the toolbar at top of the screen. The SolidWorks Simulation Command Manager menu again opens as shown in Fig. 3.

Analysis of Machine Elements using SolidWorks Simulation

6. At the top of the command manager, click to select **Command Manager**. This action adds a series of *tabs* beneath the main menu. A partial view of the toolbar appears in Fig. 5.



Figure 5 – Partial view of the **Simulation** toolbar, which is available by accessing the **Simulation** tab.

Initially the **Simulation** tab is selected and the same set of SolidWorks Simulation icons is displayed except that now they are accessible only after first selecting the **Simulation** tab.

7. Users familiar with the SolidWorks work environment are encouraged to select the **Features**, and **Sketch** tabs to view familiar icons accessible within these tabs. Return to the **Simulation** tab before continuing.

Display method #3

8. Right-click anywhere on the toolbar at top of the screen to again open the Command Manager.
9. Just below **Command Manager** at top of the pull-down menu, click to select **Use Large Buttons with Text**. This action adds a brief description beneath each icon as illustrated in a partial view of the toolbar displayed in Fig. 6.

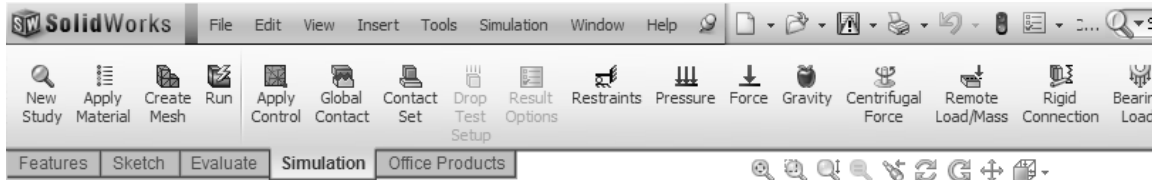



Figure 6 – Partial view of the **Simulation** toolbar with descriptive text applied beneath each icon.

Presuming the **Simulation** icons to be unfamiliar to new users, the display mode illustrated in Fig. 6 is used in throughout this chapter. Now that the work environment is established, a Study of stresses in the curved beam begins below.

10. Within the **Simulation** tab, click to select the **New Study** icon . The **Study** property manager opens (not shown).
11. In the **Name** dialogue box, replace **Study 1** by typing a descriptive name. For this example, type: **Curved Beam Analysis-YOUR NAME**. Including your name along with the Study name ensures that it is displayed on each plot. This helps to identify plots sent to public access printers.

12. In the **Type** dialogue box, verify that **Static** is selected as the analysis type.
13. Click **[OK]** ✓ (green check mark) to close the **Study** property manager. Notice that many of the icons on the **Simulation** tab are displayed in color to indicate they are now active.

The bottom half of the SolidWorks Simulation manager, seen in Fig. 7, is updated to show an outline for the current Study.

As in the previous example, the sequence of steps outlined in Fig. 7 is followed from top to bottom as the current finite element analysis is developed.

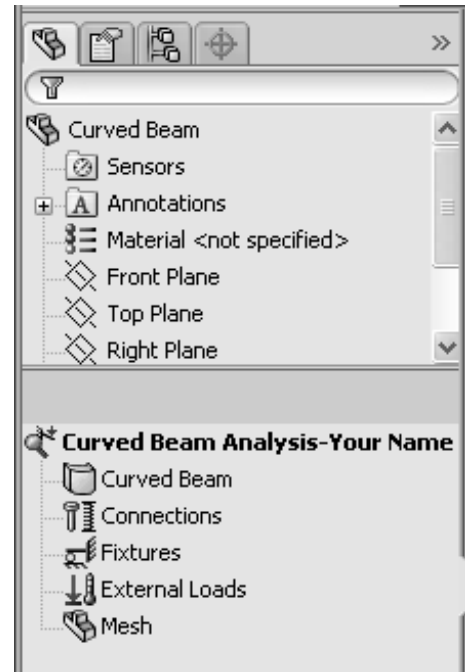



Figure 7 – Basic steps of a Study.

Assign Material Properties to the Model

Part *material* is defined as outlined below.

1. On the **Simulation** tab, click to select the **Apply Material** icon . The **Material** window opens. Alternatively, in the Simulation manager tree, right-click the **Curved Beam** folder and from the pull-down menu select **Apply/Edit Material...** The **Material** window is shown in Fig.8.

- a. Under **Select material source**, click to select **From library files**: Then, from the adjacent pull-down menu select **solidworks materials**.

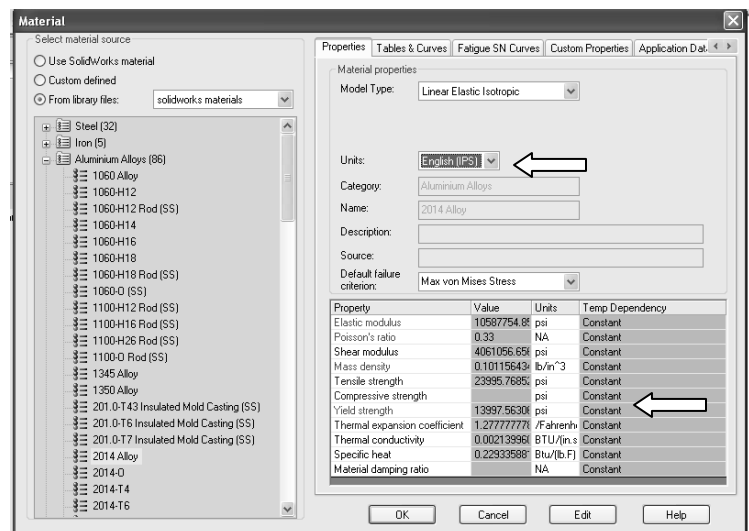


Figure 8 – Material properties are selected and/or defined in the **Material** window.

- b. Click the plus sign next to the **Aluminum Alloys (86)** and scroll down to select **2014 Alloy**. The properties of 2014 Aluminum alloy are displayed in the right half of the window.
- c. In the right-half of this window, select the **Properties** tab (if not already selected), and change **Units:** to **English (IPS)**.

In the table, note the material **Yield Strength** is a relatively low 13997.56306 psi (approximately 14000 psi). Material with a low yield strength is intentionally chosen to facilitate discussion of *Safety Factor* later in this example. Examine other values in the table to become familiar with data available in the material properties library.

Within the table, also notice that some material properties are indicated by red text and others by blue. Red text indicates a material property that must be specified based on the type of analysis being performed. Conversely, material properties highlighted in blue text are optional, and those in black text are not needed.

- d. Click **[OK]** to close the **Material** window. A check mark “✓” appears on the **Curved Beam** folder to indicate a material has been selected. The material type (**-2014 Alloy-**) is also listed adjacent to the model name.


Aside:

If at any point you wish to *change the material specification* of a part, such as during a redesign, right-click the name of the particular part whose material properties are to be changed and from the pull-down menu select **Apply/Edit Material**. The **Material** window is opened and an alternative material can be selected.

However, be aware that when a different material is specified *after* running a solution, it is necessary to run the solution again using the revised material properties.

Applying Fixtures

For a static analysis, adequate restraints must be applied to stabilize the model. In this example, the bottom surface of the model is considered “fixed.” However, because tetrahedral elements are used to mesh this model, “immovable” restraints are applied to the bottom surface.

1. In the **Simulation** toolbar click the **Restraints** icon . Alternatively, in the Simulation manager tree, right-click **Fixtures** and from the pull-down menu select **Fixed Geometry...** Either action opens the **Fixture** property manager shown in Fig. 9. *NOTE: The **Restraints** icon may be renamed **Fixtures** in newer software releases.*

2. Within the **Standard (Fixed Geometry)** dialogue box, select the **Fixed Geometry** icon (if not already selected).
3. The **Faces, Edges, Vertices for Fixture** field is highlighted (light blue) to indicate it is active and waiting for the user to select part of the model to be restrained. Rotate, and/or zoom to view the bottom of the model. Next, move the cursor over the model and when the bottom surface is indicated, click to select it. The surface is highlighted and restraint symbols appear as shown in Fig. 10. Also, **Face<1>** appears in the **Faces, Edges, Vertices for Fixture** field.

If an incorrect entity (such as a vertex, edge, or the wrong surface) is selected, right-click the incorrect item in the highlighted field and from the pop-up menu select **Delete**, then repeat step 3.

4. If restraint symbols do not appear, or if it is desired to alter their size or color, click the down arrow ▼ to open the **Symbol Settings** dialogue box, at bottom of the **Fixture** property manager and check the ☒ **Show preview** box.

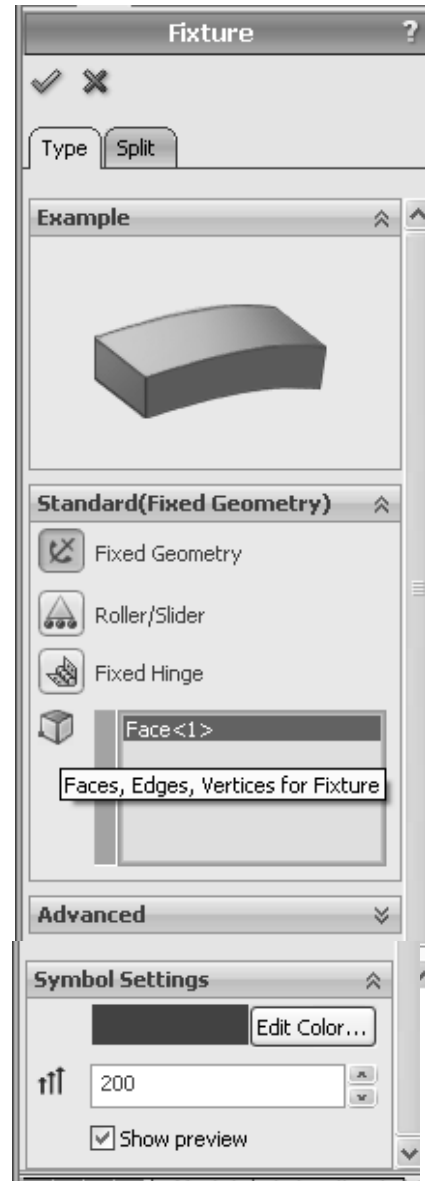


Figure 9 – **Fixture** property manager.

5. Both color and size of the restraint symbols (vectors in the X, Y, Z directions) can be changed by altering values in the **Symbol Settings** field of Fig. 9. Experiment by clicking the up ▲ or down ▼ arrows to change size of restraint symbols. A box of this type, where values can be changed either by typing a new value or by clicking the ▲ ▼ arrows, is called a “spin box.” Restraint symbols shown in Fig. 10 were arbitrarily increased to 200%. Experiment with this option.

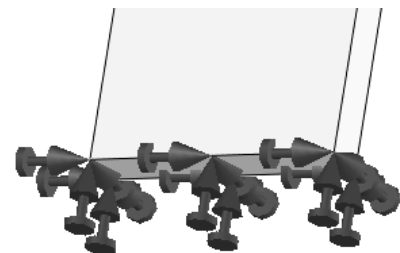


Figure 10 – Restraints applied to bottom of the curved beam model.

- Click **[OK]** ✓ (green check mark) at top of the **Fixture** property manager to accept this restraint. An icon named **Fixture-1** appears beneath the **Fixtures** folder in the Simulation manager tree.

Aside:

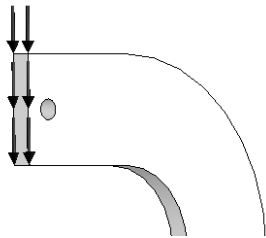
Restraint symbols shown in Fig. 10 appear as simple arrows with a small disk added to its tail. These symbols indicate **Fixed** restraints when applied to shell or beam elements. **Fixed** restraints set both *translational* and *rotational* degrees of freedom to zero (i.e., both X, Y, Z displacements and rotations (moments) about the X, Y, Z axes are zero). However, when applied to either solid models or truss elements, only *displacements* in the X, Y, and Z directions are restrained (i.e., prevented). This latter type of restraint is referred to as **Immovable**. Watch for this subtle, but important difference in future examples.

Applying External Load(s)

Next apply the downward force, $F_y = 3800$ lb, at the hole located near the top left-hand side of the model shown in Figs. 1 and 2. This force is assumed to be applied by a pin (not shown) that acts through the hole.

Analysis Insight

Because the goal of this analysis is to focus on curved beam stresses at Section A-A, and because Section A-A is well removed from the point of load application, modeling of the applied force can be handled in a number of different ways. For example, the downward force could be applied to the vertical *surface* located on the upper-left side of the model, Fig. (a). Alternatively, the force could be applied to the upper or lower *edge* of the model at the extreme left side, Fig. (b). These loading situations would require a slight reduction of the magnitude of force **F** to account for its additional distance from the left-side of the model to section A-A (i.e., the moment about section



A-A must remain the same).

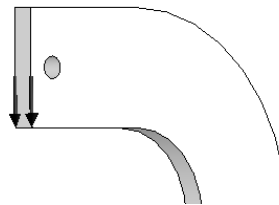


Figure (a) – Force applied to left surface.


Figure (b) – Force applied to lower edge.

The above loads are simple to apply. However, the assumption of pin loading allows us to investigate use of a *Split Line* to isolate a *portion* of the bottom surface of the hole

where contact with a pin is assumed to occur. This surface is where a pin force would be transferred to the curved beam model. The actual contact area depends on a number of factors, which include: (a) geometries of the contacting parts (i.e., relative diameters of the pin and hole), (b) material properties (i.e., hard versus soft contact surfaces of either the pin or the beam), and (c) magnitude of the force that presses the two surfaces together. This example arbitrarily assumes a reasonable contact area so that use of a *Split Line* can be demonstrated. If, on the other hand, contact stresses in the vicinity of the hole were of paramount importance, then determination of the true contact area requires use of *Contact/Gap* analysis. This form of analysis is investigated in Chapter #6.

Inserting Split Lines

The first task is to isolate a portion of the area at the bottom of the hole. This can be accomplished by using a *Split Line*. The method described below outlines the use of a reference plane to insert a *Split Line*.

1. In the drawing toolbar, reorient the model by clicking the **Trimetric**  or **Isometric** view icon.
2. From the main menu, select **Insert**. Then, from succeeding pull-down menus make the following selections: **Reference Geometry** ► followed by **Plane...** The **Plane** property manager opens *and* the SolidWorks “flyout” menu also appears at the upper-left corner of the graphics screen as shown in Fig. 11.

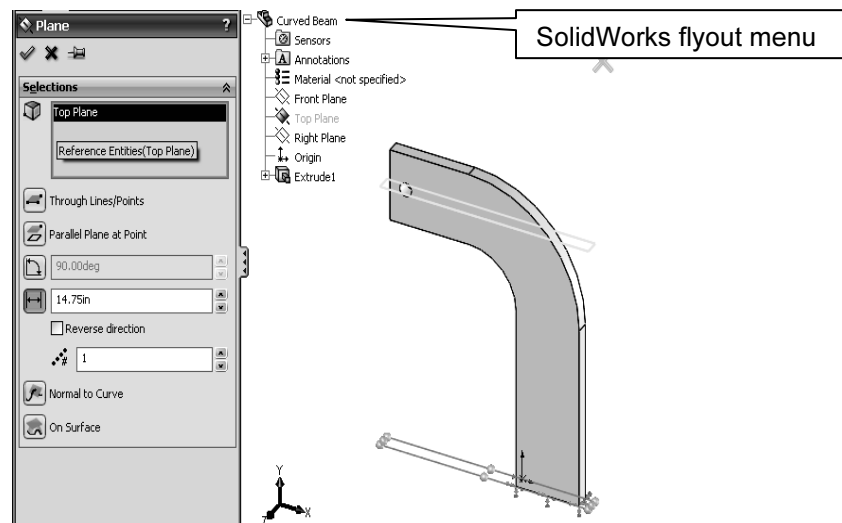


Figure 11 – The **Plane** property manager and menu selections made to create a reference plane that passes through the bottom of the hole.

3. To display the complete “flyout” menu shown in Fig. 11, it may be necessary to click the “+” sign adjacent to the SolidWorks **Curved Beam** icon. Continue to refer to Fig. 11 in steps 4 through 6.

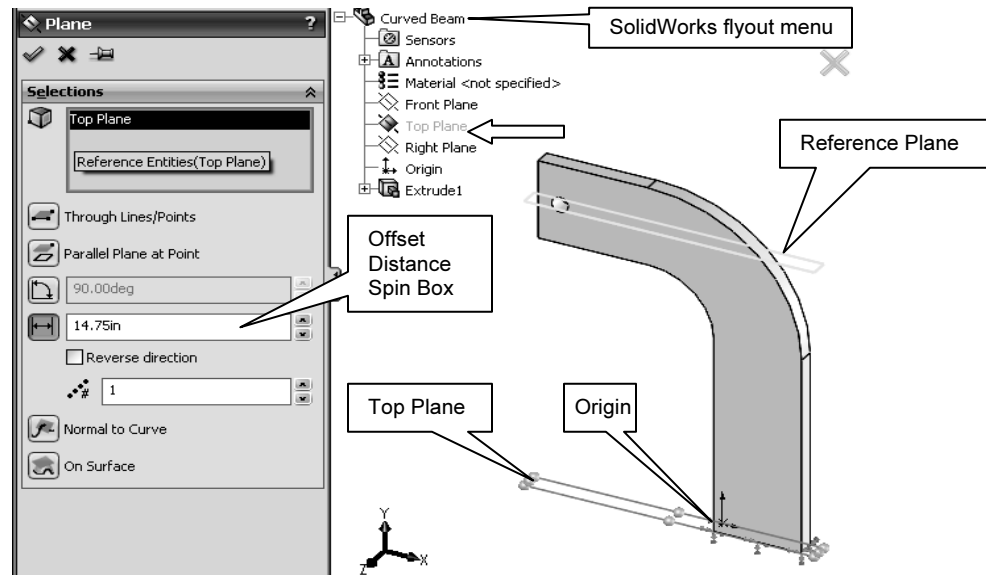


Figure 11(REPEATED) – The **Plane** property manager and various selections made to create a reference plane that passes through the bottom of the hole.

4. Within the **Plane** property manager, under **Selections**, the **Reference Entities** field is highlighted (light blue). Note text appearing in the *Status Line* at bottom left of the screen, which prompts the user to “**Select valid entities to define a plane (plane, face, edge, line or point).**” In the following steps, a horizontal reference plane that passes through the bottom portion of the hole is defined.
5. From the SolidWorks “flyout menu,” select **Top Plane**. For users who opened the **Curved Beam** part file available at the textbook web site, the top plane passes through the part origin, which is located on the bottom of the model¹ in Fig. 11.
6. Return to the **Selections** dialogue box and in the **Offset Distance** spin-box, type **14.75**. This is the distance *from* the **Top Plane** to a reference plane located so that it passes through the bottom portion of the hole.

Aside:

The 14.75 in. dimension is determined from the following calculation. Refer to Fig. 1 to determine the source of values used in the equation below.

10 in (height of straight vertical sides) + 3 in (radius of concave surface) + 1.75 in (distance from the horizontal edge beneath the hole and extending into the bottom portion of the hole) = 14.75in.

It is emphasized that the area intersected on the bottom of the hole is chosen *arbitrarily* in this example!

¹ Users who created a curved beam model from scratch can also follow these instructions. The *only* difference being specification of the proper *distance* from the **Top Plane** (used as a reference in *your* model) to the bottom of the hole.

7. Click **[OK]** ✓ to close the **Plane** property manager.

The reference plane created in the preceding steps appears highlighted in the current screen image. In the following steps this plane is used to create *Split Lines* near the bottom of the hole. These *Split Lines* enable us to define a small area on the bottom of the hole where the downward load will be applied.

8. From the main menu, select **Insert**. Then, from the pull-down menus choose: **Curve** ► followed by **Split Line...** The **Split Line** property manager opens as shown in Fig. 12
9. Beneath **Type of Split**, select ☒ **Intersection**. This choice designates the means by which *Split Lines* are defined for this example (i.e., they will be located where **Plane1** intersects the hole).
10. In the **Selections** dialogue box, **Plane1** may already appear in the **Splitting Bodies/Faces/Planes** field. If **Plane1** does not appear in this field, click to activate the field (light blue), then move the cursor onto the graphics screen and select the upper plane when it is highlighted. **Plane1** now appears in the top field.

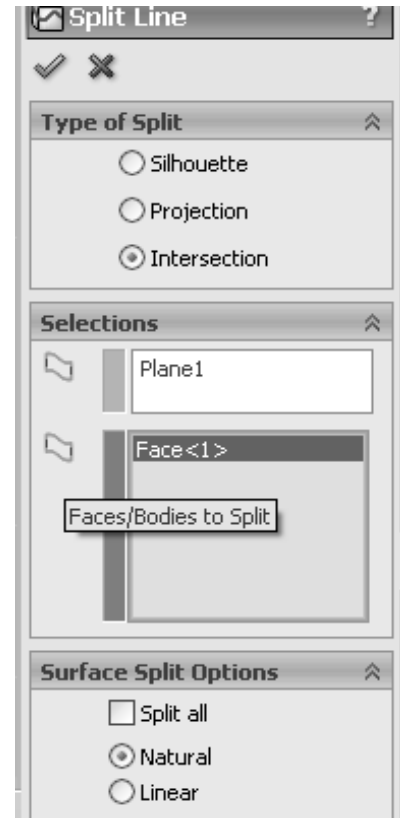


Figure 12 – **Split Line** property manager showing selections.

11. Next, click inside the **Faces/Bodies to Split** field. This field may already be active (light blue). Then move the cursor over the model and select anywhere on the *inside surface* of the hole. It may be necessary to zoom in on the model to select this surface. Once selected, **Face<1>** appears in the active field. Figure 13 shows a partial image of the model with *Split Lines* appearing where **Plane1** intersects the bottom of the hole.

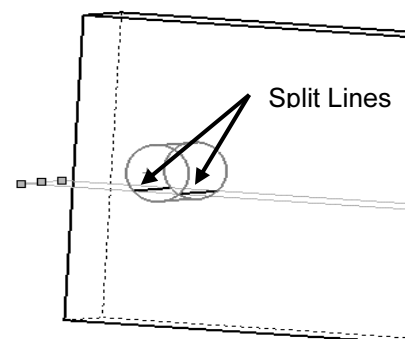




Figure 13 – Close-up view of hole showing *Split Lines* where **Plane1** intersects near the bottom of hole.

Remain zoomed-in on the model to facilitate applying a force to the inside of the hole.

12. In the **Surface Split Options** dialogue box, select ☒ **Natural**. A **Natural** split follows the contour of the selected surface.
13. Click **[OK]** ✓ to close the **Split Line** property manager.
14. If an information “flag” appears adjacent to the *Split Lines*, click ☒ to close it.

Applying Force to an Area Bounded by Split Lines

Now that a restricted area on the bottom of the hole has been identified, the next step is to apply a downward force, $F_y = 3800$ lb, on this area. Proceed as follows.

1. In the Simulation toolbar, select the **Force** icon . Alternatively, in the Simulation manager, right-click the **External Loads** icon and from the pop-up menu select **Force...** A partial view of the **Force/Torque** property manager appears in Fig. 14.
2. Within the **Force/Torque** dialogue box, click the **Force** icon  (if not already selected). Also, click ☒ **Selected Direction**. Again the **Faces, Edges, Vertices, Reference Points for Force** (light blue) field prompts the user to select an appropriate entity.
3. Move the cursor over the model and when the *bottom inside surface* of the hole is outlined, click to select it. **Face<1>** appears in the active field of the **Force/Torque** dialogue box.
4. Next, click to activate the second field from the top of the **Force/Torque** dialogue box. Passing the cursor over this field identifies it as the **Face, Edge, Plane, Axis for Direction** field. This field is used to specify the direction of the force applied to the bottom of the hole.

Because a downward, vertical force is to be applied, select a *vertical edge . . . any vertical edge . . .* on the model aligned with the Y-direction. After selecting a vertical edge, **Edge<1>** appears in the active field and force vectors appear on the model as seen in Fig. 15.

5. Set **Units** to **English (IPS)**, if not already selected.

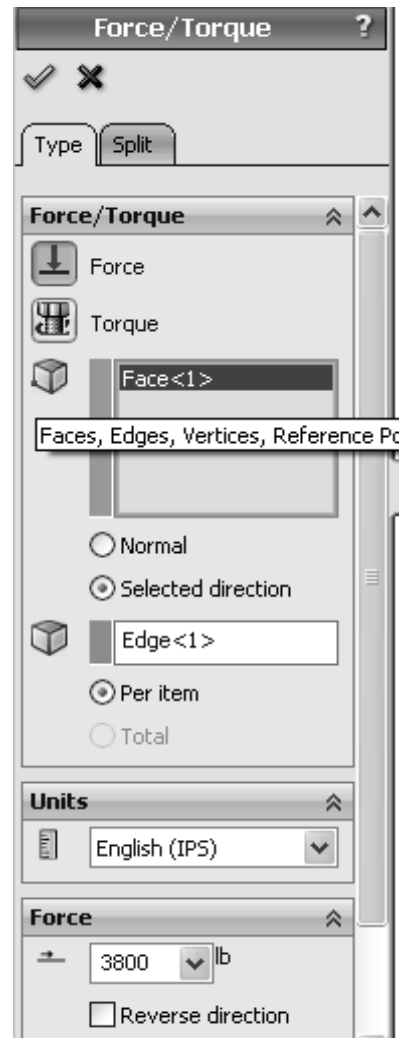


Figure 14 – Specifying a force and its direction on the hole bottom.

6. In the **Force** dialogue box, type **3800**. As noted in an earlier example, it may be necessary to check ☒ **Reverse Direction** if the force is not directed downward.
7. Click **[OK]** ✓ to accept this force definition and close the **Force/Torque** property manager. An icon named **Force/Torque-1** appears beneath the **External Loads** folder in the Simulation manager.

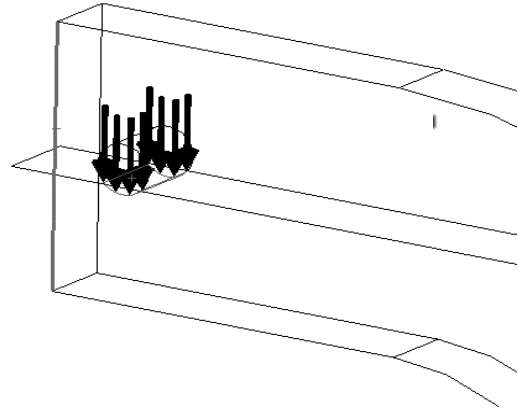





Figure 15 – Downward force applied between *Split Lines* on bottom of hole. A wireframe view of model is shown.

The model is now complete as far as material, restraints, and force definitions are concerned. The next step is to Mesh the model as described below.

Meshing the Model

1. In the Simulation toolbar, select the **Create Mesh** icon . Alternatively, in the Simulation manager tree, right-click **Mesh** and from the pull-down menu select **Create Mesh...** The **Mesh** property manager opens as shown in Figs 16 (a) and (b)

2. Click the down arrow  to open the **Mesh Parameters** dialogue box and verify that a ☒ **Standard mesh** is selected. Also set the **Unit** field to **in** (if not already selected). Accept the remaining default settings (i.e. mesh **Global Size** and **Tolerance**) at the bottom of this dialogue box.

3. Click the down arrow  to open the **Advanced** dialogue box, Fig. 16 (b). Verify that **Jacobian points** is set at **4 points**. These settings should produce a good quality mesh. However, verify that the settings are as listed and only change them if they differ.

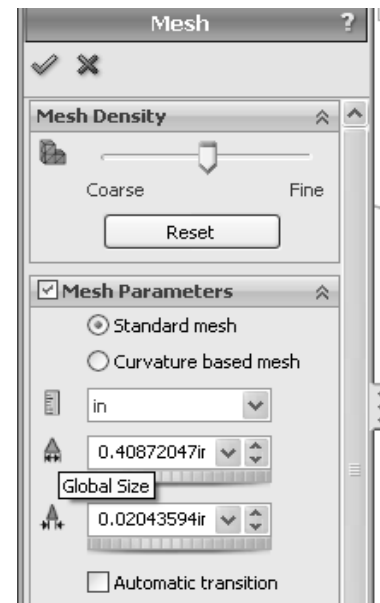


Figure 16 (a) – **Mesh** property manager showing system default **Mesh** settings applied to the current model.

4. Finally, click **[OK]** ✓ to accept these settings and close the **Mesh** property manager.

Meshing starts automatically and the **Mesh Progress** window appears briefly. After meshing is complete, SolidWorks Simulation displays the meshed model shown in Fig. 17. Also, a check mark “✓” appears on the **Mesh** icon to indicate meshing is complete.

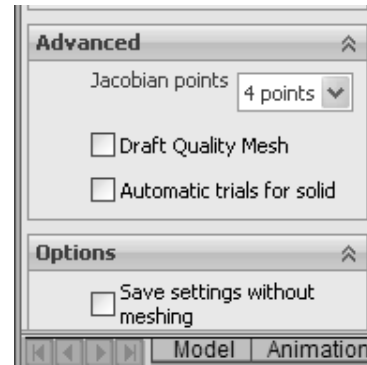


Figure 16 (b) – View of the **Advanced** portion of the **Mesh** property manager.

OPTIONAL:

5. Display mesh information by right-clicking the **Mesh** folder and select **Details...**

The **Mesh Details** window displays a variety of mesh information. Scroll down the list of information and note the number of nodes and elements for this model is 12536 nodes and 7197 elements (numbers may vary slightly due to the automated mesh generation procedure).

Rotate the model as illustrated in Fig. 17 and notice that the mesh is just two elements *thick*. Two elements across the model's thinnest dimension are considered the minimum number for which *Solid Elements* should be used. Thus, two elements are considered an unofficial dividing line between when *Shell* or *Solid Elements* should be used. Therefore, either element type could be used for this model. But, keep in mind that shell elements are typically reserved for thin parts.

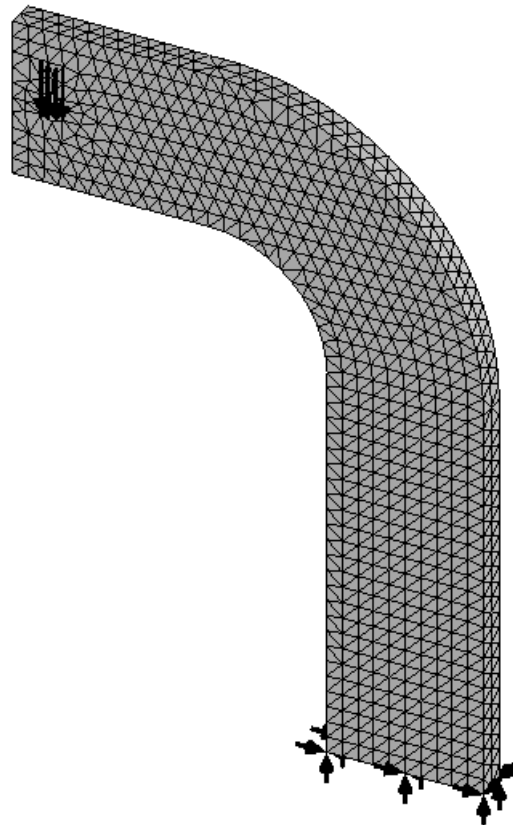



Figure 17 – Curved beam with mesh and boundary conditions illustrated.

Reminder

Recall that it is permissible to define material properties, fixtures, external loads, and create the mesh in any order. However, all these necessary steps must be completed before running a solution.

Solution

After the model has been completely defined, the solution process is initiated. During a solution the numerous equations defining a Study are solved and results of the analysis are saved for review.

1. On the Simulation tab, click the **Run** icon . Or, right-click the **Curved Beam Analysis - YOUR NAME (-Default-)** folder highlighted in Fig. 18. From the pull-down menu, select **Run**. Either action initiates the solution process.

After a successful solution, a **Results** folder appears at the bottom of the Simulation manager. This folder should include three sub-folders that contain default plots of results saved at the conclusion of each Study. These folders are named as illustrated in Fig. 18. If these folders do *not* appear, follow steps (a) through (f) outlined on page 1-14 of Chapter #1.

When the **Results** folders are displayed, alter **Units** for the **Stress** and **Displacement** plots, if necessary, as outlined below.

2. If the von Mises stress plot does not appear, then right-click **Stress1 (-von Mises-)** and from the pull-down menu, select **Show**; a stress plot appears.
3. Again, right-click **Stress1 (-vonMises-)** and from the pull-down menu select **Edit Definition...** In the **Display** dialogue box, verify **Units** are set to **psi**. If not, use the pull-down menu to change **Units** from **N/m^2** to **psi**.

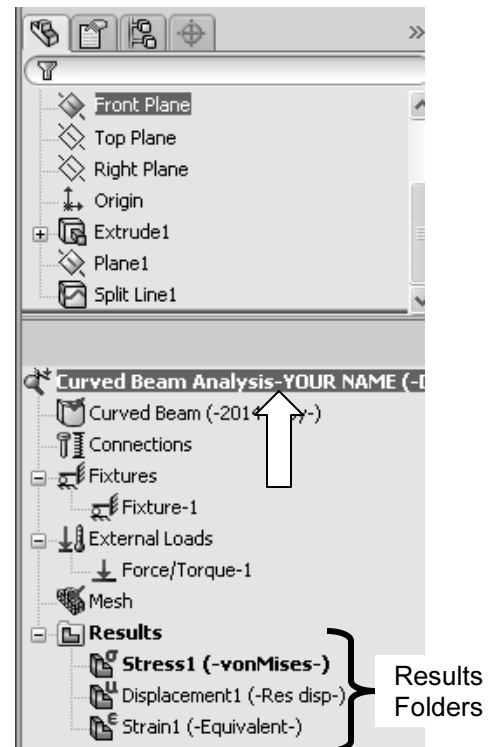


Figure 18 – **Results** folders created during the Solution process.

4. Click **[OK]** ✓ to close the **Stress Plot** property manager.
5. Repeat steps (2) through (4), however, in step (2), right-click on **Displacement1 (-Res disp-)** instead of **Stress1 (-vonMises-)** and in step (3) alter the **Units** field from **m** to **in**.

Examination of Results

Analysis of von Mises Stresses Within the Model

Outcomes of the current analysis, in the form of plots, can be viewed by accessing results stored in the various folders listed in the previous section. This is where validity of results is verified by cross-checking Finite Element Analysis (FEA) results against results obtained using classical stress equations. *Checking results is a necessary step in good engineering practice!*

1. In the Simulation manager tree, double-click the **Stress1 (-vonMises-)** folder (or right-click it and from the pull-down menu, select **Show**). A plot of the vonMises stress distribution throughout the curved beam model is displayed.

Figure 19 reveals an image *similar* to what currently appears on the screen. The following steps convert your current screen image to that shown in Fig. 19.

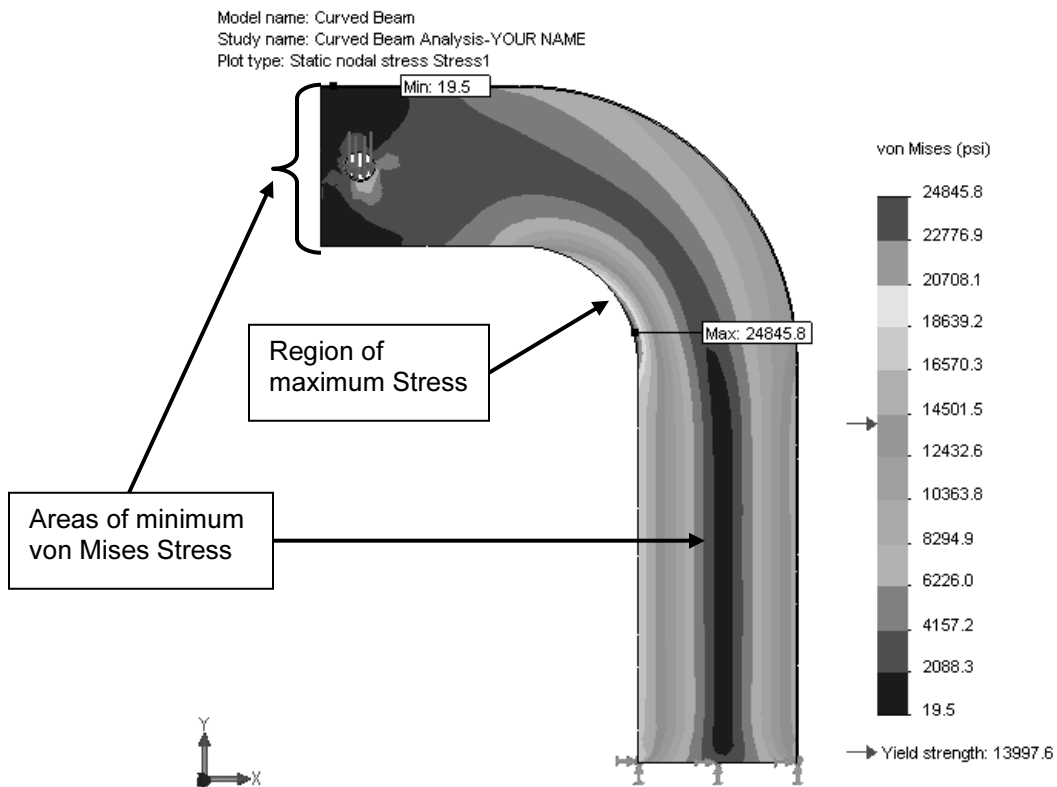


Figure 19 – Front view of the curved beam model showing von Mises stress *after* making changes outlined below. Note arrows indicating Yield Strength on the stress scale at right.

NOTE: Stress contour plots are printed in black, white, and grey tones. Therefore, light and dark color areas on your screen may differ from images shown throughout this text.

- Right-click **Stress1 (-vonMises-)** and from the pull-down menu select **Chart Options....** A portion of the **Chart Options** property manager is shown in Fig. 20.

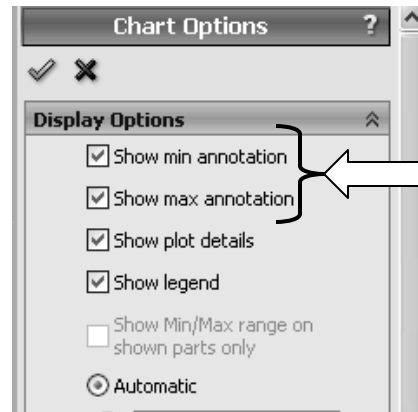


Figure 20 – Upper portion of **Chart Options** property manager showing current selections.

- Within the **Display Options** dialogue box, click to place check marks to activate ☒ **Show min annotation** and ☒ **Show max annotation**. This action causes the minimum and maximum von Mises stress locations to be labeled on the model.
- Click **[OK]** ✓ to close the **Chart Options** property manager.

- Right-click **Stress1 (-vonMises-)** and from the pull-down menu select **Settings....** The **Settings** property manager opens as illustrated in Fig. 21.

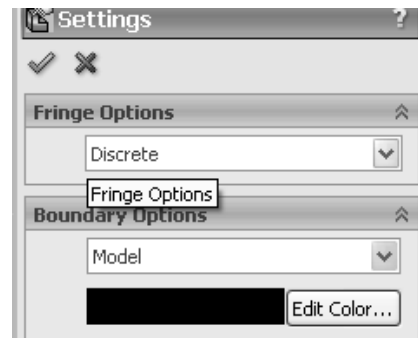



Figure 21 – Selections in the **Settings** property manager.

- From the **Fringe options** menu, select **Discrete** as the fringe type to be displayed.
- Next, in the **Boundary options** pull-down menu, select **Model** to superimpose an outline of the model on the image.
- Click **[OK]** ✓ to close the **Settings** property manager.

- In the Simulation tab, select the **Stress...**  icon. Alternatively, right-click **Stress1 (-vonMises-)** and from the pull-down menus select **Edit Definition...** Either action opens the **Stress Plot** property manager, shown in Fig. 22.

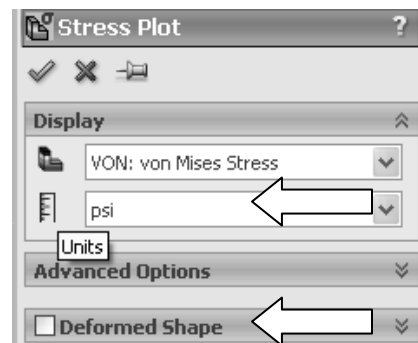


Figure 22 – **Stress Plot** property manager.

- In the **Deformed Shape** dialogue box, click to “clear” the check-box adjacent to ☐ **Deformed Shape**.
- Click **[OK]** ✓ to close the **Stress Plot** property manager.

The following observations can be made about Fig. 19.

OBSERVATIONS:

- Areas of low stress (dark blue) occur at the top-left side of the model. Also observe the dark blue line through the vertical center of the model. This corresponds to the neutral axis. The lowest von Mises stress is approximately 19.5 psi. Regions of high stress are indicated in red. The maximum stress indicated is 24845.8 psi, which occurs along the concave surface.
- Material *Yield Strength* = 13997.6 \approx 14000 psi is also listed beneath the color-coded **von Mises** stress color bar. An arrow adjacent to the color bar indicates where the Yield Strength lies relative to all stresses within the model. In this instance, it is clear that some stresses in the model *exceed* the material yield strength. *Yield Strength* and *Safety Factor* are investigated below.

Modern finite element software makes conducting a Finite Element Analysis (FEA) and obtaining results deceptively easy. As noted earlier, however, it is the validity of results and understanding how to interpret and evaluate them that is of primary importance. For these reasons, pause to consider two questions that should be intriguing or, perhaps, even bothering the reader.

First, why are all stress values positive in Fig. 19? (“+” values typically indicate tension) However, compressive stresses are known to exist along the concave surface for the given loading. Second, why does the solution show stresses that exceed the material yield strength when, clearly, stresses above the yield strength indicate yielding or failure? These, and many others, are the types of questions that should be raised continually by users of finite element software. Attempts to address these questions are included below.

To answer these questions, we briefly digress to investigate the definition of von Mises stress and a means to determine a *Safety Factor* predicted by the software.

Von Mises Stress -

The example of Chapter 1 skirted the issue about what the von Mises stress is or what it represents. That example further assumed that some readers might not be familiar with von Mises stress. For the sake of completeness and because von Mises stress typically is not introduced until later in a design of machine elements course, its basic definition is included below. Although this SolidWorks Simulation user guide is not intended to develop the complete theory related to von Mises stress, the usefulness of this stress might be summed up by the following statement-

The equation for von Mises stress “allows the most complicated stress situation to be represented by a single quantity.”² In other words, for the most complex state of stress that one can imagine (e.g., a three-dimensional stress element subject to a combination of

² Budynas, R.G., Nisbett, J. K., Shigley’s Mechanical Engineering Design, 8th Ed., McGraw-Hill, 2008, p.216.

shear and normal stresses acting on every face) these stresses can be reduced to a single number. That number is named the von Mises stress. The number represents a stress magnitude, “which can be compared against the yield strength of the material”³ to determine whether or not failure by yielding is predicted. As such, the von Mises stress is associated with one of the theories of failure for *ductile* materials; theories of failure are briefly discussed below. The von Mises stress is always a *positive, scalar* number.

The above statement answers the question about the positive nature of von Mises stress shown in Fig. 19. It also should provide some understanding about why the von Mises stress can be used to determine whether or not a part is likely to fail by yielding. *Safety Factor* and failure by yielding are explored further below.

Although the above definition indicates that von Mises stress is always a positive number, that superficial answer might continue to bother readers who intuitively recognize that compressive stresses result along the concave surface of the curved beam.

More fundamentally the issue in question gets to the heart of any analysis, whether it be a finite element analysis or stress calculations based on use of classical equations. The answer, of course, is that one must examine the *appropriate stresses* that correspond to the goals of an analysis. For example, in Chapter 1 it was decided that normal stress in the Y-direction (σ_y) was the only stress component that would yield favorable comparisons with stresses calculated using classic equations. The **Verification of Results** section below reveals the *appropriate stress* for the current example. Before continuing, answer the question, “What is the appropriate stress?” Check your answer below.

Verification of Results

In keeping with the philosophy that it is *always* necessary to verify the validity of Finite Element Analysis (FEA) results, a quick comparison of FEA results with those calculated using classical stress equations for a curved beam is included below.

Results Predicted by Classical Stress Equations

Although not all users may be familiar with the equations for stress in a curved beam, the analysis below should provide sufficient detail to enable reasonable understanding of this state of stress. The first observation is a somewhat unique characteristic of curved beams, namely, for symmetrical cross sections, its neutral axis lies closer to the center of curvature than does its centroidal axis. This can be observed in Fig. 23. By definition the centroidal axis, identified as r_c , is located half-way between the inside and outside radii of

³ Ibid

curvature. However, the neutral axis, identified by r_n , lies closer to the inside (concave) surface. Based upon the above observation, a free body diagram of the upper portion of the curved beam is shown in Fig. 23. Included on this figure are important curved beam dimensions used in the following calculations. Dimensions shown are defined below.

w = width of beam cross-section = 4.00 in (see Fig. 1)

d = depth of beam cross-section = 0.75 in (see Fig. 1)

A = cross-sectional area of beam = $w \cdot d = (0.75 \text{ in})(4.00 \text{ in}) = 3.00 \text{ in}^2$

r_i = radius to inside (concave surface) = 3.00 in

r_o = radius to outside (convex surface) = 7.00 in

r_c = radius to centroid of beam = $r_i + w/2 = 3.00 + 2.00 = 5.00 \text{ in}$

r_n = radius to the neutral axis = $w/\ln(r_o/r_i) = 4.00/\ln(7.00/3.00) = 4.72 \text{ in}$. [determined by equation for a curved beam having a rectangular cross-section]

c_i = distance from the neutral axis to the inside surface = $r_n - r_i = 4.72 - 3.00 = 1.72 \text{ in}$

c_o = distance from the neutral axis to the outside surface = $r_o - r_n = 7.00 - 4.72 = 2.28 \text{ in}$

e = distance between the centroidal axis and neutral axis = $r_c - r_n = 5.00 - 4.72 = 0.28 \text{ in}$

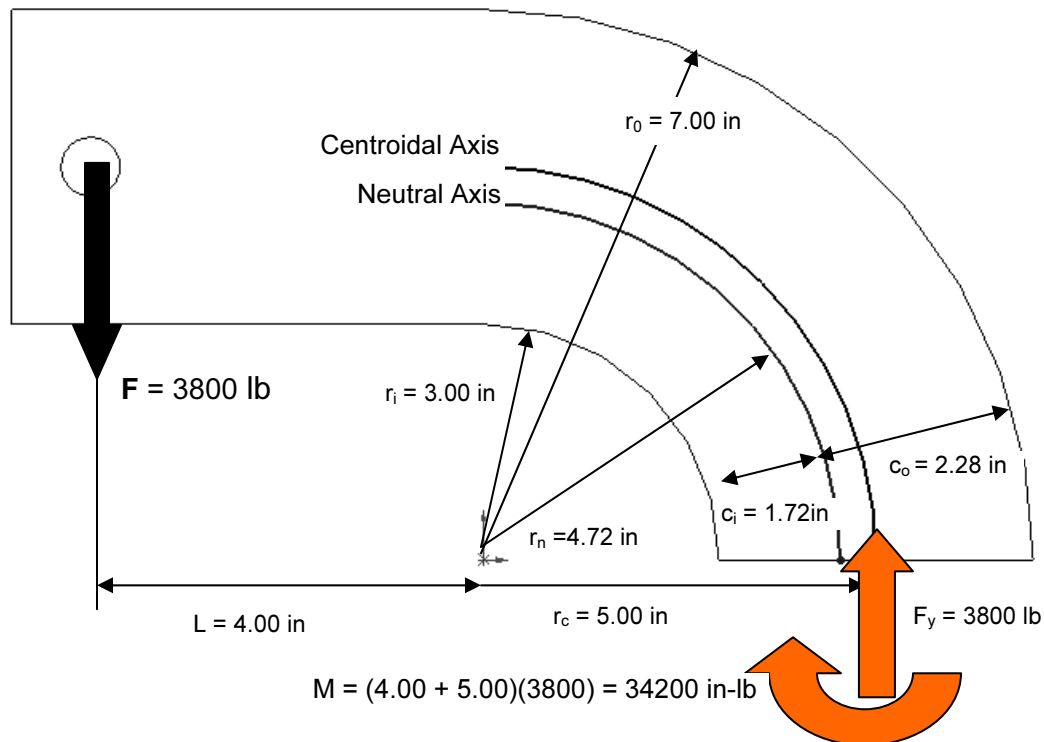


Figure 23 – Geometry associated with calculation of stress in a curved beam.

The reaction force F_y and moment M acting on the cut section are necessary to maintain equilibrium of the upper portion of the curved beam. Equations used to compute the combined bending and axial stresses that result from these reactions are included below. Each equation is of the general form,

$$\text{Curved beam stress} = \pm \text{bending stress} \pm \text{axial stress}$$

Where the “ \pm ” sign for bending stress depends on what side of the model is being investigated. Bending stress, caused by moment **M**, is compressive on the concave surface of the curved beam. Hence a minus “-” sign is assigned to the bending stress term in equation [1]. However, on the convex side of the beam bending stress causes tension on the beam surface thereby accounting for a “+” sign associated with the first term in equation [2]. Reaction force **F_y** acts to produce a compressive stress on the cut section. Therefore, a minus “-” sign is used with the axial stress component in both equations [1] and [2] below. In what direction do both of these stresses act?

Stress at the inside (concave) surface:



$$\sigma_i = \frac{Mc_i}{Aer_i} - \frac{F_y}{A} = \frac{-(34200 \text{ in-lb})(1.72 \text{ in})}{(3.00 \text{ in}^2)(0.28 \text{ in})(3.00 \text{ in})} - \frac{3800 \text{ lb}}{3.00 \text{ in}^2} = -24610 \text{ psi} \quad [1]$$

Stress at the outside (convex) surface:

$$\sigma_o = \frac{Mc_o}{Aer_o} - \frac{F_y}{A} = \frac{(34200 \text{ in-lb})(2.28 \text{ in})}{(3.00 \text{ in}^2)(0.28 \text{ in})(7.00 \text{ in})} - \frac{3800 \text{ lb}}{3.00 \text{ in}^2} = 11990 \text{ psi} \quad [2]$$

Comparison with Finite Element Results

In addition to serving as a quick check of results, this section reviews use of the **Probe** tool. Both the bending and axial stresses act normal to the cut surface in Fig. 23. Therefore, it is logical that the Finite Element Analysis stress in the Y-direction (σ_y) should be compared with values computed using equations [1] and [2] above. You are encouraged to produce a plot of stress σ_y on your own. However, abbreviated steps are outlined below if guidance is desired.

1. In the **Simulation** tab, click the **Stress...**  icon. Alternatively, right right-click the **Results** folder and from the pull-down menu select **Define Stress Plot...** Either of these actions opens the **Stress Plot** property manager.
2. In the **Display** dialogue box, select **SY: Y Normal Stress** from the pull-down menu. Also in this dialogue box, set the **Units** field to **psi**.
3. Click to un-check ☐ **Deformed Shape**.
4. Click **[OK]** ✓ to close the **Stress Plot** property manager. A new plot named **Stress2 (-Y normal-)** now appears beneath the **Results** folder and a plot of stress σ_y is displayed as shown in Fig. 24. If the plot does not appear, right-click **Stress2 (-Y normal-)** and select **Show** from the pull-down menu.
5. Click the **Plot Settings...** icon  to open the **Settings** property manager. Alternatively, right-click **Stress2 (-Y normal-)** and from the pull-down menu select **Settings...**

6. Within the **Settings** property manager, set the **Fringe Options** to **Discrete**.

A plot of normal stress σ_y in the Y-direction should now appear as shown in Fig. 24. The following observations can be made about Fig. 24.

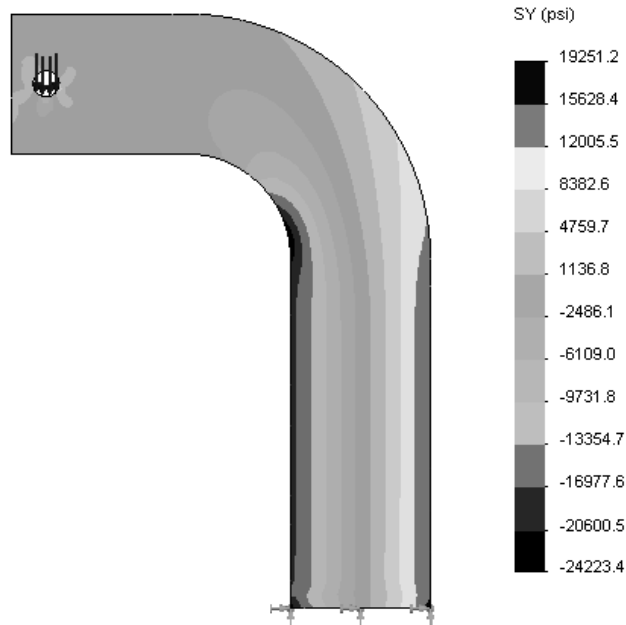


Figure 24 – Plot of **SY: Y Normal Stress** (σ_y) on the curved beam model

OBSERVATIONS:

- Areas of high stress, shown in orange, occur along the right vertical side of the model. Because this region is subject to tensile stress (i.e., positive “+” stress) positive stress magnitudes are expected.
- Compressive, (i.e., negative “-” stress) is shown by dark blue regions located along the left vertical and concave regions of the model. Once again compressive stress should correspond with the user’s intuitive sense of stress in that region.
- Low stress regions, corresponding to the neutral axis, or neutral plane, run through the vertical center of the model. Notice the sign change from “+” to “-” in the green color coded region of the stress chart adjacent to the model.
- To see the highest tensile stress (red), it is necessary to zoom in on the lower right-hand corner and right side of the model. Likewise to see the highest compressive stress (dark blue), zoom in on the concave arc of the model.

The model is next prepared to examine stresses at section A-A shown in Fig. 25.

7. Within the **Settings** property manager, set the **Boundary Options** to **Mesh**.

8. Click **[OK]** ✓ to close the **Settings** property manager.
9. Zoom in on the model to where the curved beam section is tangent to the straight, vertical section, shown as **A_i** and **A_o** in Fig. 25.

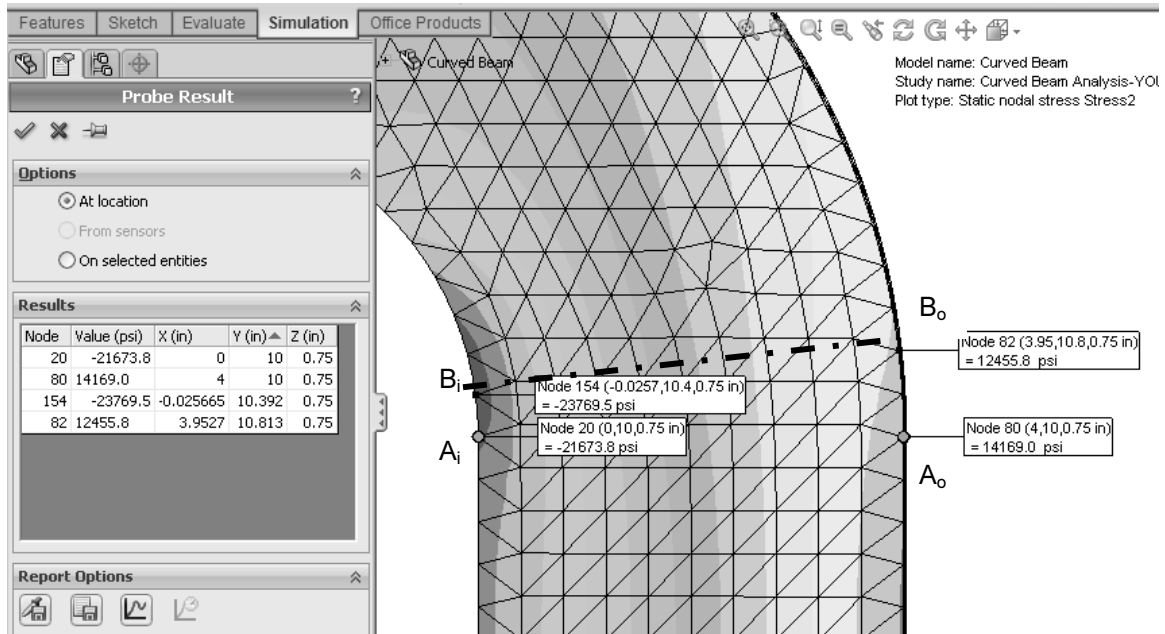



Figure 25 – Use the **Probe** tool to determine stress magnitudes at locations on the concave and convex sides of the curved beam model.

10. On the **Simulation** tab click the **Probe** tool  icon or right-click **Stress2 (-Y normal-)** and from the pull-down menu, select **Probe**. The **Probe Result** property manager opens as shown on the left side of Fig. 25.
11. In the **Options** dialogue box, select **At location** (if not already selected).
12. Move the cursor over the straight vertical edges on the left and right sides of the model. Each edge is highlighted as the cursor passes over it. Click to select two nodes (one on the left and one on the right) located at the *top* of each line. These nodes are located at the *intersection* between the straight vertical section and the beginning of the curved beam section. Selected nodes correspond to **A_i** and **A_o** in Fig. 25. If an incorrect node is selected, simply click **At location** in the **Options** dialogue box to clear the current selection and repeat the procedure. *Do not close the **Probe Result** property manager at this time.*

The above action records the following data in the **Results** dialogue box: **Node** number, **Value (psi)** of the plotted stress (σ_y), and the **X, Y, Z** coordinates of the selected node. Also, a small “flag” appears adjacent to each node on the model and repeats data listed in the **Results** table. *It may be necessary to click-and-drag column headings to view values in the **Results** table.*

Table I contains a comparison of results found by using classical equations and Finite Element Analysis at these locations.

Table I – Comparison of stress (σ_y) from classical and finite element methods at Section A-A.

| Location | Manual Calculation (psi) | Probe Tool Results (psi) | Percent Difference (%) |
|----------------------|--------------------------|--------------------------|------------------------|
| Point A _i | -24610 | -21673 | 13.6% |
| Point A _o | 11990 | 14169 | 15.4% |

As an engineer, you should be disappointed and, in fact, quite concerned at the significant difference between the results of these two methods. After all, how can one have confidence in a design when two seemingly rational approaches yield results that differ by 13.6% or more? When results differ by this order of magnitude it is appropriate to *investigate further* to determine the cause for the disparity and not simply “write off” the differences as due the fact that two alternative approaches are used. Can you provide valid reasons *why* such large differences exist?

Further consideration should reveal the fairly obvious conclusion that St. Venant’s principle is once again affecting results. In this instance, a traditional engineering approach would dictate using classical equations for a straight beam in the straight vertical segment of the model below Section A-A, see Fig. 1 (repeated), and a different set of equations in the curved beam portion of the model above Section A-A. However, common sense suggests that there is a *transition region* between these two segments where neither set of classical equations is entirely adequate. In fact, due to the finite size of elements in this region, it is logical to presume that a Finite Element Analysis provides a more accurate solution than do classical equations in this transition region.

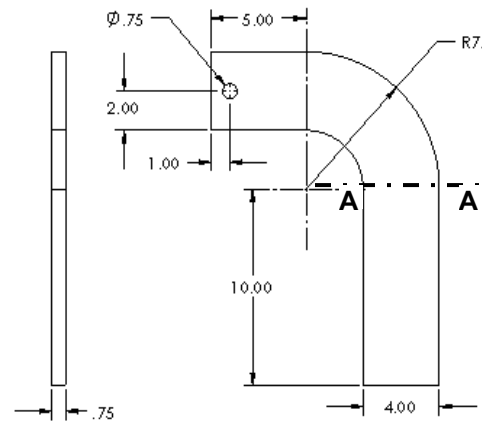


Figure 1 (Repeated) – Basic geometry of the curved beam model.

Given the above observations, we next proceed to sample stress magnitudes at Section B_i-B_o in Fig. 25, which is located slightly above the transition region. Proceed as follows.

13. Move the cursor over the curved edges of the model and on the concave side, click to select the *first* node *above* the previously selected node.
14. Next, on the convex side of the model, select the *second* node *above* the previously selected node.

Observe the two new stress magnitudes listed in the **Results** dialogue box and compare them to values listed in Table II below. Nodes **B_i** and **B_o** thus selected lie on a radial line that forms an approximate angle of 7.5° above the horizontal. Stress values calculated using the classical equations are modified to account for a slight shift of the centroidal axis due to beam curvature and for the change in angle of the axial force. Based on these values, a comparison of classical and FEA results in Table II reveals that values differ by at most 4.0%; a significant improvement over the initial calculations.

Table II – Comparison of stress (σ_y) for classical and finite element methods at Section B_i-B_o.


| Location | Manual Calculation (psi) | Probe Tool Results (psi) | Percent Difference (%) |
|----------------------------|--------------------------|--------------------------|------------------------|
| Point B_i | -24515 | -23769 | 3.1% |
| Point B_o | 11940 | 12455.8 | 4.1% |

15. Click **[OK]** ✓ to close the **Probe Result** property manager.

This concludes the verification of Finite Element results, but note that even better results would be expected at locations further from the transition region.

Assessing Safety Factor for the Curved Beam

SolidWorks Simulation provides a convenient means for the designer to determine and view a plot of Factor of Safety distribution throughout the curved beam model. To use this capability, proceed as follows.

1. In the **Simulation** tab, click the **Factor of Safety** icon . Alternatively, right-click the **Results** folder and from the pull-down menu, select **Define Factor Of Safety Plot...** The **Factor of Safety** property manager opens as shown in Fig. 26 and displays the first step of a three step procedure.
2. Read text in the **Message** dialogue box. This message indicates that the software automatically selects a failure criterion to determine the factor of safety.
3. In the upper pull-down menu of the **Step 1 of 3** dialogue box, select either **All** or **Curved Beam-Split Line1** as the model to be analyzed. Because there is only one part, the result is the same in this case.

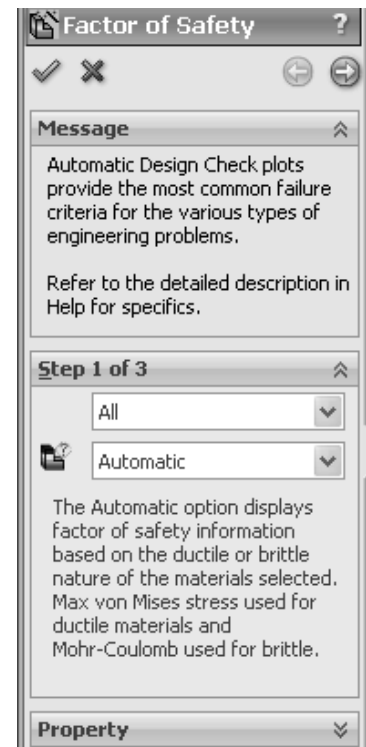


Figure 26 – **Factor of Safety**, Step #1 of safety factor check.

4. Next, in the **Criterion** field, second field from the top, click the pull-down menu to reveal the four failure criteria available for determination of the safety factor.

A brief overview of the four failure criteria is provided below.

- **Max vonMises Stress** – This failure criterion is used for ductile materials (aluminum, steel, brass, bronze, etc.). It is considered the best predictor of actual failure in ductile materials and, as such, provides a good indication of the true safety factor. This criterion is also referred to as the “Distortion Energy Theory” in many design of machine elements textbooks.
- **Max Shear Stress (Tresca)** – This criterion also applies to ductile materials. However, it is a more conservative theory thereby resulting in lower predicted safety factors. As a consequence of its conservative nature, parts designed using this criterion may be somewhat oversized.
- **Mohr-Coulomb Stress** – This failure criterion is applied to the design and analysis of parts made of brittle material (cast iron, concrete, etc.) where the ultimate compressive strength exceeds the ultimate tensile strength ($S_{uc} > S_{ut}$).
- **Max Normal Stress** – Also applicable for brittle materials, this failure criterion does not account for differences between tensile and compressive strengths within SolidWorks Simulation. This theory is also regarded as the least accurate of the methods available.
- Other failure criteria apply for *shell* elements made of composite materials. These criteria are not described here.

5. Because the curved beam is made of aluminum and because a good estimate of safety factor is desired, choose **Max von Mises Stress** from the pull-down menu.

Upon making the above selection, the **Factor of Safety** property manager changes to that illustrated in Fig. 27. Immediately below the **Criterion** field notice that the factor of safety check is currently defined as

$$\frac{\sigma_{vonMises}}{\sigma_{Limit}} < 1$$

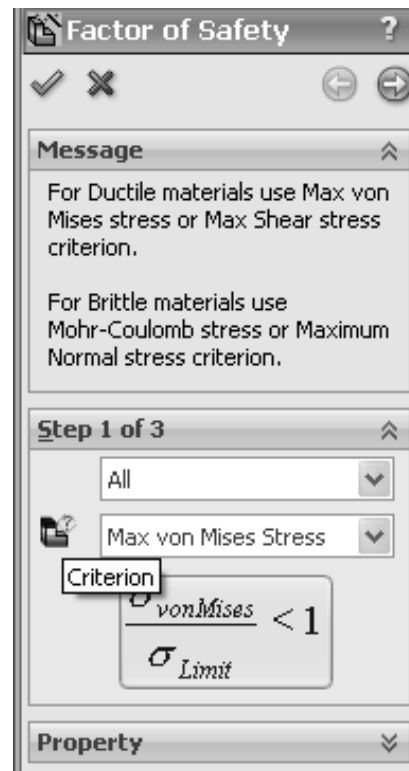



Figure 27 – First **Factor of Safety** property manager. The comparison stress (failure criteria) is selected here.

In other words, the previous equation is currently set to identify locations in the model where the ratio of von Mises stress to the “limiting” value of stress (i.e., the Yield Strength) is < 1 .

Thus, the above criterion identifies locations where yielding of the model is *not* predicted because model Yield Strength, the denominator, is greater than the von Mises stress, the numerator. As initially defined, the above ratio is the *inverse* of the traditional safety factor definition, where:

$$\text{Safety Factor} = n = \text{strength}/\text{stress}$$

To plot only critical regions of the part, i.e., regions where the Yield Strength is exceeded and the safety factor is < 1 , proceed as follows –

6. Advance to the second step by clicking the right facing arrow button  at the top of the **Factor of Safety** property manager. The **Step 2 of 3** dialogue box appears as shown in Fig. 28.
7. In the top pull-down menu, select **psi** as the set of **Units** to be used.
8. Under **Set stress limit to**, click to select ☒ **Yield strength** (if not already selected).
9. Do *not* change the **Multiplication factor**.

Notice that the Yield and Ultimate strengths for the model appear at the bottom of this dialogue box.

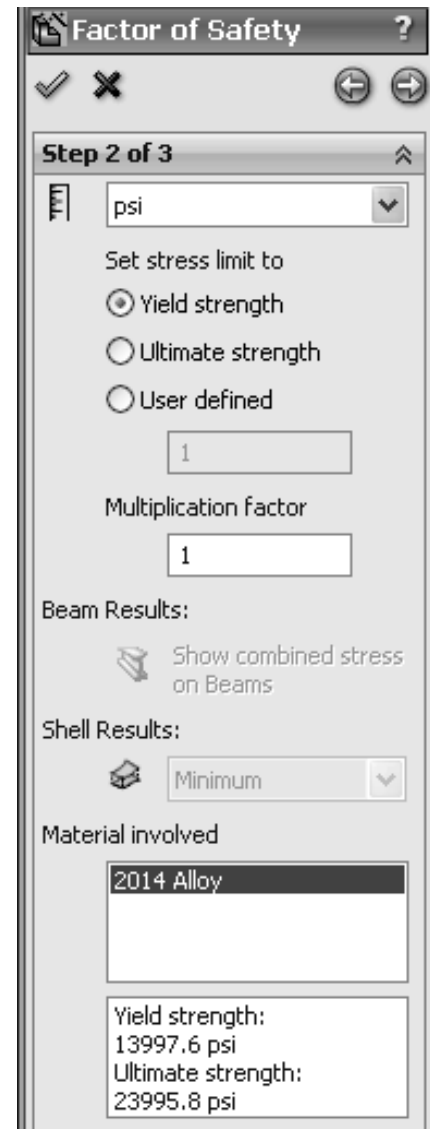



Figure 28 – **Step 2 of 3** in the **Factor of Safety** process.

Design Insight

In the event that a *brittle* material is being analyzed by the Mohr-Coulomb or the Max Normal Stress failure criterion, then it is appropriate to select the ☐ **Ultimate strength** as the failure criterion.

The **User defined** option is provided for cases where a user specified material is not found in the **Material Property** table or SolidWorks material library.

10. Click the right facing arrow button  at top of this property manager to proceed to **Step 3 of 3** in the **Factor of Safety** property manager shown in Fig. 29.

Two options are available for displaying the factor of safety. Brief descriptions of each are provided below.

- **Factor of safety distribution** – Produces a plot of safety factor variation throughout the entire part.
- **Areas below factor of safety** – A target value of safety factor is entered in the field beneath this option. The resulting display shows all areas below the specified safety factor in red and areas with a safety factor greater than specified in blue. This approach easily identifies areas that need to be improved during the design process.

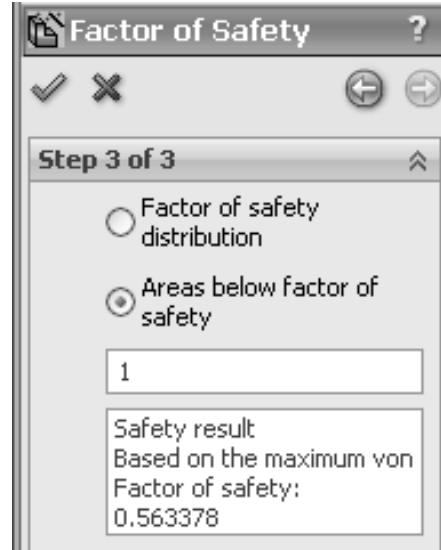



Figure 29 – Redefinition of *Safety Factor* and values to be displayed on the new plot.

11. Beneath **Step 3 of 3**, select ☒ **Areas below factor of safety** and type “1” in the **Factor of Safety:** field (if not already “1”).

At the bottom of this dialog box the **Safety result** field informs the user that the factor of safety is **0.563378** indicating that the design is *not* safe in some regions of the model. Recall that this value is based on a comparison between Yield Strength and the maximum von Mises stress. (*Values may vary slightly from those shown*).

Also note that this value of safety factor closely matches that computed by the reciprocal of the equation appearing in the first **Factor of Safety** window. That is:

$$\frac{\sigma_{Limit}}{\sigma_{vonMises}} = \frac{13997}{24846} = 0.56335$$

12. Click **[OK]**  to close the **Factor of Safety** property manager. A new plot folder, named, **Factor of Safety1 (- Max von Mises Stress-)**, is listed beneath the **Results** folder. Also, a plot showing regions of the model where the Safety Factor is < 1.0 (red) and where the Safety Factor is > 1.0 (blue) is displayed.
13. Right-click **Factor of Safety1 (-Max von Mises Stress-)**, and from the pull-down menu, select **Chart Options....** The **Chart Options** property manager opens.

14. In the **Display Options** dialogue box, check ☒ **Show min annotation** and click **[OK]** ✓ to close the **Chart Options** property manager.

The preceding step labels the location of minimum Safety Factor on the curved beam as shown in Fig. 30. As expected, this location corresponds to the location of highest stress previously illustrated in Fig. 19.

The figure now on the screen should correspond to Fig. 30. This figure shows regions where the factor of safety is less than 1 (unsafe regions) in red. Regions with a factor of safety greater than 1 (safe regions) are shown in blue. Localized regions, along the right and left vertical edges and extending into the concave region, have a safety factor less than one.

The line of text, circled near the top-left in Fig. 30, provides a “key” to interpret safe and unsafe regions on the model.

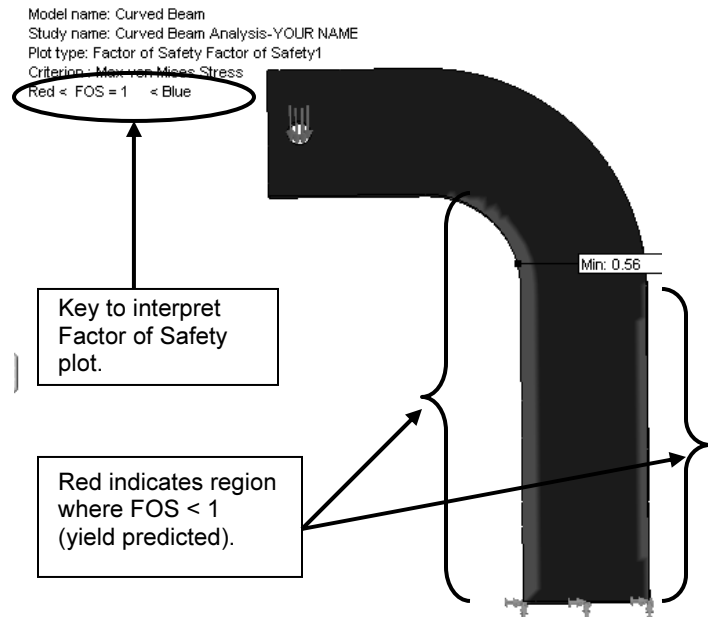


Figure 30 – Curved beam model showing areas where $FOS > 1$ (safety predicted) and where $FOS < 1$ (yield predicted).

15. Double-click **Factor of Safety (-Max von Mises Stress-)** and repeat steps 1 through 12 above, but this time set the **Areas below factor of safety to 2** instead of **1**, in step 11. How does the plot change?

A designer can repeat this procedure for any desired level of safety factor check.

In summary, an important aspect of the von Mises stress is that it can be used to predict whether or not a part might fail based on a comparison of its stress *value* to the magnitude of yield strength. This topic is aligned with the study of theories of failure found in most mechanics of materials and design of machine elements texts.

Analysis Insight

Faced with the above information, a designer would be challenged to redesign the part in any of several ways, depending upon design constraints. For example, it might be possible to change part dimensions to reduce stress magnitudes in the part. Alternatively, if part geometry cannot be changed, a stronger material might be selected or some combination of these or other possible remedies might be applied. Because this aspect of the current example is open-ended, it is not pursued here.

It is always good engineering practice to verify that results obtained correlate well with the given information. One simple way to confirm that results “make sense” is to check whether or not reaction forces are consistent with external loads applied to the finite element model. This section examines how to determine reaction forces at the base of the curved beam model. To accomplish this, proceed as follows.

1. In the SolidWorks Simulation manager, right-click the **Results** folder and from the pull-down menu, select **List Result Force...** The **Result Force** property manager opens as shown in Fig. 31.
2. In the **Options** dialogue box, verify that ☒ **Reaction Force** is selected.
3. In the **Selection** dialogue box, set **Units** to **English (IPS)**, if not already selected.
4. The **Select Faces, Edges, or Vertices** field is active (highlighted) and awaiting selection of the entity on which reaction forces are to be determined. Rotate the model so that its bottom (restrained) *surface* is visible and click to select it. **Face<1>** appears in the active field. This is the only face where reactions occur.
5. Click the **[Update]** button and the **Reaction Force (lb)** table at the bottom of the property manager is populated with data. Also, X, Y, and Z reaction force components appear at the base of the model.



Figure 31 – Data appearing in the **Result Force** property manager.

The **Component** column of this table lists names for the sum of reaction forces in the X, Y, and Z directions and the Resultant reaction. Magnitudes of the X, Y, Z reaction forces listed in the **Selection** column are identical to those in the **Entire Model** column. This result is expected since the entire model is restrained at only this one location.

Results interpretation is as follows:

SumX: 0.36773 (essentially zero) No force applied to model in X-direction.
SumY: 3799.3 (essentially 3800 lb = the applied force)
SumZ: 0.13439 (essentially zero) No force applied to model in Z-direction.
Resultant: 3799.3 (essentially 3800 lb = the applied force)

It should be noted that a *moment* reaction at the base of the curved beam is missing from the **Reaction Force** table. Also, checking the **Reaction Moment (lb-in)** dialogue box, at the bottom of the property manager, reveals no data entries. This outcome does not agree with the usual conventions for reactions associated with a free-body diagram, but it is consistent with our understanding of **Immovable** restraints applied to three-dimensional, solid, tetrahedral elements. The **Immovable** restraint only restricts translations in the X, Y, Z directions at each restrained node. This observation accounts for the fact that there are only three force reactions and no moments in the **Reaction Force** table of Fig. 31. Note, however, that calculated reaction forces do not exactly match the applied forces; slight mathematical errors exist.

6. Click **[OK]** ✓ to close the **Result Force** property manager.

The results above are valid for the **Entire Model**. However, in many instances a model is supported (i.e., restrained) at more than one location. In those instances it is necessary to determine reaction forces at other locations on a model. Performing a reaction check is quite simple and can be viewed as an additional means for users to verify the validity of boundary conditions applied to a model.

Although a surface was selected to examine reaction forces in the above example, it should be evident that other geometric features, such as edges or vertices can also be selected at other restrained locations on a particular model.

Logging Out of the Current Analysis

This concludes an introduction to analysis of the curved beam model. It is suggested that this file not be saved. Proceed as follows.

1. On the Main menu, click **File** followed by choosing **Close** from the pull-down menu.
2. The **SolidWorks** window, Fig. 32, opens and asks, “**Save changes to Curved Beam?**” Select the **[No]** button. This closes the file without saving any changes or results.

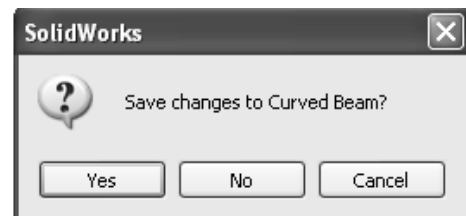


Figure 32 – **SolidWorks** window prompts users to either save changes or not.

EXERCISES

End of chapter exercises are intended to provide additional practice using principles introduced in the current chapter plus capabilities mastered in preceding chapters. Most exercises include multiple parts. In an academic setting, it is likely that parts of problems may be assigned or modified to suit specific course goals.

⚡ *Designates problems that introduce new concepts. Solution guidance is provided for these problems.*

EXERCISE 1 – Curved Beam Stresses in a “C”- Clamp

C-clamps, like that illustrated below, must pass minimum strength requirements before they can be qualified for general purpose use. Clamps are tested by applying equal and opposite loads acting on the gripping faces. Part of the federal test criteria requires that the movable (lower) jaw be extended a certain percentage of the distance of the fully-open state to ensure that column failure of the screw is an integral part of the test. Presuming that the movable jaw of the clamp satisfies the prescribed test criteria, perform a finite element analysis of the C-clamp subject to the following guidelines.

Open file: **C-Clamp 2-1**

- Material: **Cast Carbon Steel** (use S.I. units)
- Mesh: **High Quality** tetrahedral elements
- Fixture: **Fixed** applied to the upper gripping surface.
- External Load: **950 N** applied normal to lower gripping surface.

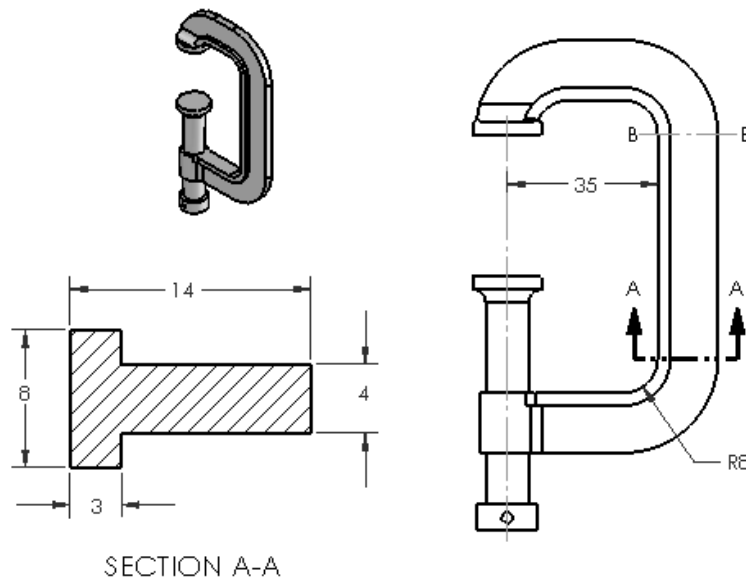


Figure E2-1 – “C”-clamp frame and cross-section dimensions. Stress to be determined at Section A-A.

Determine the following:

- a. Use classical equations to compute stress at the inside and outside surfaces of the C-clamp frame at section A-A. Section A-A is located where the straight and curved sections are tangent.
- b. Create a stress contour plot of von Mises stress in the frame of the C-Clamp. Include automatic labeling of maximum von Mises stress on this plot.
- c. Use the **Probe** feature to produce a graph of the *most appropriate stress* across section A-A. In other words, because values from this plot are to be compared with manual calculations of part (a), it is necessary to choose the corresponding stress from those available within the finite element software. Include a descriptive title and axis labels on this graph. When using the **Probe** feature, begin at the concave (left inside) surface and select nodes across the model continuing to the outside of the “T” cross-section. Use equation [1] to compare percent differences between classical and FEA determination of stresses at the inside and outside surfaces.

$$\% \text{ difference} = \frac{(\text{FEA result} - \text{classical result})}{\text{FEA result}} * 100 = \quad [1]$$

- d. Assuming the C-clamp is made of a ductile material, produce a plot showing regions where safety factor < 1.5.
- e. Question: Is there justification for using **High** quality elements for analysis of the C-clamp? Justify your answer by providing reasons either “for” or “against” use of **High** quality elements for this model.

EXERCISE 2 – Curved Beam Stresses in Hacksaw Frame

A common, metal-cutting “hacksaw” is shown in Fig. E 2-2. A solid model of the hacksaw is available as file: **Hacksaw 2-2**. The model is simplified to include two 0.125 inch diameter holes that pass through the lower left and lower right ends of the hacksaw “backbone” labeled in Fig. E 2-2. For analysis purposes, the inside surface of the left-hand hole is to be considered **Fixed** (i.e. immovable). Use split lines to create a small “patch” of area on the inside surface of the hole located at the right end of the backbone. On this surface apply a 50 lb force induced by a tensile load in the saw blade that is ordinarily held in place between these two holes. Assume the following.

- Material: **AISI 1020 Steel** (use English units)
- Mesh: **High quality** tetrahedral elements; use default mesh size.
- Units: **English (IPS)**

Analysis of Machine Elements using SolidWorks Simulation

- Fixture: **Fixed** applied to inside of left hole.
- External Load: **50 lb** applied parallel to the X-direction on the inner surface of right-hand hole (split lines needed; placement of these lines is user defined).

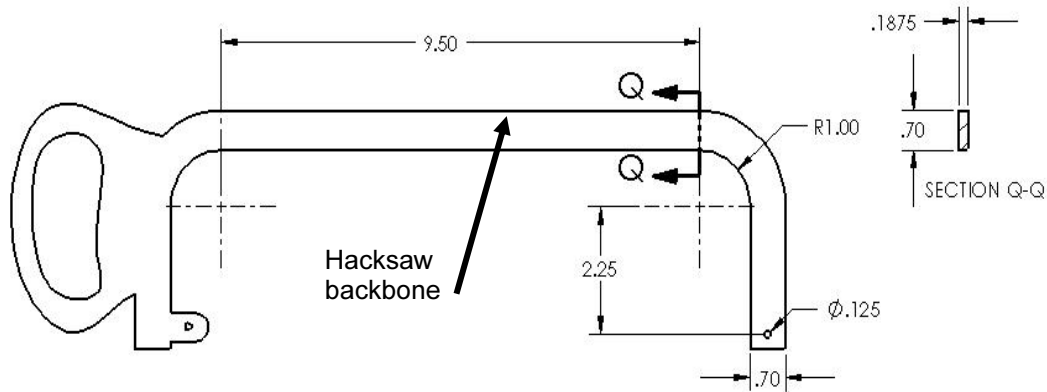


Figure E2-2 – Basic geometry of a hacksaw frame. Stress is to be determined at Section Q-Q.

Determine the following:

- Use classical equations to compute stress at the inside and outside surfaces of the hacksaw frame at section Q-Q. Section Q-Q is located where the straight and curved sections are tangent.
- Include a zoomed-in image of the right-hand hole so that the 50 lb applied load can clearly be seen to act between user specified *Split Lines*.
- Create a stress contour plot of von Mises stress in the saw backbone. Include automatic labeling of maximum von Mises stress on the plot.
- Use the **Probe** feature to produce a graph of the *most appropriate stress* across section Q-Q, beginning at the inside (concave) surface and continuing to the outside (convex) surface of the backbone cross-section. Use the **Stress Plot** property manager to select the *appropriate stress* for this plot to enable comparison with manual calculations of part (a) above. Include a descriptive title and axis labels on this graph. Also, below the graph, cut-and-paste a copy of the **Probe Results** table showing values used in this comparison. Then use equation [1], repeated below, to compute the percent difference between classical and finite element solutions at the inside and outside surfaces of the saw backbone.

$$\% \text{ difference} = \frac{(\text{FEA result} - \text{classical result})}{\text{FEA result}} * 100 = \quad [1]$$

- Based on von Mises stress, create a plot showing all regions of the model where Safety Factor < 2.2 and circle these regions on the plot. Include a software applied label indicating the maximum and minimum values of Safety Factor.

- f. Question: Is there justification for using **High** quality elements for analysis of the hacksaw backbone? Justify your answer by providing reasons either “for” or “against” using **High** quality elements for this model.

EXERCISE 3 – Stresses in a Curved Anchor Bracket

The curved beam shown in Fig. E 2-3 is subject to a horizontal load applied by means of a pin (not shown) that passes through a hole in its upper end. A solid model of this part is available as file: **Anchor Bracket 2-3**. The lower-left end of the part is attached to a rigid portion of a machine frame (not shown). Thus, because three-dimensional tetrahedral elements are to be used to model this part, the restraint at this location should be considered **Immovable**. Use split lines to create a small “patch” of area on the inside surface of the 16 mm diameter hole. Locate these split lines 24 mm from the right edge of the model. On this surface apply a horizontal force of 8600 N acting in the positive X-direction (to the right). Assume the following.

- Material: **AISI 1010 Steel, hot rolled bar** (use SI units)
- Mesh: **High Quality** tetrahedral elements; use default mesh size.
- Fixture: **Fixed** applied on the inclined surface.
- External Load: **8600 N** in the X-direction applied on the right, inside surface of the 16 mm diameter hole.

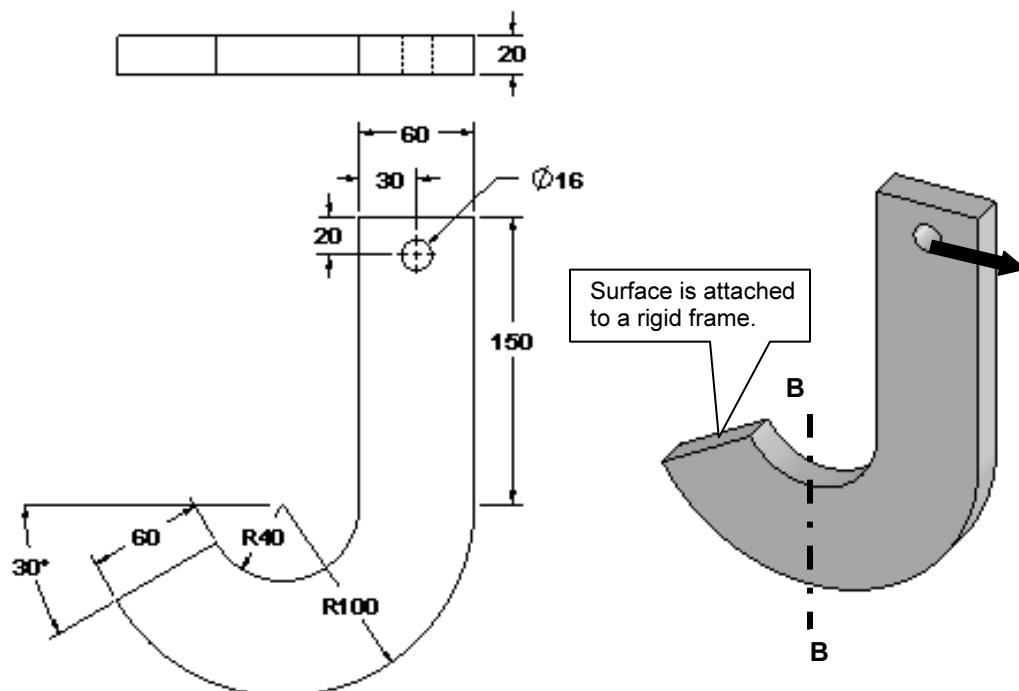


Figure E2-3 Dimensioned view of the Anchor Bracket. Stress is to be determined at Section B-B.

Determine the following:

- Use classical equations to compute stress at the inside (concave) surface and the outside (convex) surface of the anchor bracket at section B-B. Section B-B passes through the center of curvature of the curved beam and is considered to be a vertical line. Include a labeled free body diagram of the portion of the anchor bracket to the right of section B-B.
- Include a zoomed-in image of the hole so that the force $F_x = 8600 \text{ N}$ can clearly be seen to act between user specified *Split Lines*.
- Create a stress contour plot of von Mises stress in the anchor bracket. Include automatic labeling of maximum von Mises stress on the plot.
- Using von Mises stress, create a plot showing all regions of the model where Safety Factor < 1.0 . Indicate regions (if any) where the Safety Factor < 1 occurs by circling their location(s) on the figure and labeling them as FOS < 1 . Include a software applied label indicating locations of maximum and minimum values of Safety Factor.
- Use the **Probe** feature to produce a graph of the *most appropriate stress* across the bracket at section B-B. Begin at the inside (concave) surface and continue to the outside (convex) surface. (See the “*HINT*” at bottom of this page for guidance when making this graph). Use the **Stress Plot** property manager to select the *appropriate stress* for this plot to enable comparison with manual calculations of part (a) above. Add a descriptive title and axis labels to this graph.


Below the graph or on a separate page either: (a) cut-and-paste a copy of the **Probe Results** table that includes values used for this comparison, or (b) click the

Save icon , located in the **Report Options** dialogue box, to create an Excel spreadsheet containing all values in the **Probe Results** table.

After determining both classical and FEA results at section B-B, use equation [1] to compute the percent difference between classical and finite element solutions at the inside and outside surfaces of the bracket.

$$\% \text{ difference} = \frac{(\text{FEA result} - \text{classical result})}{\text{FEA result}} * 100 = \quad [1]$$

HINT: Because the mesh generation scheme within SolidWorks Simulation creates an optimized mesh, it is probable that (a) a straight line of nodes does *not* exist across the model at section B-B (thus, choose the best straight line), and (b) it is also *unlikely* that node points *occur exactly on a vertical centerline through the center of curvature*. For these reasons, and to obtain the best estimate of stress on a vertical line through the curved section, proceed as follows.

- Zoom in on a front view of the model at section B-B.
- On the **Simulation** tab, click the **Probe** icon . This action opens the **Probe Results** property manager.
- On the SolidWorks flyout menu, click the “+” sign to display the complete flyout menu.
- Move the cursor over **Right Plane** in the flyout menu. This action highlights an edge view of the **Right Plane**, which will assist in locating nodes closest to a vertical line at Section B-B. Unfortunately, the line disappears when the cursor is moved, but it is very useful none the less.

EXERCISE 4 – Stresses in a Curved Photoelastic Model

(Special Topics: Custom material defined, using a “Hinge” joint for Fixture)

Figure E 2-4 shows a model of curved beam made from a photoelastic material. The beam is subject to an axial load **F**. Beams such as this are commonly used in experimental stress analysis laboratories where photoelastic techniques are studied. Photoelastic material has a unique optical property known as birefringence. Thus, when a photoelastic model is subject to applied loads in a field of polarized light, the light passing through the model undergoes changes of wave length that produce visible “fringes” within the model as shown in Fig. E 2-5. These “fringes” are analogous to, but not equal to, stress contour plots produced when upon completion of a finite element analysis. In this exercise, stresses produced within the curved beam model are examined.

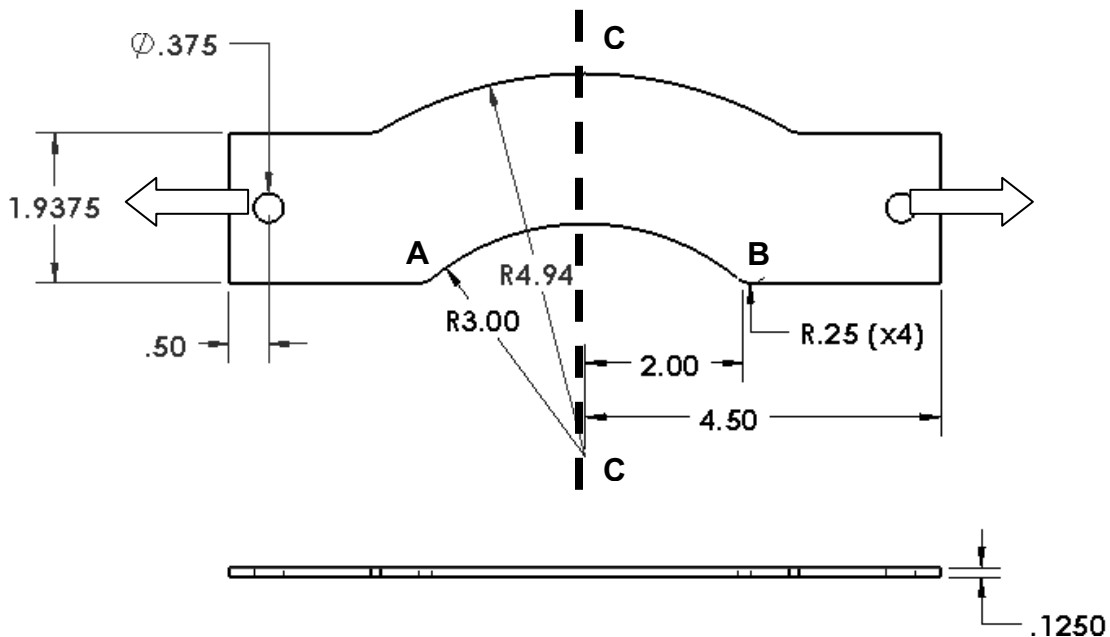


Figure E2-4 – Top view (above) and front (edge) view of a curved beam model.

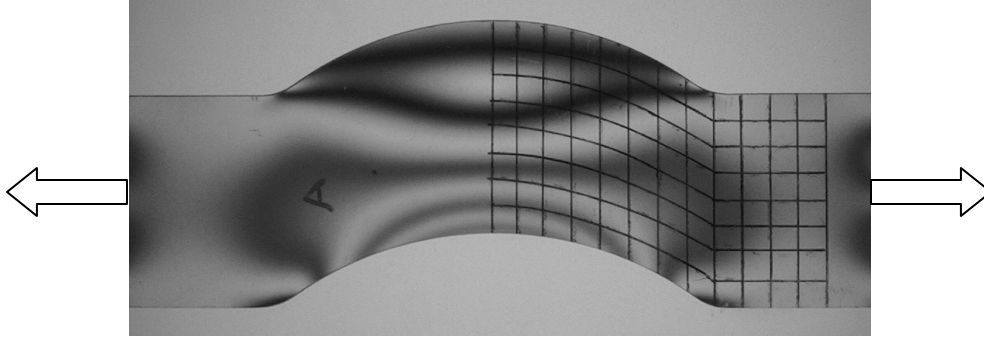


Figure E2-5 – “Fringes” appearing in a photoelastic model subject to a tensile load. A grid is superimposed on the model to facilitate locating specific stress magnitudes and directions.

Create a finite element model of this part that includes custom material specification, fixtures, external loads, a default high quality solid mesh, solution, and results analysis.

Open the file: **Curved Bracket**

- Material: Material properties not found in SolidWorks material library (use custom properties below)
 $E = 360e3$ psi Modulus of Elasticity (use English units)
 $\nu = 0.38$ Poisson’s ratio
 $S_y = 1200$ psi Yield Strength
- Mesh: **High Quality** tetrahedral elements; use default mesh size.
- Fixture: **Hinge Joint** applied at hole on left end of the model.
- External Load: **72 lb** in the X-direction applied between split lines on the inner surface of the hole located at the right end of the model..

Two aspects of this exercise are unique. First, properties of the photoelastic material are not available in the SolidWorks material library. And second, fixture at the left hole of the curved bracket is considered to be a **Fixed Hinge** joint. Guidance in the application of these two topics is provided below.

Solution Guidance

Custom Material Specification (It is assumed that the user has started a Study in SolidWorks Simulation.)

- Open the **Material** window by right-clicking the **Part** folder and selecting **Apply/Edit Material...**

Simulation Guidance (continued)

- Beneath **Select material source**, choose **Custom defined**.
- On the **Properties** tab, located in the right half of the window, select **Units:** as **English (IPS)**.
- Change **Category:** to **Plastic**; change the **Name:** to **Photoelastic material**. The **Description** and **Source:** fields can be left blank.
- Within the **Property** column of the lower table notice that red, blue, and black colors are used to indicate different **Property** names. Red lettering indicates information *required* for the solution. Blue lettering indicates *desirable, but unnecessary* information. And, property names appearing in black *are not required* for the current solution. Enter the values listed beneath “Material” in the problem statement. Ignore the **Mass density** field, blank fields, and values listed in black type.
- Click **[OK]** to close the **Material** window. A check “✓” appears on the **Curved Bracket** part folder.

Fixed Hinge Specification A **Fixed Hinge** joint acts like a door hinge. This joint type allows rotation about a fixed axis on the model, but prevents translation along that axis. For the **Curved Bracket** loaded in tension, using a **Fixed Hinge** at one end of the model prevents translations in the X, Y, Z directions, but allows the model to remain aligned with the external load as the part deforms. Proceed as follows.

- Right-click the **Fixtures** folder and from the pull-down menu select **Fixed Hinge...** The **Fixture** property manager opens as shown in Fig. E 2-6.
- In the **Standard (Fixed Hinge)** dialogue box, the **Cylindrical Faces for Fixture** field is highlighted (light blue).
- Zoom in on the left hole of the model and select its *inner surface*. **Face<1>** appears in the highlighted field, also shown in Fig. E 2-6. This cylindrical surface has an axis perpendicular to the model face.
- Click **[OK]** ✓ to close the **Fixture** property manager.

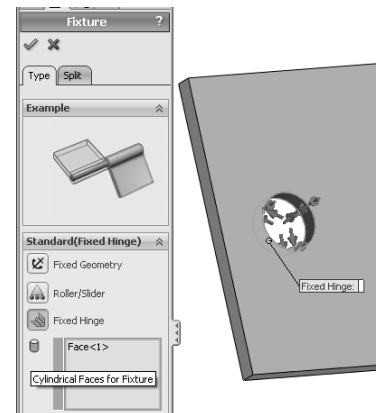


Figure E2-6 – Specifying a **Fixed Hinge** restraint.

A **Fixed Hinge** restraint allows the model to undergo rotations about the selected hole, but no translations.

Determine the following:

- Use classical equations to compute stress at the inside (concave) surface and the outside (convex) surface of the curved bracket model at section C-C. Section C-C passes through the center of curvature and is considered to be a horizontal line in the top view. Include a labeled free body diagram of the portion of the model to the right of section C-C.
- Include a zoomed-in image of the right hole so that the external load, $F_x = 72 \text{ lb}$, can clearly be seen to act between user specified *Split Lines* on the inner surface of the hole.
- Create a stress contour plot showing the *most appropriate stress* that should be analyzed acting across section C-C. Include automatic labeling of maximum and minimum stresses on this plot. This stress should correspond to the stress calculated in part (a)
- Use the **Probe** feature to produce a graph of the *most appropriate stress* across section C-C, beginning at the inside (concave) surface and continuing to the outside (convex) surface of the beam. (See the “*HINT*” at end of this problem for guidance when making this graph). Use the **Stress Plot** property manager to select the *appropriate stress* for this plot to enable comparison with manual calculations of part (a) above. Include a descriptive title and axis labels to this graph.

Below the graph, or on a separate page, either: (a) cut-and-paste a copy of the **Probe Results** table that includes values used for this comparison, or (b) click the



Save icon, located in the **Report Options** dialogue box. This action creates an Excel spreadsheet containing all values in the **Probe Results** table. This spreadsheet can be inserted onto the page beneath the current graph. In either table, circle the magnitudes of stress on the concave and convex surfaces.


After determining both classical and FEA results at section C-C, use equation [1] (repeated below) to compute the percent difference between these results at the concave and convex surfaces of the model.

$$\% \text{ difference} = \frac{(\text{FEA result} - \text{classical result})}{\text{FEA result}} * 100 = \quad [1]$$

- On the **Probe** graph created in part (d), label the distance of the neutral axis (neutral plane) from the concave edge of the model at section C-C. Write a brief statement indicating how this value was determined.
- Return to the plot produced in part (c). This time, use the **Probe** tool to sample stress magnitudes along the concave edge of the model beginning at point **A** and proceeding from node to node until reaching point **B**. Include a graph of these results with your analysis. Add a descriptive title and axis labels to this graph.

- g. Is the variation of stress through the middle of the model shown in the graph of part (d) expected? Why? Is the variation of stress shown on the graph of part (f) expected? Why? Is the location of the neutral axis (neutral plane) determined in part (e) located where it is expected to occur on the curved bracket model? Does the neutral axis of a curved member subject to axial load *always* occur at a location *like* that shown in the plot of part (e)? Explain why or why not.
- h. Return to the **Results** folder and produce a stress plot of **SI: Stress Intensity**. Compare the general shape of stress contours displayed on this plot with the photoelastic “fringes” shown in Fig. E 2-5. You should notice a similarity between shapes of the stress contours and the photoelastic fringe pattern. These two plots are related, but not identical. Therefore, no direct comparison of stress magnitudes is made here because comparison with photoelastic stress analysis results is beyond the scope of this text.

HINT: Because the mesh generation scheme within SolidWorks Simulation creates an optimized mesh, it is probable that (a) a straight line of nodes does *not* exist across the model at Section C-C, and (b) it is also *unlikely* that node points *occur exactly on a horizontal centerline through the center of curvature*. For these reasons, and to obtain the best estimate of stress on a straight line through the curved section, proceed as follows.

- Zoom in on a top view of the model at section C-C.
- In the Simulation toolbar, click the **Probe** icon . This action opens the **Probe Results** property manager.
- On the SolidWorks flyout menu, click the “+” sign to display the complete flyout menu.
- In the flyout menu, move the cursor over **Right Plane**. This action highlights an edge view of the **Right Plane**. This edge should provide assistance in locating nodes closest to a straight line through section C-C. Unfortunately, the line disappears when the cursor is moved, never the less it is very useful.

Textbook Problems

In addition to the above exercises, it is highly recommended that additional curved beam problems be worked from a design of machine elements or mechanics of materials textbook. Textbook problems provide a great way to discover errors made in formulating a finite element analysis because they typically are well defined problems for which the solution is known. Typical textbook problems, if well defined in advance, make an excellent source of solutions for comparison.