

# Analysis of Machine Elements

using COSMOSWorks 2008



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

**SDC**  
PUBLICATIONS

Schroff Development Corporation  
[www.schroff.com](http://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 COSMOSWorks *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

**Restraint** types within COSMOSWorks 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.

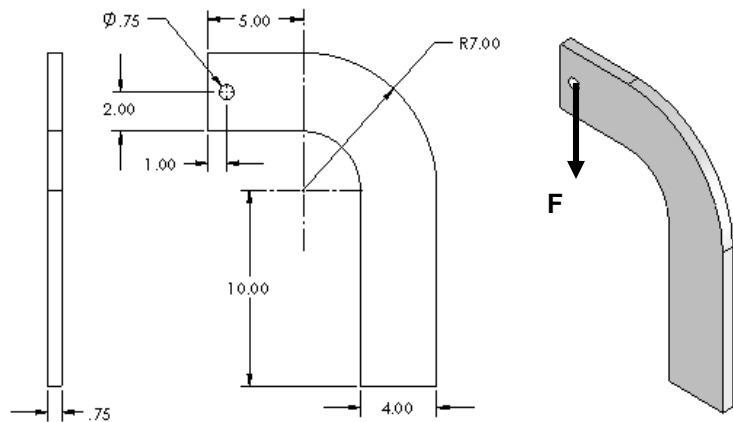


Figure 1 – Three dimensional model of curved beam.

### Design Insight

Numerous mechanical elements occur in the shape of initially curved beams. Examples include: C-clamps, punch-press frames, 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 additional capabilities of the COSMOSWorks software listed above.

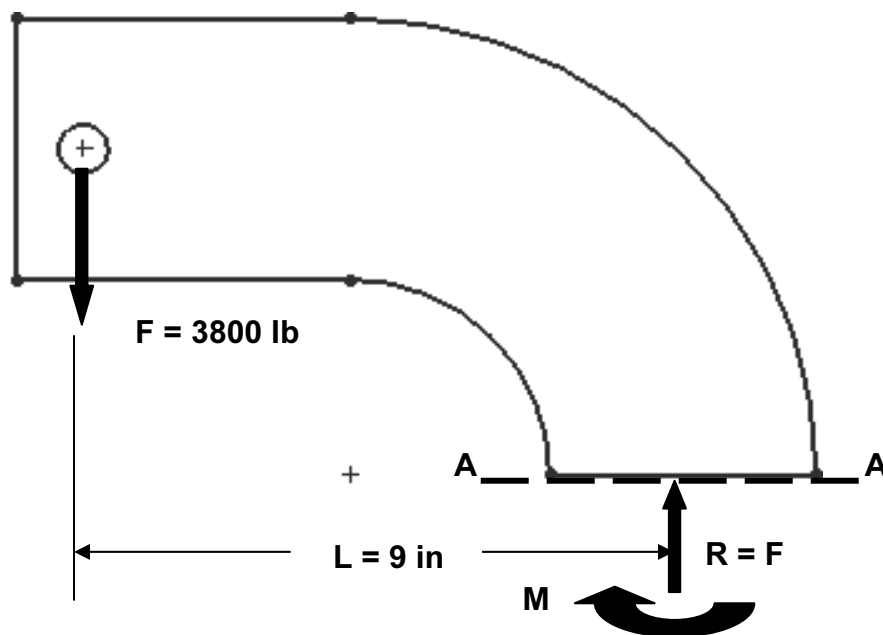


Figure 2 – Traditional free-body diagram of the upper portion of a curved beam model showing applied force  $F$  acting at a hole, and reactions  $R = F$ , and moment  $M$  acting on section A-A.

### Creating a Static Analysis (Study)

1. Open SOLIDWorks by making the following selections. These steps are similar to opening any Windows program. (Note: A “/” is used below to separate successive menu selection)

**Start / All Programs / SolidWorks 2008**

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

If a pop-up COSMOSWorks 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 **COSMOSWorks** listed in the main menu of the SolidWorks screen, click **Tools/Add-Ins...**, then in the **Add-Ins** window check  **COSMOS-Works 2008** in both the **Active Add-Ins** and the **Start Up** columns, then click [OK]. This action adds the **COSMOSWorks** icon to the top of the manager tree and to the main menu.

3. Click the COSMOSWorks analysis manager icon located at the top right-side of the manager tree shown in Fig. 3. This action switches the user into COSMOSWorks.

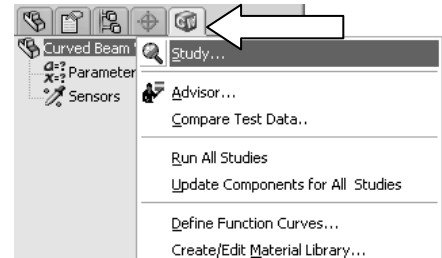


Figure 3 – Opening a Study in COSMOSWorks.

Because another goal of this analysis is to introduce use of COSMOSWorks icons, verify that the icons shown in Fig. 4 appear in the toolbar on your screen. If they do not appear, proceed as follows, otherwise skip to step 6.

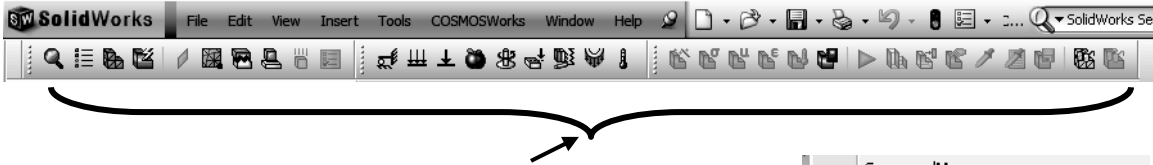


Figure 4 – Standard COSMOSWorks icons.

4. Right-click anywhere in the toolbar at top of the screen. This action opens the COSMOSWorks Command Manager menu shown in Fig. 5.
5. Within the Command Manager menu, select:  
**COSMOSWorks Loads**  
**COSMOSWorks Main**  
**COSMOSWorks Result Tools**


The above selections are shown “boxed” in Fig. 5.



Figure 5 – Opening standard COSMOSWorks toolbars.

Toolbars may be displayed in different locations on the screen than those shown in Fig. 4. Reposition the toolbars by clicking-and-dragging the “handles” located at the left-end of each toolbar.

## Analysis of Machine Elements using COSMOSWorks

6. In the COSMOSWorks manager, Fig. 3, right-click **Curved Beam** and from the pull-down menu select **Study...** Alternatively, from the COSMOSWorks toolbar, select the **Study** icon . The **Study** property manager opens (not shown).
7. In the **Name** dialogue box, replace **Study 1** by typing a descriptive name. For this example, type: **Curved Beam Analysis-YOUR NAME**. Recall that including your name along with the Study name ensures that it is displayed on each plot.
8. Examine other fields within the **Study** property manager to ensure that a **Solid Mesh** is specified and that **Static** is selected as the analysis **Type**.
9. Click **[OK]** ✓ (green check mark) to close the **Study** property manager.

The COSMOSWorks manager, seen in Fig. 6, is updated to show an outline for the current Study.

As in the previous example, the sequence of steps outlined in Fig. 6 is followed from top to bottom as the current finite element analysis is developed.

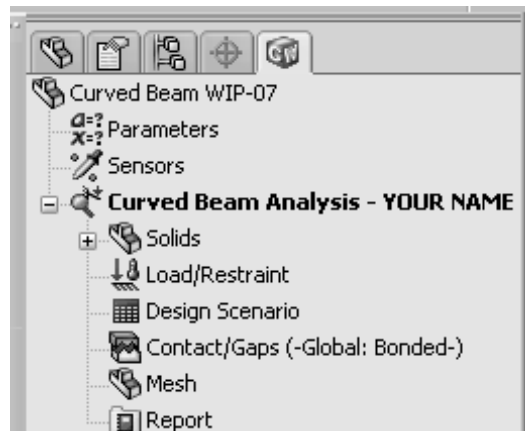



Figure 6 – Basic steps of a Study.

## Assign Material Properties to the Model

Part *material* is defined as described below.

1. In the COSMOSWorks manager tree, right-click the **Solids** folder and from the pull-down menu select **Apply Material to All...** Alternatively, click the “+” sign adjacent to the **Solids** folder followed by clicking the folder labeled **Curved Beam**. Then, in the newly added toolbar, select the **Apply Material to Selected Components** icon . The **Material** window opens as shown in Fig. 7.
  - a. Under **Select material source**, click to select  **From library files**.
  - b. From the pull-down menu, select **cosmos materials** (if not already selected).

- c. Click the plus sign next to the **Aluminum Alloys (85)** and scroll down to select **2014 Alloy**. The properties of 2014 Aluminum alloy are displayed in the right-half of the window.

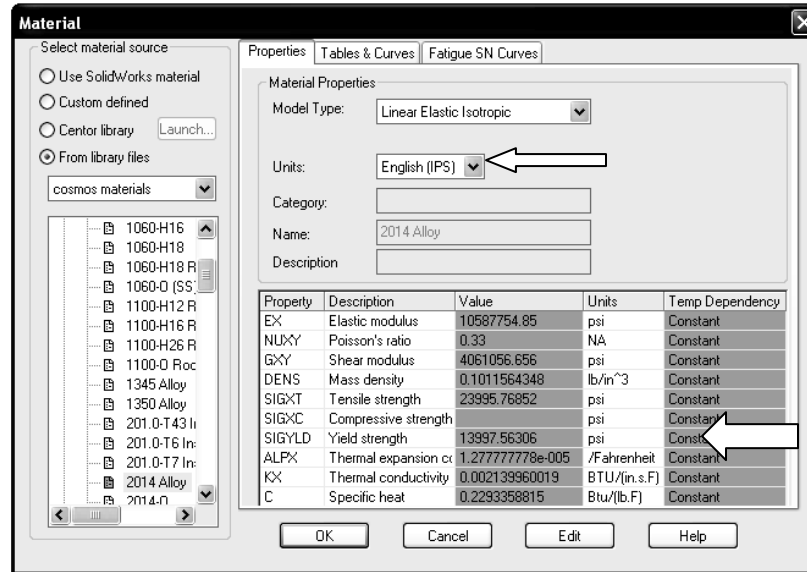


Figure 7 – Material properties are selected and/or defined in the **Material** window.

- d. In the right-half of this window, select the **Properties** tab (if not already selected). Change **Units:** to **English (IPS)**.

Note the material Yield strength denoted by **SIGYLD Yield Strength** is a relatively low 13997.563 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.

- e. Click **[OK]** to close the **Material** window. A check “✓” mark appears on the **Solids** folder to indicate a material has been selected.


**Aside:**

If at any point you wish to *change the material specification* of a part, such as during a redesign, right-click the **Solids** icon in the manager tree and select **Apply Material to All....** The **Material** window is opened and an alternative material can be selected or defined.

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 Restraints

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, recall when defining this **Study** that a *solid* mesh was specified. Therefore, because tetrahedral elements are used to mesh this model, “Immovable” restraints are applied to the bottom surface. Bear in mind that “fixed” restraints apply only to *Shell* elements. Shell elements are introduced in a later chapter. Proceed as follows to specify this restraint.

1. In the COSMOSWorks toolbar left-click the **Restraints** icon . Alternatively, in the COSMOSWorks manager, right-click **Load/Restraint** and select **Restraints...**

The **Restraint** property manager opens as shown in Fig. 8.

2. Under **Type**, click to open the pull-down menu and select **Immovable (No translation)**. This designation restricts translations in X, Y, and Z directions only.
3. The **Faces, Edges, Vertices for Restraint** field immediately below the pull-down menu 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, provided the  **Show preview** box is checked, restraint symbols appear as shown in Fig. 9. Also, **Face<1>** appears in the **Faces, Edges, Vertices for Restraint** field.

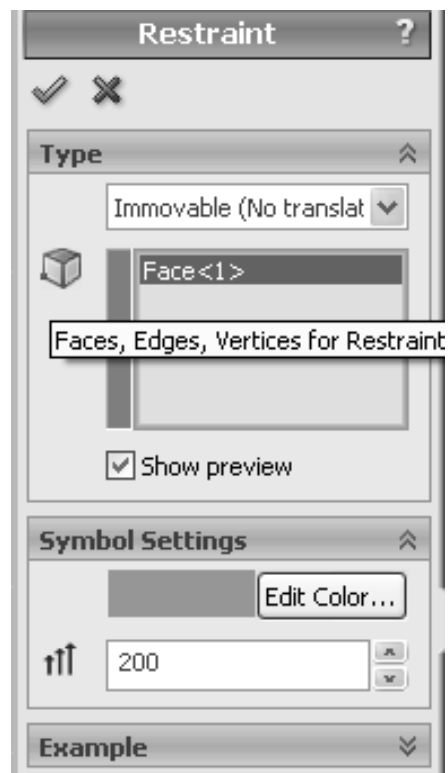



Figure 8 – **Restraint** property manager.

If an incorrect entity (such as a vertex, edge, or the wrong surface) is selected, right-click the incorrect item in the **Faces, Edges, Vertices for Restraint** field and select **Delete**, then repeat step 3.

4. Click the down arrow  to open the **Symbol settings** dialogue box.

5. Both color and size of the restraint symbols (vectors in the X, Y and Z directions) can be changed by altering values in the **Symbol Settings** field of Fig. 8. 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. 9 were arbitrarily increased to 200%. Experiment with this option.

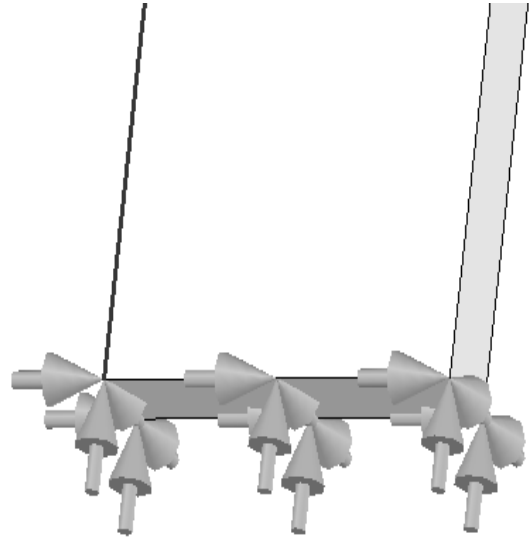


Figure 9 – **Immovable** restraints applied to bottom of the curved beam model.

6. Click **[OK]** ✓ (green check mark) at top of the **Restraint** property manager to accept this restraint. An icon named **Restraint-1** appears beneath the **Load/Restraint** folder in the COSMOSWorks manager.

**Aside:**

**Immovable** restraint symbols shown in Fig. 9 appear as simple arrows or vectors with a straight shaft. These symbols indicate that *displacements* in the X, Y and Z directions are restrained (i.e., prevented).

**Fixed** restraints are introduced in Chapter 4 where a small disk is added to the tail of each restraint symbol. This difference of symbols indicates that *rotations* about the X, Y and Z axes are also restrained. Watch for this subtle difference in future examples.

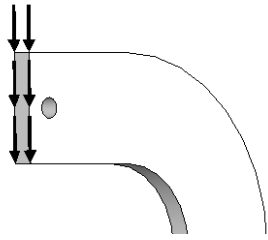
## Applying Load(s)

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).

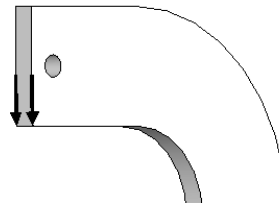


Figure (a) – Force applied to left surface.

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 of the hole surface 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 geometries of the contacting parts (i.e., relative diameters of the pin and hole) and material properties (i.e., hard versus soft contact surfaces of either the pin or the beam). 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. 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. 10.
2. On the graphics screen it may be necessary to click the “+” sign adjacent to the SolidWorks **Curved Beam** icon to display the complete “flyout” menu as illustrated in Fig. 10. Continue to refer to Fig. 10 in steps 3 through 5 below.

3. 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.
4. 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 model<sup>1</sup> in Fig. 10.

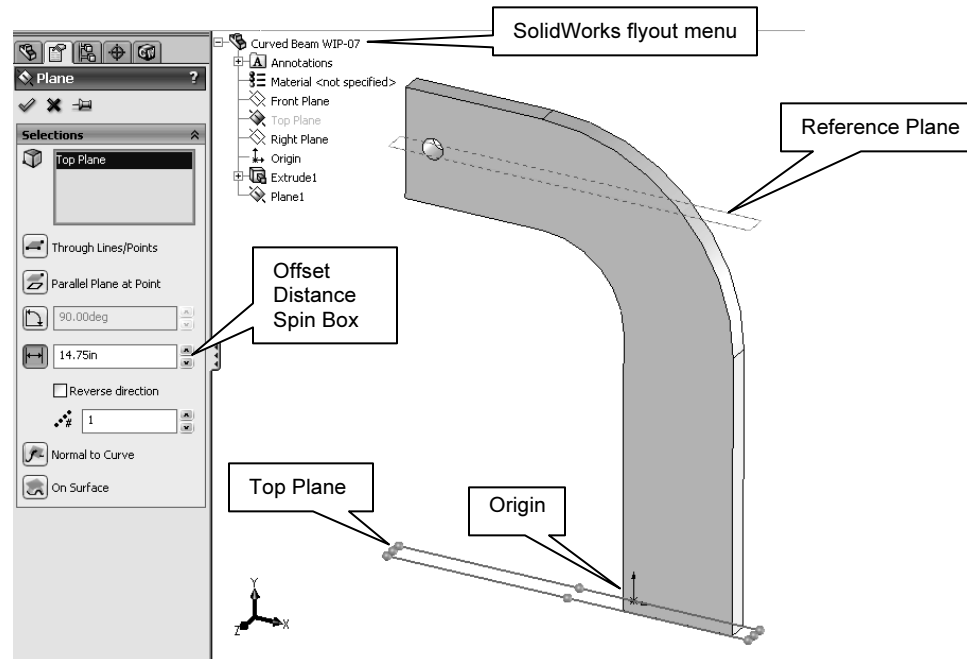


Figure 10 – The **Plane** property manager and various selections made to create a reference plane that passes through the bottom of the hole.

5. In the **Plane** property manager, 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!

<sup>1</sup> 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.

6. 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.

7. 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. 11.
8. 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).
9. 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.

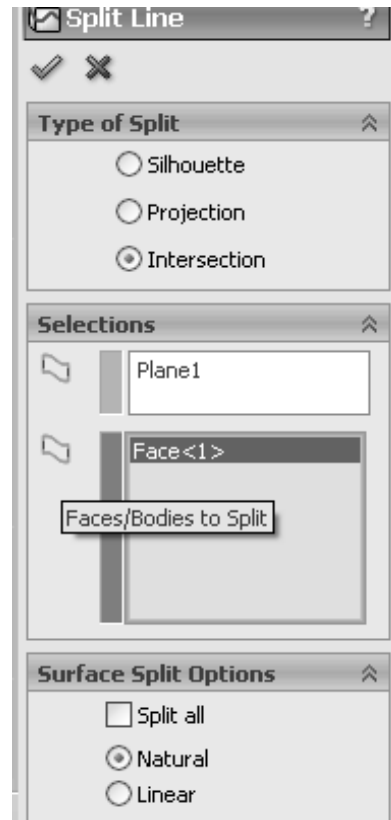


Figure 11 – **Split Line** property manager showing selections.

10. 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 12 shows a partial image of the model with *Split Lines* appearing where **Plane1** intersects the bottom of the hole.

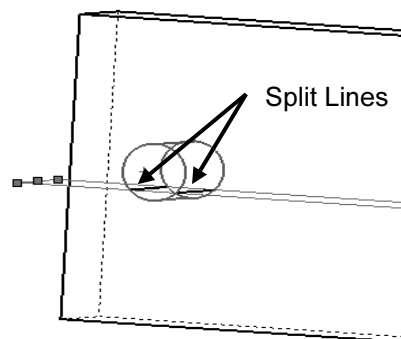



Figure 12 – Close-up view of hole showing *Split Lines* where **Plane1** intersects near the bottom of hole.

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

11. In the **Surface Split Options** dialogue box, select  **Natural**. A **Natural** split follows the contour of the selected model surface.
12. Click **[OK]** ✓ to close the **Split Line** property manager.
13. 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 COSMOSWorks toolbar, select the **Force** icon . Alternatively, in the COSMOSWorks manager, right-click the **Load/Restraint** icon and from the pop-up menu select **Force....** A partial view of the **Force** property manager appears in Fig. 13.

2. Under **Type**, click to select  **Apply force/moment**. 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 **Type** dialogue box.
4. Next, click to activate the second field from the top of the **Type** 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. 14.

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

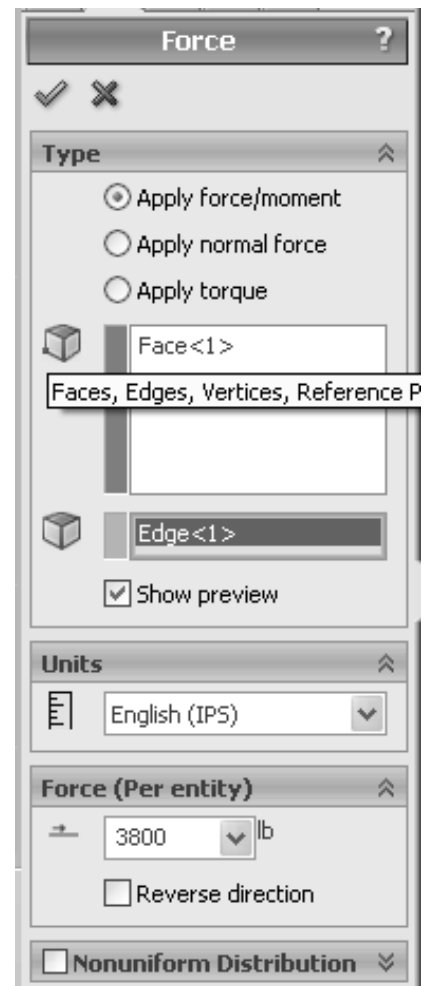


Figure 13 – Specifying a force and its direction on the hole bottom.

6. In the **Force (Per entity)** field, 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** property manager. An icon named **Force-1** appears beneath the **Load/Restraint** folder of the COSMOSWorks manager.

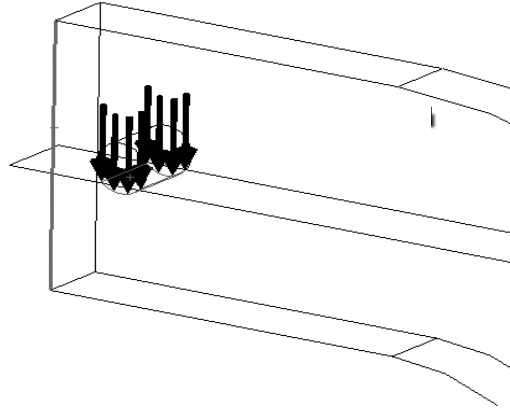




Figure 14 – Downward force applied between *Split Lines* on bottom of hole. A wireframe view of model is shown.

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

## Meshing the Model

1. In the COSMOSWorks toolbar, select the **Mesh** icon . Alternatively, in the COSMOSWorks manager, right-click **Mesh** and from the pull-down menu select **Create Mesh...** The **Mesh** property manager opens as shown in Figs. 15 (a & b).
2. In the **Mesh Parameters:** dialogue box, set **Unit** to **in** (if not already selected). Accept remaining default settings (i.e. mesh size and tolerance) in this dialogue box. These system-determined values usually provide sufficient definition for an initial analysis on a non-complex part.
3. Click the down arrow  to open the **Options** dialogue box. Most of the following selections should appear as default settings. These settings should produce a reasonably good quality mesh. However, verify that the settings are as listed below and only change them if they differ.

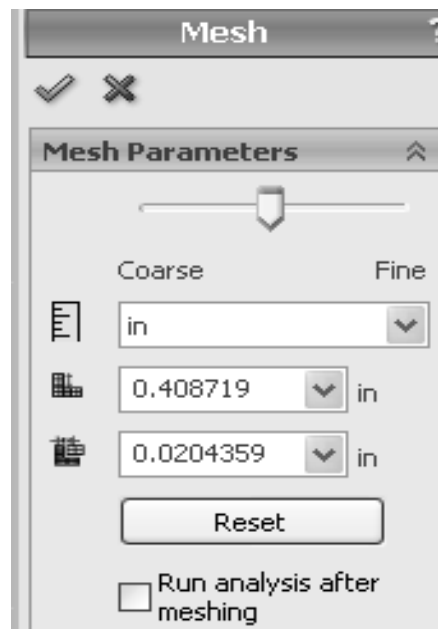


Figure 15 (a) – **Mesh** property manager showing system default **Mesh** settings applied to the current model.

4. Under **Quality**, select  **High**.
5. Under **Mesher:**, select  **Standard**.
6. Under **Mesher options:** check  **Jacobian check for solid:** and the pull-down menu should be set to **4 points**.
7. Click the **[Apply]** button to place these settings into effect.
8. 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, COSMOSWorks displays the meshed model shown in Fig. 16. Also, a check mark “✓” appears on the **Mesh** folder to indicate meshing is complete.

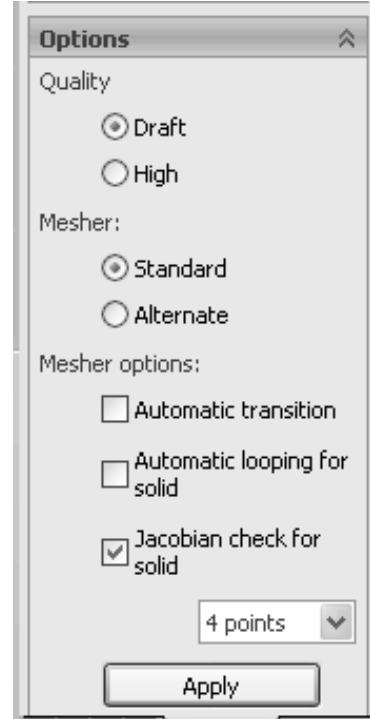


Figure 15 (b) – View of the **Options** portion of the **Mesh** property manager.

OPTIONAL:

9. 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 12554 nodes and 7213 elements (numbers may vary slightly due to the automated mesh generation procedure).

Rotate the model as illustrated in Fig. 16 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.

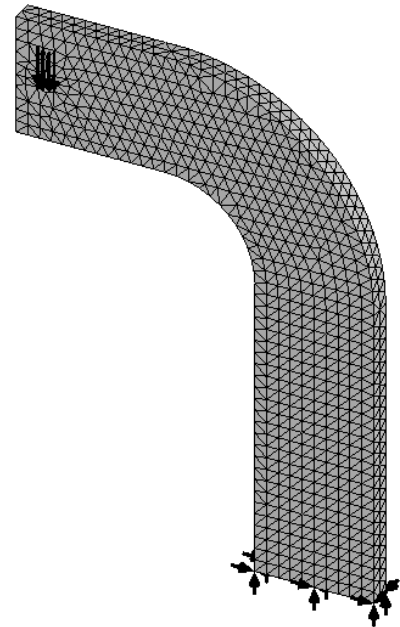


Figure 16 – Curved beam with mesh and boundary conditions illustrated.

**Reminder**

Recall that it is permissible to define material properties, restraints, forces, 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. To run the analysis, click the **Run** icon . Or, right-click the **Curved Beam Analysis - YOUR NAME (-Default-)** folder highlighted in Fig. 17. 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 COSMOSWorks 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. 17. If these folders do *not* appear, follow steps (a) through (e) outlined on page 1-13 of Chapter 1.

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

2. Right-click the **Stress1 (-vonMises-)** folder 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 change the **Units** from **N/m<sup>2</sup>** to **psi** using the pull-down menu.

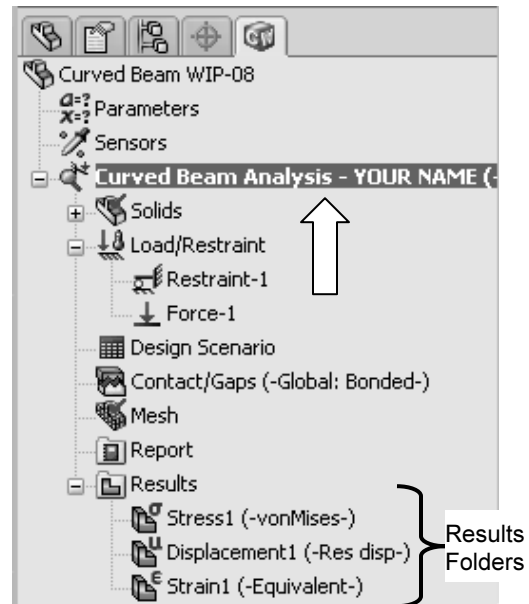


Figure 17 – **Results** folders created during the Solution process.

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

## 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 based on classical stress equations. *Checking results is a necessary step in good engineering practice!*

1. In the COSMOSWorks 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 18 reveals an image *similar* to what currently appears on the screen. The following steps convert your current screen image to that shown in Fig. 18.

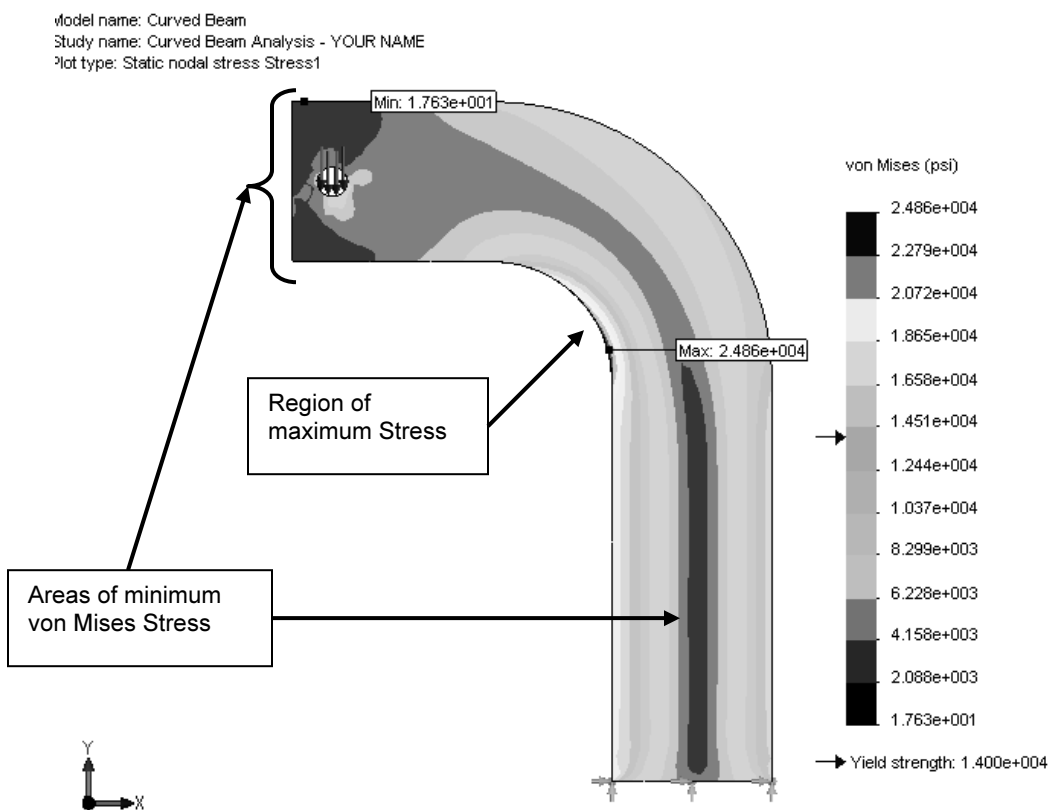


Figure 18 – Front view of the curved beam model showing von Mises stress *after* making changes outlined below. Note arrows indicating Yield Strength on the stress scale at right.

NOTE: *Stress contour plots are printed in black, white, and gray 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. 19.

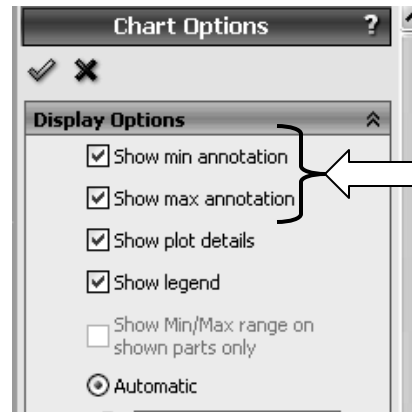


Figure 19 – Upper portion of **Chart Options** property manager showing current selections.

- 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. 20.

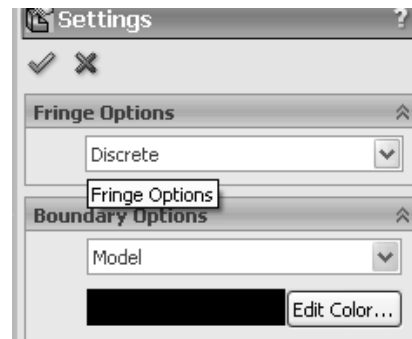



Figure 20 – Selections in the **Settings** property manager.

- 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 COSMOSWorks toolbar, select the **Stress...**  icon. Alternatively, right-click **Stress1 (-vonMises-)** and from the pull-down menu, select **Edit Definition....** Either action opens the **Stress Plot** property manager, shown in Fig. 21.

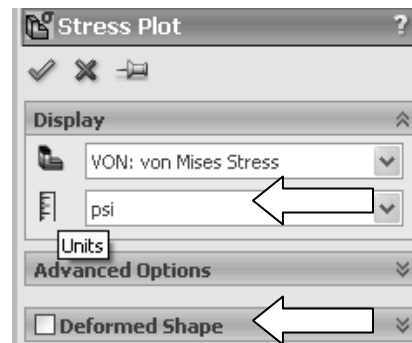


Figure 21 – **Stress Plot** property manager.

- In the **Deformed Shape** property manager, 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. 18.

**OBSERVATIONS:**

- Areas of low stress (dark blue) occur at the top-left side of the model and also run through the vertical center of the model. The lowest stress is approximately  $1.763e+001 = 17.63$  psi. Regions of high stress are indicated by red areas. The maximum stress indicated is  $2.486e+004 = 24,860$  psi, which occurs along the concave surface.
- Material *Yield Strength*  $\approx 14000$  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. 18 (“+” values typically indicate tension) when 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 in 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 COSMOSWorks 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.”<sup>2</sup> In other words, for the most complex state of stress that one can imagine

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

(e.g., a three-dimensional stress element subject to a combination of 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”<sup>3</sup> 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. 18. 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 ( $\sigma_y$ ) was the only stress component that would yield favorable comparisons with stresses calculated using classic equations. It is left as an exercise to determine what the *appropriate stress* is to show variation from compression on the concave surface to tension on the convex side of the curved beam model.

## 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 that its neutral axis lies closer to the center of curvature than does its centroidal axis. This can be observed in Fig. 22. By definition the centroidal axis,

---

<sup>3</sup> Ibid

identified as  $r_c$  is located half-way between the inside and outside radii of 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. 22. 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 =  $r_i + w/2 = 3.00 + 2.00 = 5.00$  in

$r_n$  = radius to the neutral axis =  $w/\ln(r_o/r_i) = 4.00/\ln(7.00/3.00) = 4.72$  in.

[determined by equation for a rectangular cross-section of a curved beam]

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

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

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

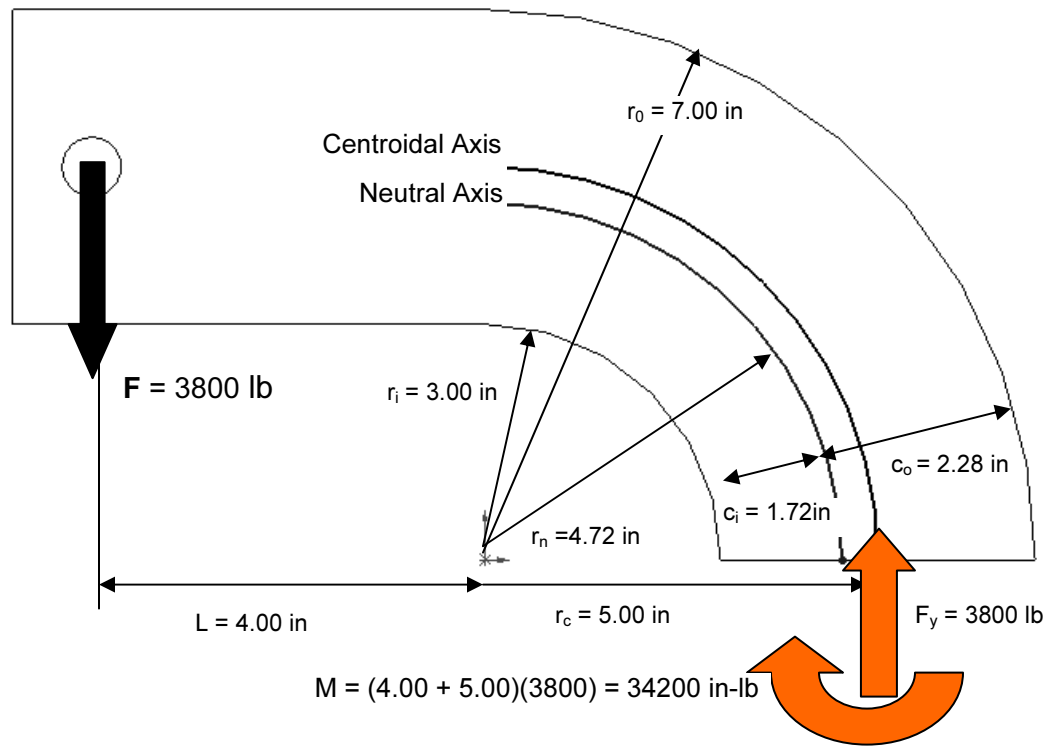


Figure 22 – Geometry associated with calculation of stress in a curved beam.

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 “±” 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 the “+” sign associated with the first term in equation [2]. Reaction force **F<sub>y</sub>** 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.

Stress at the inside 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 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. 22. Therefore, it is logical that the Finite Element Analysis stress in the Y-direction ( $\sigma_y$ ) should be compared with values computed using equations [1] and [2] above. You are encouraged to produce a plot of stress  $\sigma_y$  on your own. However, abbreviated steps are outlined below if guidance is desired.

1. In the COSMOSWorks toolbar, click the **Stress...**  icon to open the **Stress Plot** property manager. Alternatively, right right-click the **Results** folder and from the pull-down menu select **Define Stress Plot...**
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  $\sigma_y$  is displayed. If the plot does not appear, right-click **Stress2 (-Y normal-)** and select **Show** from the pull-down menu.

Next, alter the display to show the mesh and discrete fringes on the model. Proceed on your own and skip to step 8 or, if guidance is desired, follow the steps 5 through 7 below.

5. Click the **Settings...** icon  to open the **Settings** property manager. Alternatively, right-click **Stress2 (-Y normal-)** and from the pull-down menu select **Settings...**
6. In the **Fringe Options** pull-down menu, select **Discrete** as the fringe display mode and in the **Boundary Options** dialogue box, select **Mesh**.
7. Click **[OK]** ✓ to close the **Settings** property manager.
8. Zoom in on the model to where the curved beam section is tangent to the straight, vertical section as shown in Fig. 23.

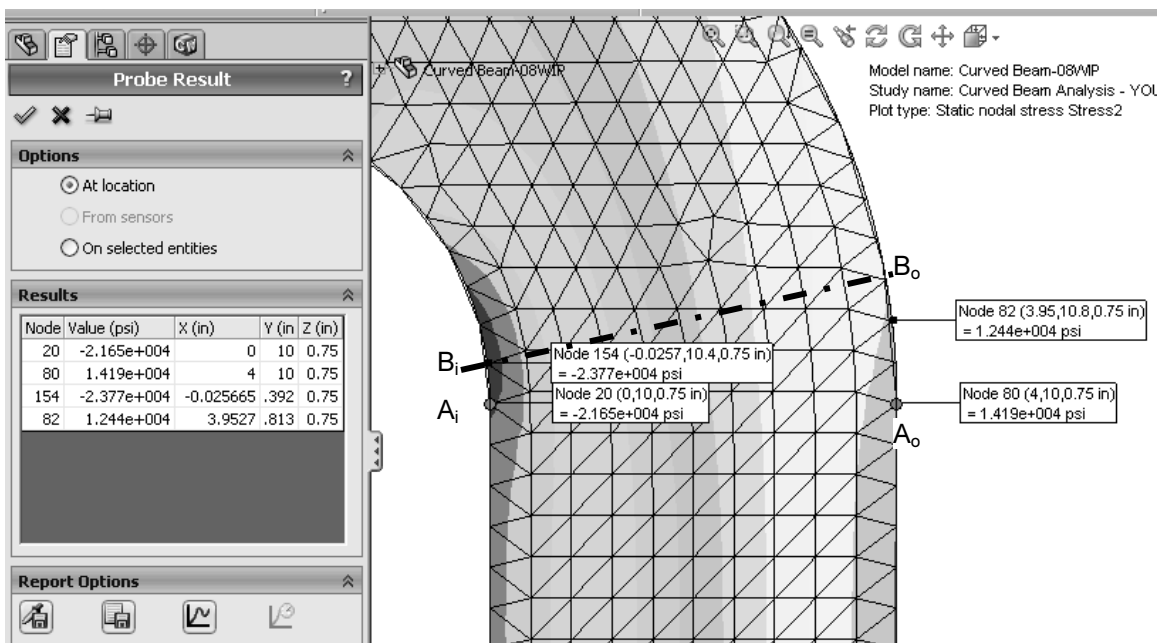



Figure 23 – Using the **Probe** tool to determine stress magnitudes at locations on the concave and convex sides of the curved beam model.

9. In the COSMOSWorks toolbar 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.
10. In the **Options** dialogue box, select  **At location** (if not already selected).
11. Move the cursor over the straight vertical edges on the left and right sides of the model. Each edge is highlighted (by a red line) 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<sub>i</sub>** and **A<sub>o</sub>** in Fig. 23. If an incorrect node is selected, simply click  **At**

**location** in the **Options** dialogue box to clear the 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 ( $\sigma_y$ ), and the **X, Y, Z** coordinates of the selected node. 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 ( $\sigma_y$ ) from classical and finite element methods at Section A-A.**

Location	Manual Calculation (psi)	Probe Tool Results (psi)	Percent Difference (%)
Point <b>A<sub>i</sub></b>	-24610	-21650	13.7%
Point <b>A<sub>o</sub></b>	11990	14190	15.5%

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 15% 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 in this transition region.

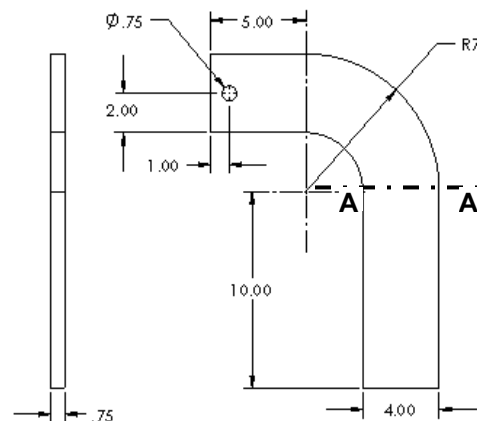


Figure 1 (Repeated) – Basic geometry of the curved beam model.

Given the above observations, we next proceed to sample stress magnitudes at Section B<sub>i</sub>-B<sub>o</sub> in Fig. 23, which is located slightly above the transition region. Proceed as follows.

12. 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.
13. Next, on the convex side of the model, select the *second* node above the previously selected node.

Observe stress magnitudes listed in the **Results** dialogue box and compare them to values listed in Table II below. Nodes **B<sub>i</sub>** and **B<sub>o</sub>** 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 ( $\sigma_y$ ) for classical and finite element methods at Section B<sub>i</sub>-B<sub>o</sub>.**


Location	Manual Calculation (psi)	Probe Tool Results (psi)	Percent Difference (%)
Point B <sub>i</sub>	-24515	-23770	3.1%
Point B <sub>o</sub>	11940	12440	4.0%

14. 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

COSMOSWorks provides a convenient means for the stress analyst 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 COSMOSWorks toolbar, click the **Design Check Wizard** icon . Alternatively, right-click the **Results** folder and from the pull-down menu, select **Define Design Check Plot...**. The **Design Check** property manager opens as shown in Fig. 24 and displays the first step of a three step procedure.

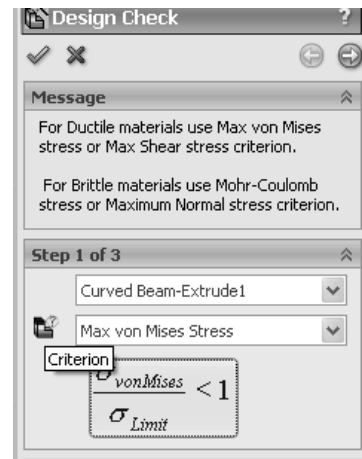


Figure 24 – **Design Check**, Step #1 of safety factor check.



## Analysis of Machine Elements using COSMOSWorks

2. In the **Message** dialogue box read the guidelines for selecting the proper failure criterion to be applied to the current model. Additional details about each failure criterion are provided below.
3. In the upper pull-down menu of the **Step 1 of 3** dialogue box, select **Curved Beam-Extrude1** as the model to be analyzed.
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 von Mises 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 COSMOSWorks. This theory is also regarded as the least accurate of the methods available.

5. Because the curved beam is made of aluminum and because a good estimate of safety factor is desired, choose **Max vonMises Stress** (if not already selected) from the pull-down menu.

Also, immediately below this field notice that the Design Check goal is currently defined as

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

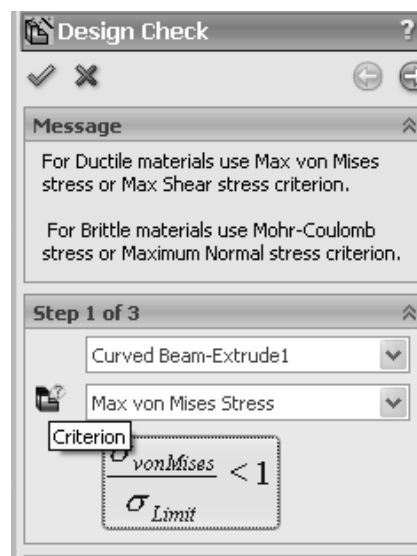



Figure 24 (repeated) – First **Design Check** property manager. The comparison stress (failure criteria) is selected here.

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. Proceed to the second step by clicking the right facing arrow button  at the top of this property manager. The **Design Check, Step 2 of 3** dialogue box appears as shown in Fig. 25.
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).

Notice that the Yield and Ultimate strengths for the model appear near the bottom of this dialogue box. Also, the bottom box lists the maximum von Mises stress found during the solution. By examining the values in these two boxes it is obvious that, somewhere in the part, the maximum von Mises stress *exceeds* the material Yield Strength.

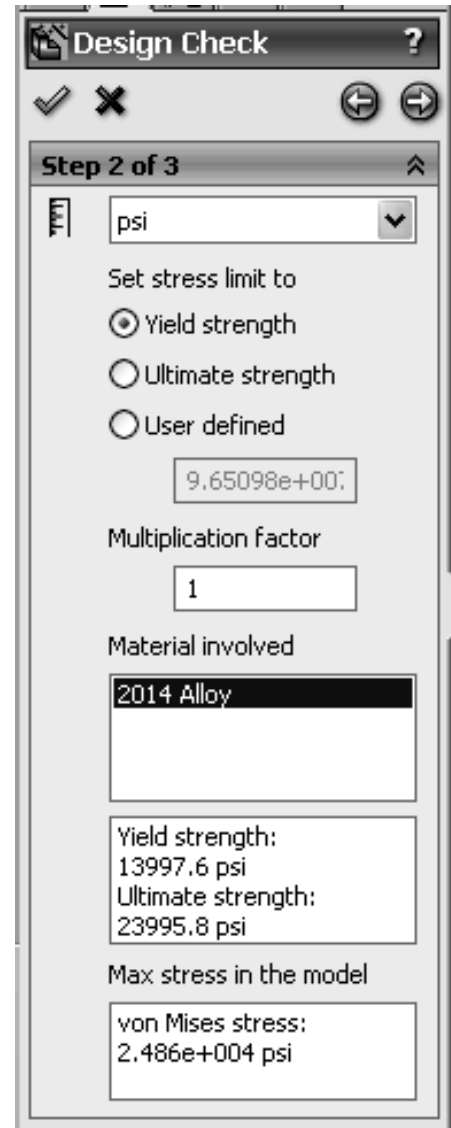



Figure 25 – Step 2 of 3 in the **Design Check** process.

### 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 library.

## Analysis of Machine Elements using COSMOSWorks

- Click the right facing arrow button  at top of this property manager to proceed to **Step 3 of 3** in the **Design Check** property manager shown in Fig. 26.

Three 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 part.
- **Non-dimensional stress distribution** – Compares stress level at any location in the part to the maximum stress level and plots the ratio of these stresses as a number between 0 and 1 throughout the 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.

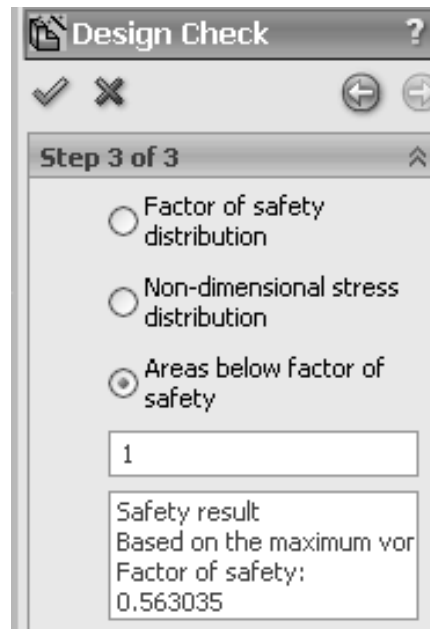


Figure 26 – Redefinition of *Safety Factor* and values to be displayed on the new plot.

- Beneath **Step 3 of 3**, select  **Areas below factor of safety** and type “1” in the 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 **0.563035** 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 **Design Check** window. That is:

$$\frac{\sigma_{Limit}}{\sigma_{vonMises}} = \frac{13997}{24860} = 0.56303$$

- Click **[OK]**  to close the **Design Check** property manager. A new plot folder, named, **Design Check1 (-Criterion: 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.

12. Right-click **Design Check1 (-Criterion: Max von Mises Stress-)**, and from the pull-down menu, select **Chart Options...**. The **Chart Options** property manager opens.
13. 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. 27. As expected, this location corresponds to the location of highest stress previously illustrated in Fig. 18.

The figure now on the screen should correspond to Fig. 27. 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. 27, provides a “key” to interpret safe and unsafe regions on the model.

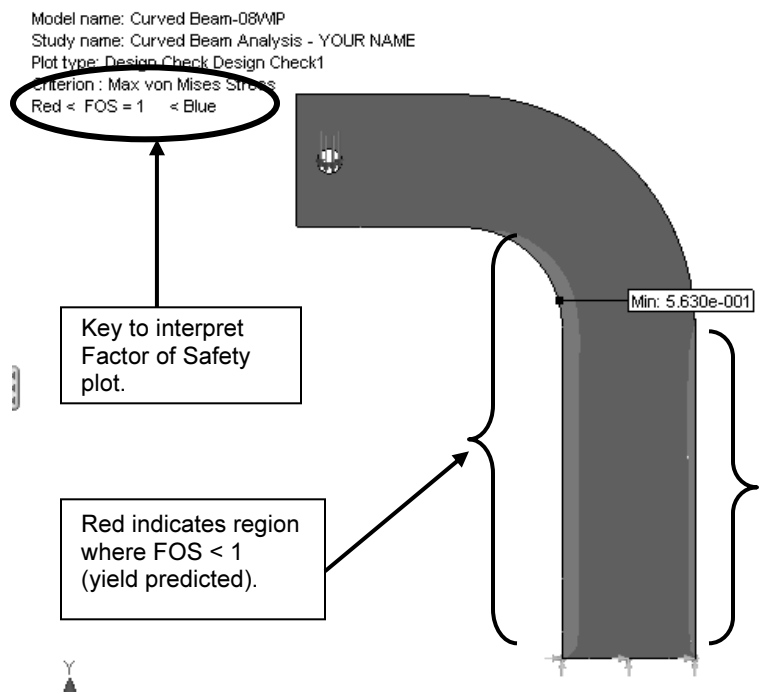


Figure 27 – Curved beam model showing areas where FOS  $> 1$  (safety predicted) and where FOS  $< 1$  (yield predicted).

14. Double-click **Design Check1 (-Criterion: Max von Mises Stress-)** and repeat steps 3 through 11 above, but this time set the **Areas below factor of safety to 2** instead of **1**, in step 10. 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.

### 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.

## Reaction Forces

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 COSMOSWorks 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. 28.
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.



Figure 28 – 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. Force reactions in the X, Y, Z directions found in the **Selection** column are identical to those for the **Entire Model**. This result is expected since the entire model is restrained at only this one location.

Results interpretation is as follows:

**SumX:** -0.109 (essentially zero) No force applied to model in X-direction.  
**SumY:** 3800.2 (essentially 3800 lb = the applied force)  
**SumZ:** -0.10173 (essentially zero) No force applied to model in Z-direction.  
**Resultant:** 3800.2 (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. 28.

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 locations that might be restrained 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. 29, opens and asks, “**Save changes to Curved Beam Analysis?**” Select the **[No]** button. This closes the file without saving any changes.

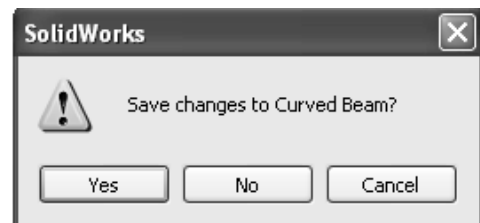


Figure 29 – **SolidWorks** window prompts users to either save changes or not.

## 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. Maximum benefit is realized by working all parts. However, in an academic setting, it is likely that parts of problems will be assigned or modified to suit specific course goals.

1. C-clamps, like that illustrated below, must pass minimum strength requirements before they are 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 criterion, 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
- Restraint: **Immovable** applied to the upper gripping surface.
- Force: **950 N** applied normal to lower gripping surface.

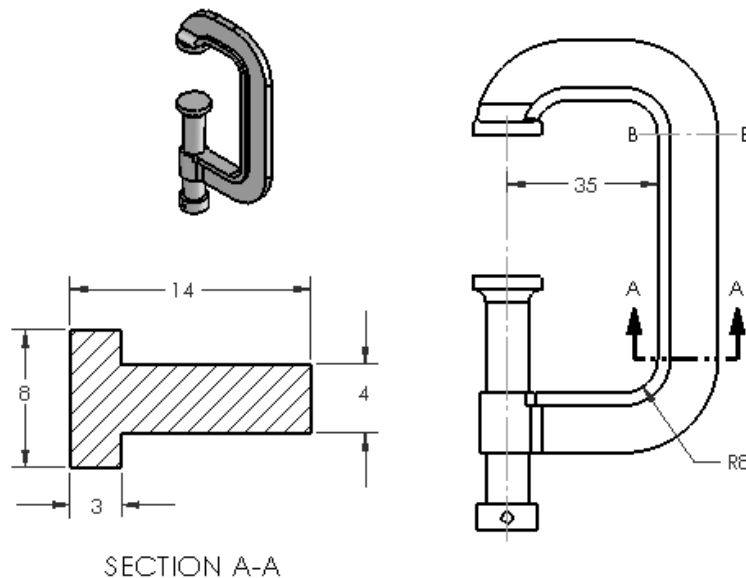


Figure E 2-1 – C-clamp frame and cross-section dimensions. Stress to be determined at Section A-A.

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. When using the **Probe** feature, begin at the inside (left) 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.
2. 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 round holes that pass through the lower-left and 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 **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)**
  - Restraint: **Immovable** applied to inside of left hole.
  - Force: **50 lb** applied to inner surface of right-hand hole (split lines needed).



## Analysis of Machine Elements using COSMOSWorks

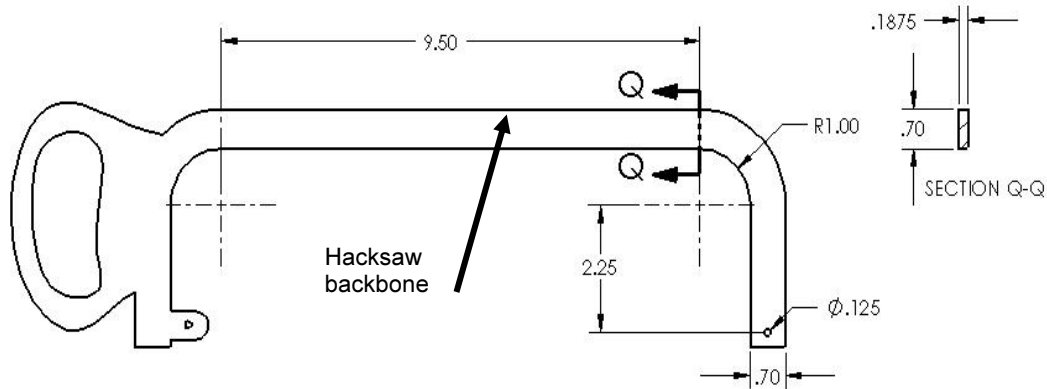


Figure E 2-2 – Basic geometry of a hacksaw frame. Stress is to be determined at Section Q-Q.

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 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. 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.
- 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.

3. 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.
- Restraint: **Immovable** applied on the inclined surface.
- Force: **8600 N** in the X-direction applied on the right, inside surface of the

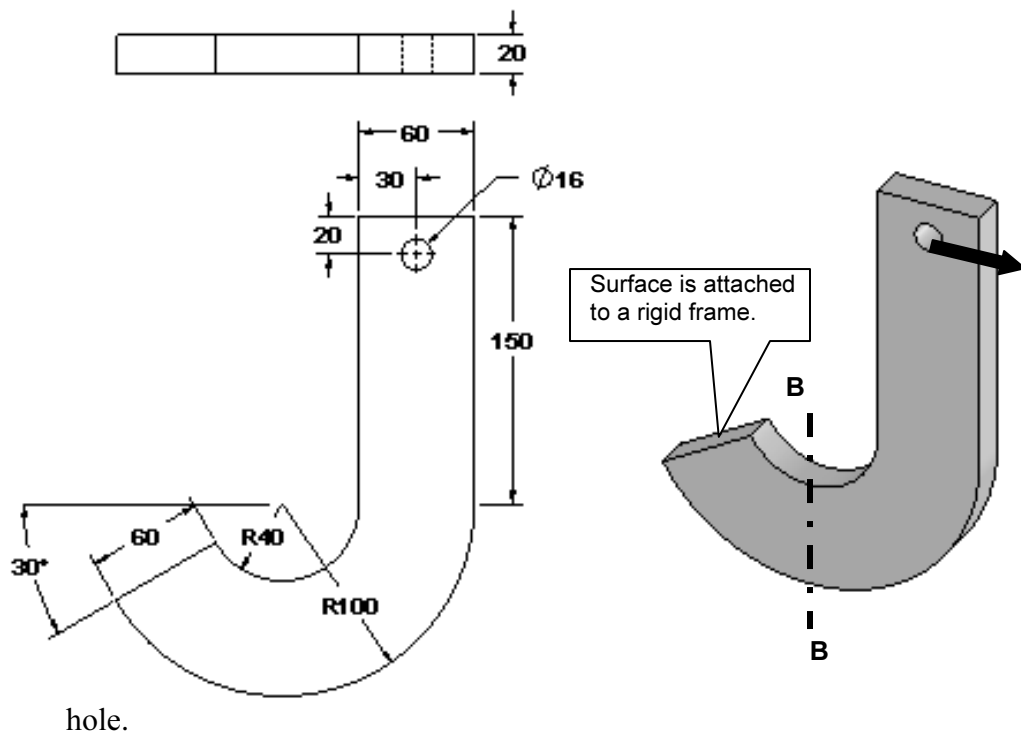


Figure E 2-3 Dimensioned view of the Anchor Bracket. Stress is to be determined at Section B-B.

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.

## Analysis of Machine Elements using COSMOSWorks

- b. Include a zoomed-in image of the hole so that the applied load can clearly be seen to act between user specified *Split Lines*.
- c. Create a stress contour plot of von Mises stress in the saw backbone. Include automatic labeling of maximum von Mises stress on the plot.
- d. 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.
- e. Use the **Probe** feature to produce a graph of the *most appropriate stress* across section B-B, beginning at the inside (concave) surface and continuing to the outside (convex) surface of the bracket 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. Also, below the graph, cut-and-paste a copy of the **Probe Results** table showing values used in this comparison. Use equation [1], repeated below, 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 COSMOSWorks creates an optimized mesh, it is probable that (a) a straight line of nodes does *not* exist across Section B-B (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 at on a vertical line through the curved section, proceed as follows.

- Zoom in on the curved section of the model.
- Open the **Probe Result** property manager. Simultaneously the SolidWorks flyout menu appears. Click the “+” sign to display this menu.
- In a front view of the model, move the cursor over the **Right Plane** in the flyout menu. This action highlights an edge view of the **Right Plane** to assist in locating nodes closest to a vertical line at Section B-B.

### 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.