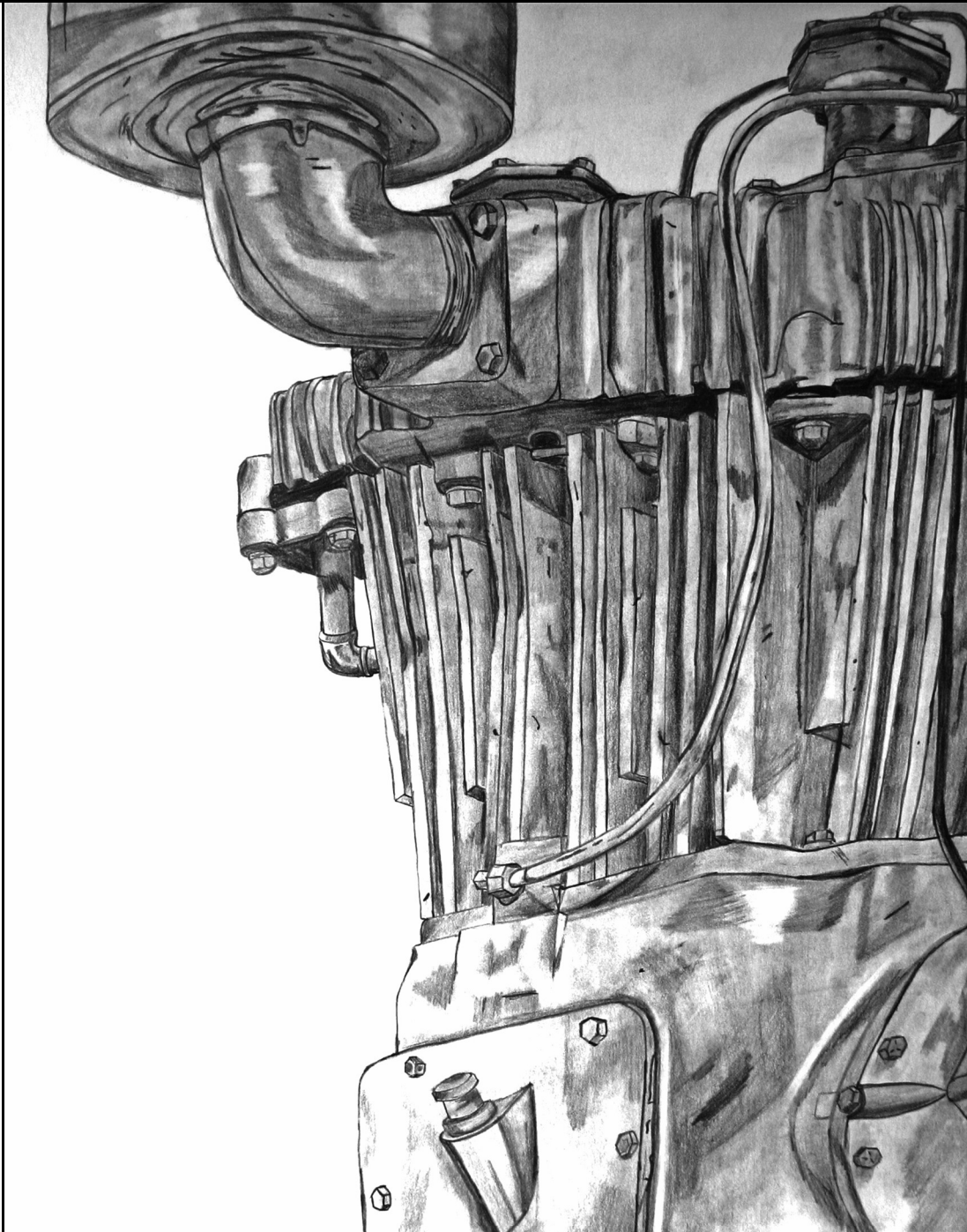


Analysis of Machine Elements using COSMOSWorks Professional COSMOSWorks 2006

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



SDC
PUBLICATIONS

Schroff Development Corporation

www.schroff.com
www.schroff-europe.com

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, in those areas where new information or procedures are introduced, additional details are included. For consistency throughout this text, a common procedure is used for the solution of each problem.

Learning Objectives

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

- 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 for 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 $F_y = 3200$ 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 moments about the X, Y, Z axes are considered fixed. However, recall from Chapter 1 that names for **Restraint** types within COSMOSWorks also depend on the type of element to which they are applied. Therefore, because solid tetrahedral elements are used in this example, **Immovable** restraints are used.

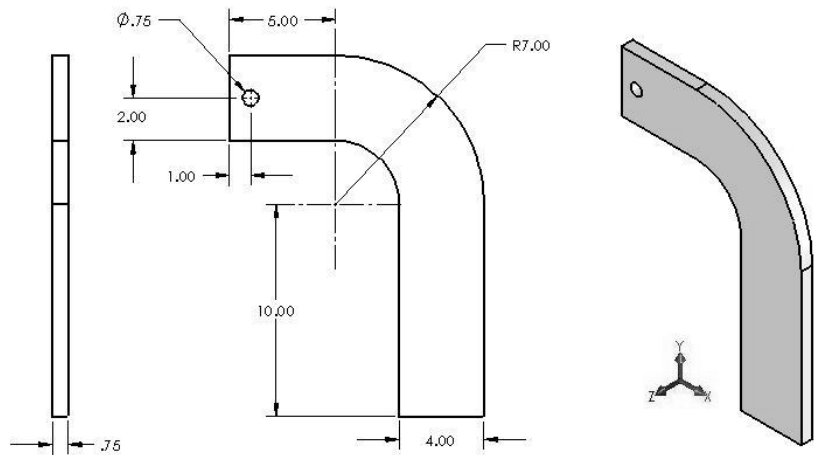


Figure 1 – Three dimensional model of curved beam.

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 the maximum bending moment M and reaction force $R = F$ at that location. Accordingly, classical equations for stress in a curved beam predict maximum stress at this location. In this example, the validity of this common assumption is investigated while exploring the additional capabilities of COSMOSWorks software listed above.

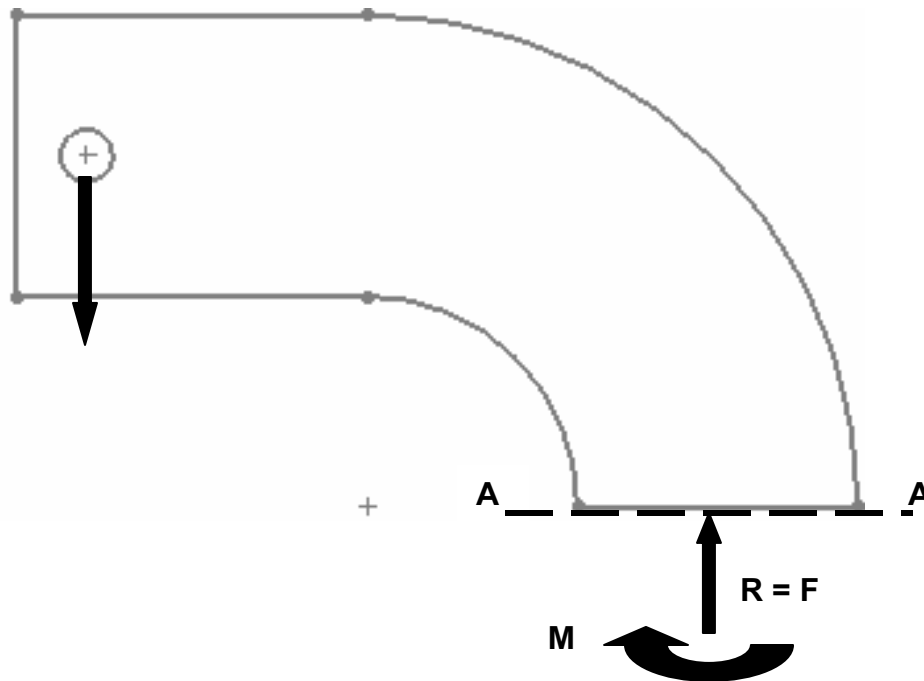


Figure 2 – Traditional free-body diagram of the upper portion of a curved beam model.

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 to separate successive menu selection)

Start / All Programs / SolidWorks 2006

2. When SolidWorks is open, select **Files / Open**. Then use procedures common to your computer environment to 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 **COSMOSWorks 2006** and click [OK]. This action causes the **COSMOSWorks** icon to appear at the top of the manager tree. Also, **COSMOSWorks** is added 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 activates COSMOSWorks.
4. In the COSMOSWorks Analysis Manager, Fig. 3, right-click **Curved Beam** and from the pull-down menu, select **Study...** The **Study** property manager opens (not shown).

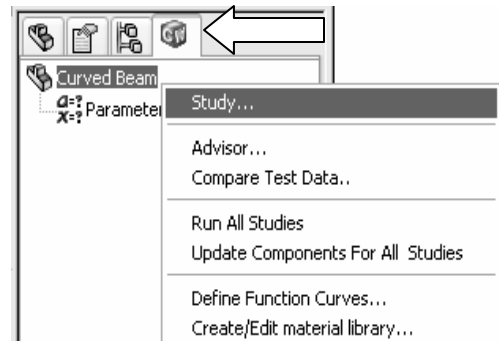


Figure 3 – Opening a COSMOSWorks Study.

5. In the field beneath **Name**, replace “**Study 1**” by typing a descriptive name. For this example type: “**Curved Beam Analysis**” (omit quotes).
6. Examine other fields within the **Study** property manager to ensure that a **Solid mesh** is selected and that **Static** is shown as the analysis **Type**.
7. Click [OK] to close the **Study** property manager.

The COSMOSWorks analysis Manager seen in Fig. 4, shows an outline for the current Study.

As in the previous example, the sequence of steps outlined in Fig. 4 is followed from top to bottom as the finite element analysis is developed.

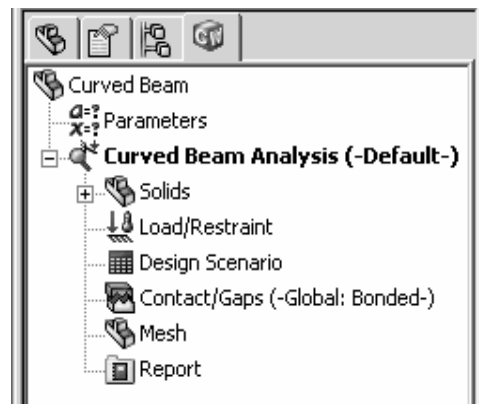


Figure 4 – Basic steps of a Study.

Assign Material Properties to the Model

Part *material* is defined first 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...** The **Material** window opens as shown in Fig. 5.
 - 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 **Aluminium Alloys** and scroll down to select **2014 Alloy**. The properties of 2014 Aluminium alloy appear in the right-half of the window.

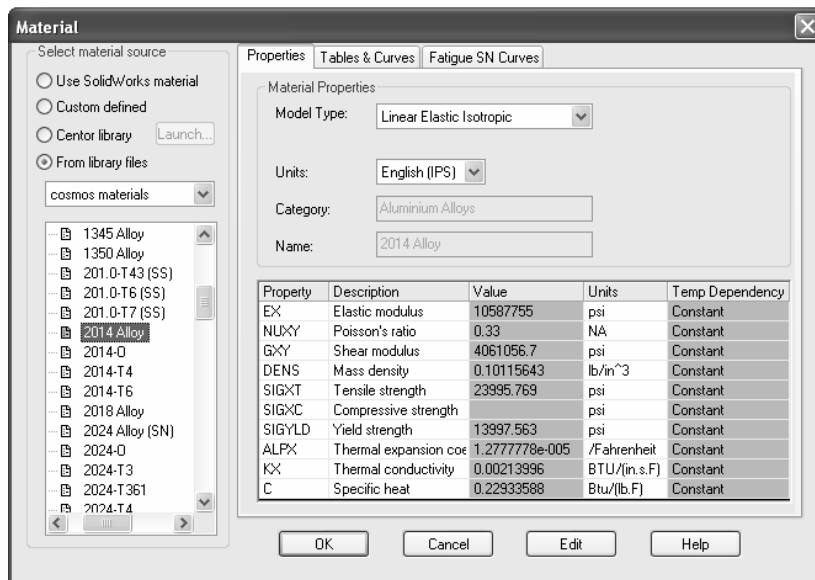


Figure 5 – Material properties are selected and/or defined in the **Material** window.

- 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** is a relatively low 13997.536 psi (approximately 14000 psi). A material with 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 above 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 part icon at top of the manager tree and select **Create/Edit Material library...** 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. Proceed as follows to specify this restraint.

1. In the COSMOSWorks manager tree, right-click **Load/Restraint** and select **Restraints...**

The **Restraint** property manager opens as seen in Fig. 6.

2. Under **Type**, click to open the pull-down menu and select **Immovable (No translation)** from the list of available restraint types. This designation restricts translations in X, Y, and Z directions only.

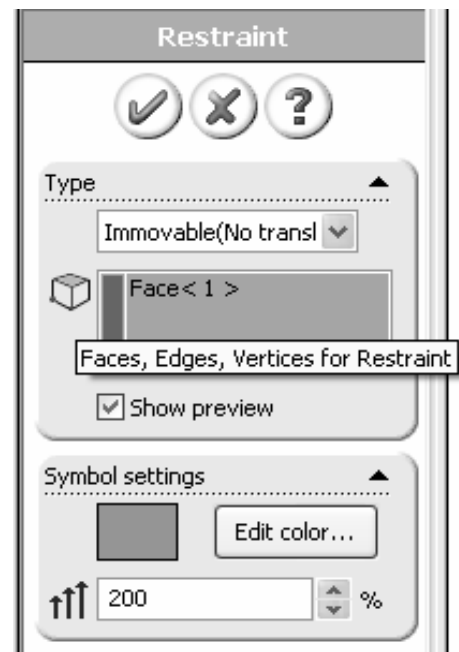


Figure 6 – **Restraint** property manager.

3. The **Faces, Edges, Vertices for Restraint** field immediately below the pull-down menu is highlighted (salmon color) 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. 7 provided the **Show preview** box is checked. Also, **Face<1>** appears in the **Faces, Edges, Vertices for Restraint** field.

If an incorrect entity (such as a vertex, edge, or surface) is selected, click the incorrect item in the **Faces, Edges, Vertices for Restraint** field and press the **[Delete]** key, then repeat step #3.

- 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. 6. 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. 7 were arbitrarily increased to 200%. Experiment with this option.

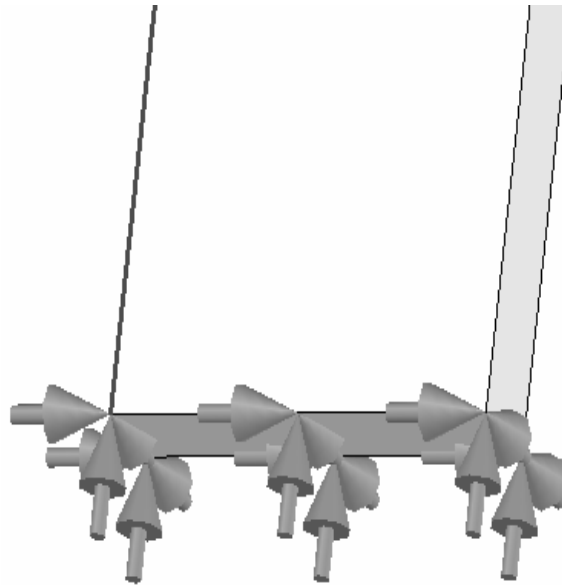


Figure 7 – **Immovable** restraints applied to bottom of the curved beam model.

- Click **[OK]** 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 tree.

Aside:

Immovable restraint symbols shown in Fig. 7 appear as simple arrows or vectors with a straight shaft. These symbols indicate *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 symbolizes 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, $F_y = 3200$ 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 points of load application and restraints, 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. Alternatively, the force could be applied to the upper or lower *edge* of the model at the extreme left side. These loading situations would require slight modification to the magnitude of force F_y to account for the additional distance from the hole to the left-side of the model.

However, the assumption of pin loading allows us to demonstrate use of a *Split Line* to isolate a portion of the hole surface where contact with a pin is assumed to occur. This surface is where pin force is transferred to the curved beam model. The actual contact area depends upon a number of factors, primary of which are 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 model). Somewhat arbitrarily, this example 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 approach is demonstrated in a future example.

Inserting Split line(s)

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 only one method, which uses a reference plane. Also, to enhance our familiarity with support facilities within the software, several references to the *Status Line*, located at the bottom left-side of the graphics screen, are made as the analysis progresses.

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 (expanded) in Fig. 8.
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 illustrated in Fig. 8. Continue to refer to Fig. 8 in steps 3 through 5 below.

3. Within the **Plane** property manager, under **Selections**, the **Reference Entities** field is highlighted (salmon color). Note text appearing in the *Status Line*, which prompts the user to “**Select valid entities to define a plane (plane, face, edge, line or point).**” In the next step, 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 elected to open the **Curved Beam** part file available at the textbook web site, the top plane passes through the origin, which is located on the bottom of the model¹ in Fig. 8.

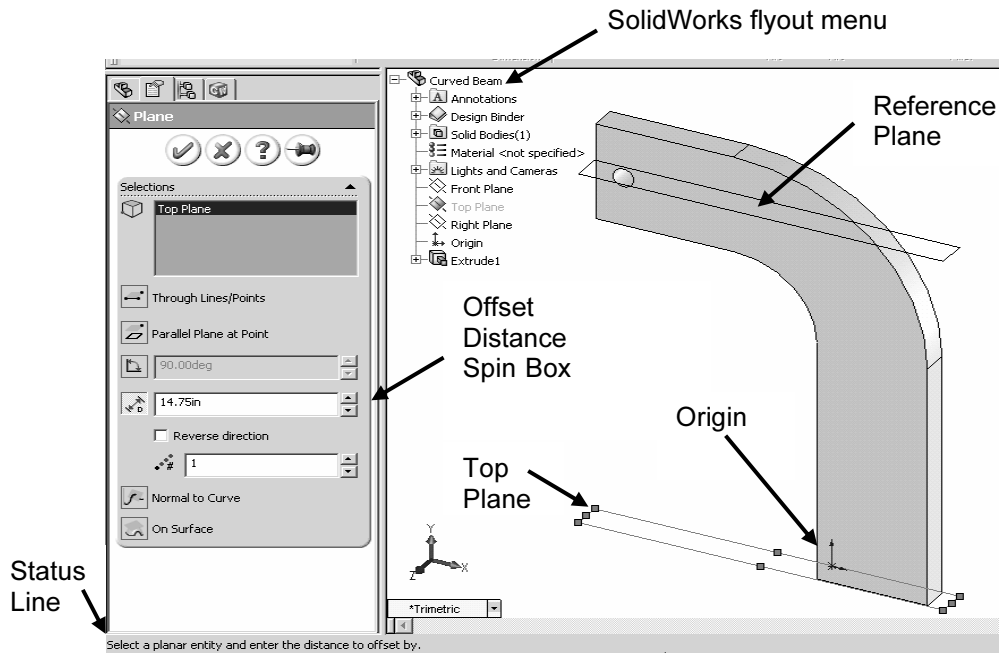


Figure 8 – The **Plane** property manager and various selections made to create a reference plane through the bottom of the hole.

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.

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

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 *distance* from the **Top Plane** (used as a reference in *your* model) to the bottom of the hole.

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

The reference plane created in the preceding steps appears highlighted and is labeled **Plane1** in the current screen image. In the following steps, **Plane1** is used to create *Split Lines* near the bottom of the hole. These *Split Lines* enable us to define the downward load distributed over small area identified on the bottom of the hole.

- 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. 9.

The *Status Line* prompt (at bottom-left of screen) reads, “**Change the type or select tool entities and target entities to split.**” This advice is followed in the subsequent steps.

- 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).
- Beneath **Selections** the **Splitting Bodies/Faces/Planes** field should appear highlighted. If **Plane1** does not appear in this field, move the cursor onto the graphics screen and click to select **Plane1** when it is highlighted. **Plane1** appears in the upper field.

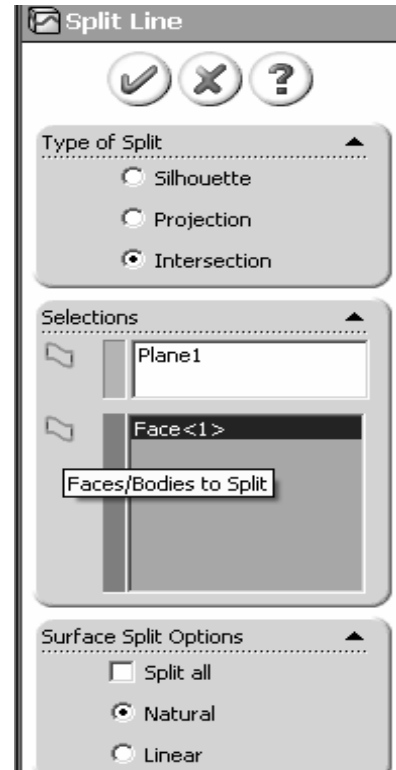


Figure 9 – **Split Line** property manager showing selections.

- Click inside the **Faces/Bodies to Split** field to activate this field (salmon color), then move the cursor over the model and select the *inside surface* of the hole. It may be necessary to zoom-in on and rotate the model to select this surface. Once selected, **Face<1>** appears in the field. Figure 10 shows a partial image of the model with *Split Lines* appearing where **Plane1** intersects the bottom of the hole.

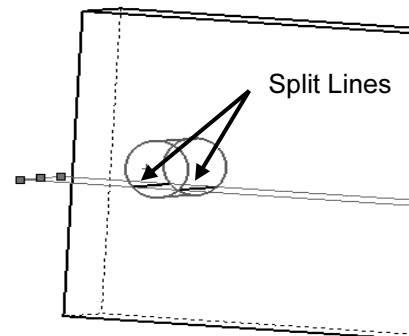


Figure 10 – Close-up view of hole showing *Split Lines* near bottom of hole.

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

11. In the **Surfaces Split Options** make sure that **Natural** is selected, then click [OK].

Applying Force to 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 the downward force $F_y = 3200$ lb on this area. Proceed as follows.

12. In the COSMOSWorks manager tree, 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. 11.
13. Under **Type**, click to select **Apply force/moment**. Again the **Faces, Edges, Vertices for Force** (salmon colored field) prompts the user to select an appropriate entity.
14. Move the cursor over the model and when the *bottom inside surface* of the hole is highlighted, click to select it. **Face<1>** appears in the active field of the **Force** property manager.
15. 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 force applied to the bottom of the hole.

Because a downward, vertical force is to be applied, select a *vertical edge . . . any vertical edge . . .* aligned with the Y-direction on the model. After selecting a vertical edge, **Edge<1>** appears in the active field and force vectors appear on the model as shown in Fig. 12.

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

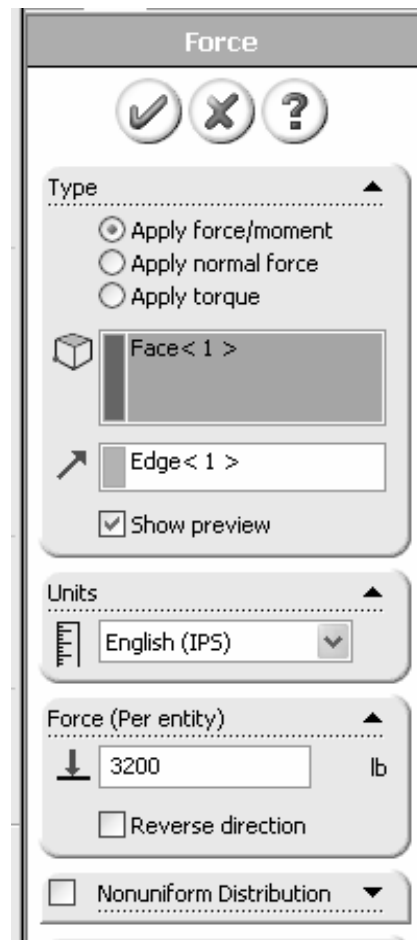


Figure 11 – Specifying a force and its direction on the hole bottom.

17. In the **Force (Per entity)** field, type **3200**. As noted in an earlier example, it may be necessary to select **Reverse Direction** if the force is not directed *downward*.
18. Click **[OK]** to accept this force definition and close the **Force** property manager. An icon named **Force-1** appears in the **Load/Restraint** folder of the manager tree

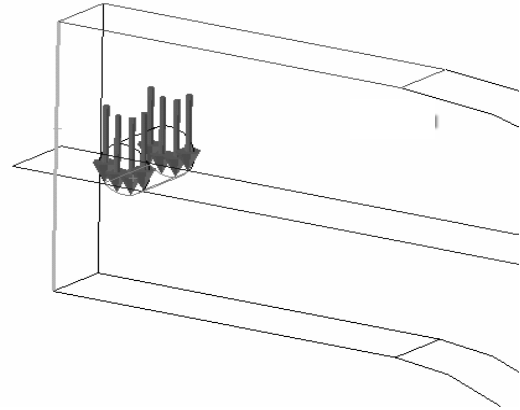


Figure 12 – Downward force applied between *Split Lines* on bottom of hole. A wireframe view of model is shown.

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 manager tree, right-click **Mesh** and from the pull-down menu select **Options...** The **Options** window opens with the **Mesh** tab pre-selected. Within this tab all or 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. *NOTE: A **Draft** quality mesh is used for this model.*
2. Under **Mesh quality**, select **Draft**. ← *Note use of a DRAFT quality mesh!*
3. Under **Mesher type**, select **Standard**.
4. **Jacobian check:** should be set to **4 points**.
5. Under **Mesh control**, select **Smooth surface**. Other entries on the **Mesh** tab can be ignored. However, do *not* close the **Options** window.
6. Next, select the **Units** tab and change **System of units:** to **English (IPS)** and **Length Units:** to **in**. Temperature and angular velocity units are not relevant to this problem and can be ignored.
7. Click **[OK]** to close the **Options** window.

8. Next, in the COSMOSWorks manager tree, right-click **Mesh** and select **Create...** The **Mesh** property manager opens as shown in Fig. 13.
9. The **Mesh** property manager shows suggested **Mesh Parameters** (i.e. mesh size and tolerance). These system-determined values usually provide sufficient definition for an initial analysis on a non-complex part. Therefore, click **[OK]** to accept the default values and close the **Mesh** window.
10. Meshing starts automatically and the **Mesh Progress** window appears briefly. After meshing is complete, COSMOSWorks displays the meshed model shown in Fig. 14. Also, a check mark “✓” appears on the **Mesh** folder to indicate meshing is complete.

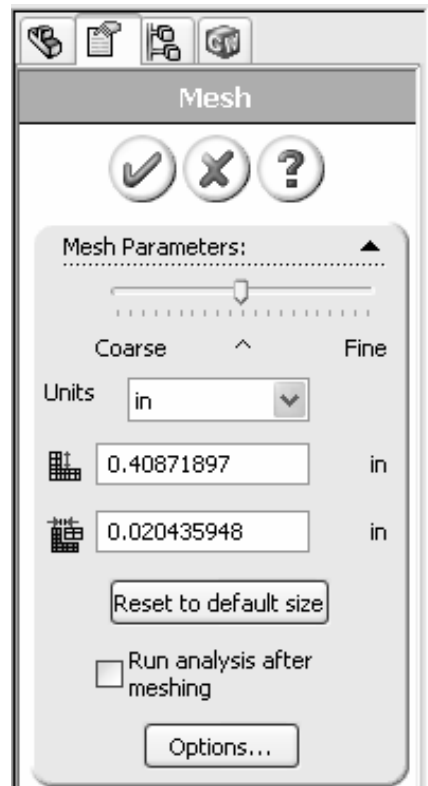


Figure 13 – Mesh Property manager showing system default **Mesh** settings applied to the current model.

OPTIONAL:

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

The **Mesh Details** window displays a variety of mesh information. The *approximate* number of nodes and elements for this model is 1828 nodes and 6347 **Draft quality** elements (numbers may vary).

Rotate the model slightly as illustrated in Fig. 14 and notice that the mesh is just two elements *thick*. Two elements across the model thickness is considered the minimum number for which *Solid Elements* can 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.

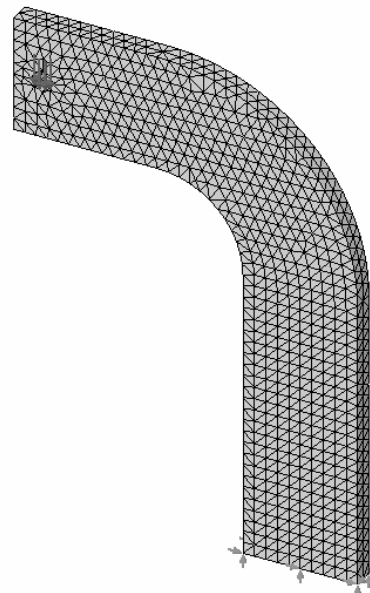


Figure 14 – Curved beam with mesh and boundary conditions illustrated.

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. In this portion of the program numerous equations defining the study are solved and default results of the analysis are prepared for review.

1. To run the analysis, right-click the **Curved Beam Analysis (-Default-)** folder highlighted in Fig. 15. From the pull-down menu, select **Run**. This action initiates the solution process.
2. When the solution is complete, click **[OK]** to close the **COSMOSWorks** window that indicates **[Static Analysis Completed]**. Note, a warning window appears only if the solution is unsuccessful.

After a successful solution, the **Stress**, **Displacement**, **Strain**, **Deformation**, and **Design Check** folders appear at the bottom of the COSMOSWorks Analysis Manager as illustrated in Fig. 15. These folders contain default plots created as a result of the static analysis. In addition to the **Stress** folder examined in the first example, the **Displacement**, and **Design Check** folders are examined below. However, feel free to explore the contents of other folders as your time and interest permit.

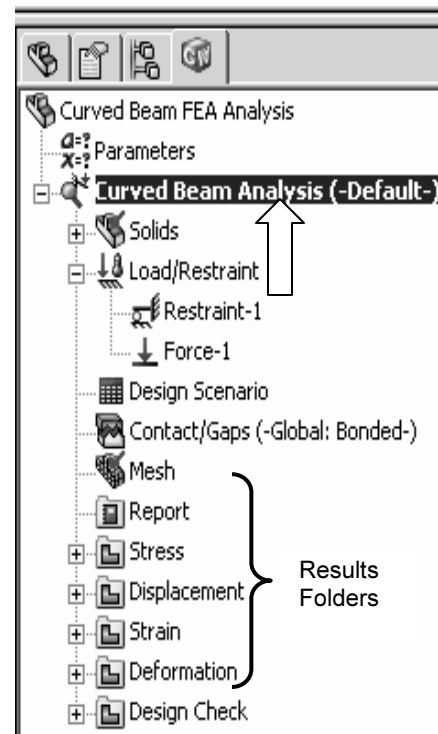


Figure 15 – Folders containing results of the current Solution.

Examination of Results

Analysis of von Mises Stresses Within the Model

Outcomes of the current analysis, in the form of graphs, plots, and data, can be viewed by accessing results stored in the various folders listed above. This is where validity of results is investigated by crosschecking 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, click the “+” sign in front of the **Stress** folder. The folder opens and reveals that it contains one plot titled **Plot1 (-vonMises-)**.
2. Double-click **Plot1 (-vonMises-)** and the von Mises stress plot is displayed. This is the default stress plot produced by COSMOSWorks.

Figure 16 reveals a *near* image of what currently appears on your screen. The following steps convert your current screen image to that shown in Fig. 16.

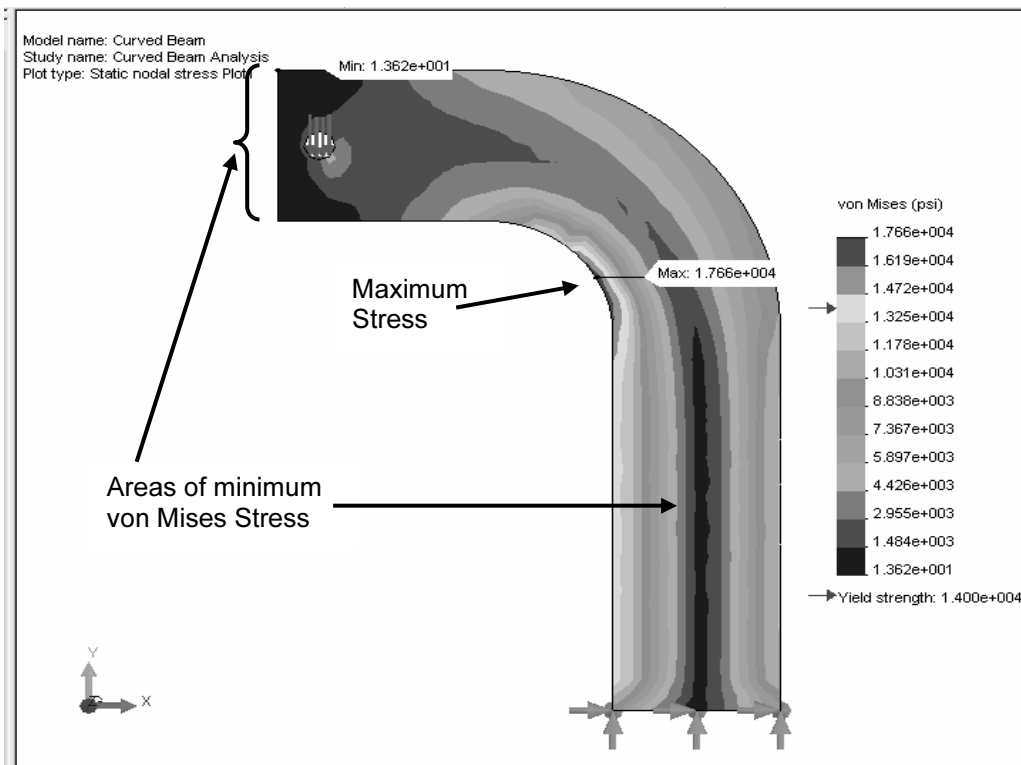


Figure 16 – Image of curved beam model showing von Mises stress *after* making changes outlined below.

NOTE: *Stress contour plots are printed in black, white, and grey tones. Therefore, light and dark color areas on your screen will differ from images include here.*

3. Right-click **Plot1 (-vonMises-)** and from the pull-down menu select **Chart Options**. The **Chart Options** property manager opens as shown in Fig. 17.
4. Within the **Display Options** section, click to place a check mark to activate **Show min annotation** and **Show max annotation**. This action causes the minimum and maximum stress locations to be labeled on the model.

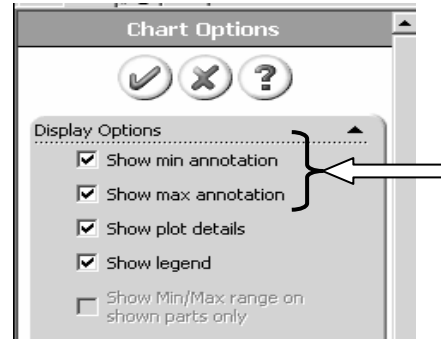


Figure 17 – Upper portion of **Chart Options** property manager showing current selections.

5. Click **[OK]** to close the **Chart Options** property manager.
6. Right-click **Plot1 (-vonMises-)** and from the pull-down menu select **Settings**. The **Settings** property manager opens as illustrated in Fig. 18.

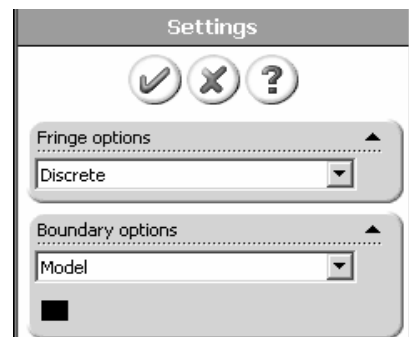


Figure 18 – Selections in the **Settings** property manager.

7. From the pull-down menu beneath **Fringe options**, select **Discrete** as the fringe type to be displayed.
8. Next, in the **Boundary options** pull-down menu, select **Model** to superimpose an outline of the model on the image.

9. Click **[OK]** to close the **Settings** property manager.
10. Right-click **Plot1 (-vonMises-)** and from the pull-down menu select **Edit Definition...** The **Stress Plot** property manager, shown in Fig. 19, opens.

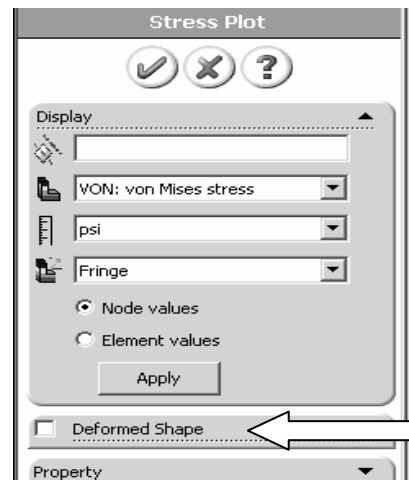


Figure 19 – **Stress Plot** property manager.

11. Only one change will be made at this time. Click to “clear” the check-box adjacent to **Deformed Shape**.
12. Click **[OK]** to close the **Stress Plot** property manager.

The following observations can be made about Fig. 16.

OBSERVATIONS:

- Areas of low stress (dark blue) occur at the top-left side of the model and also run through the center of the model. The lowest stress is approximately $1.362e+001 = 13.62$ psi. Regions of high stress are indicated by red areas. The maximum stress indicated is $1.766e+004 = 17660$ psi, which occurs along the concave surface
- Material *Yield Strength* ≈ 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. 16 (“+” 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 failure and thus are invalid? 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 follow.

To answer these questions we briefly digress to investigate the definition of von Mises stress and the 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 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.”² That quantity is known as the von Mises stress. In other words, for

² Shigley, J.E., Mischke, C.R., Budynas, R.G., Mechanical Engineering Design, 7th Ed., McGraw-Hill, 2004, p.263.

the most complex state of stress that you can imagine (e.g., a three-dimensional stress element subject to a combination of shear and normal stresses 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” to determine whether or not failure by yielding is predicted. 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. 16. 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 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 desired stress component that would yield favorable comparisons with stresses calculated using classic equations of mechanics of materials. It is left as an exercise to determine the *appropriate stress* to show the variation from compressive stress at the concave surface to tensile stress at the outer radius of the curved beam model.

Assessing Safety Factor for the Curved Beam

To view the Factor of Safety distribution in the curved beam model, proceed as follows -

1. Click the plus sign “+” next to the **Design Check** folder. The **Plot1 (-FOS-)** icon appears, where **FOS** means “Factor of Safety.”
2. Double-click **Plot1(-FOS-)** and the plot displayed in Fig. 20 appears.

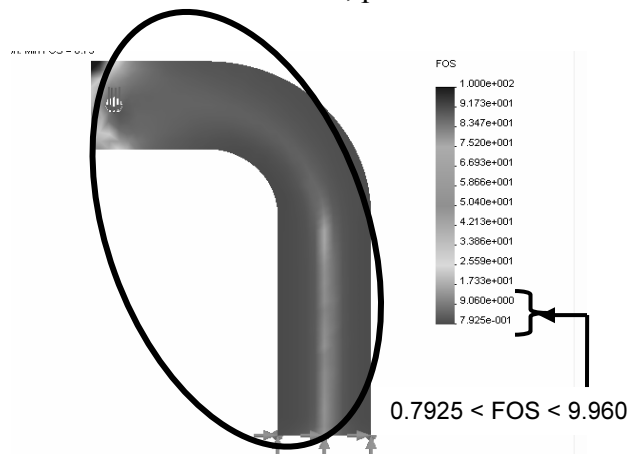


Figure 20 – Default Image produced by the **Design Check** plot.

The following observations can be made about Fig. 20.

OBSERVATIONS:

Most of the circled region on the model appears red (on your screen) indicating safety factors ranging somewhere between 7.925e-001 (0.79 rounded) to approximately 9.060e+000 (9.1 rounded). Refer to the bracketed region at the bottom of the Factor of Safety (FOS) color-scale shown to the right of the model where these values are indicated.

However, due to the lack of color *gradient* in much of that range (i.e., large areas of the model appear a similar red color), it is very difficult to differentiate between unsafe regions, where the safety factor is <1, and regions where safety factor is > 1.

To assist in determining the safety factor with greater clarity, the **Design Check** capability is explored below.

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 –

3. In the COSMOSWorks manager tree, right-click the **Design Check** folder and from the pull-down menu select **Define...** The first of three **Design Check property managers** appears as shown in Fig. 21.
4. In the **Criterion** field (second field from top), click the pull-down menu to select **Max von Mises stress**, if not already selected.

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

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

In other words, the above 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.



Figure 21 – First **Design Check** property manager. The *comparison* stress (criteria) is selected here.

Thus, the current criteria 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.

5. Proceed to the next step by clicking arrow button [→] at the bottom of this dialogue box. The **Design Check, Step 2 of 3** dialogue box appears as seen in Fig. 22.
6. Under **Set stress limit:**, click to select **to Yield strength** (if not already selected). Notice that the model yield and ultimate strengths, defined earlier when material properties were selected, appear near the middle of this dialogue box. Finally, the bottom box lists the maximum von Mises stress found during the solution. By examining the values in these two boxes it should be obvious that, somewhere in the part, the maximum von Mises stress exceeds the material yield strength.
7. Click the arrow button [→] at bottom of this dialogue box to proceed to **Step 3 of 3** in the **Design Check** property manager shown in Fig. 23.

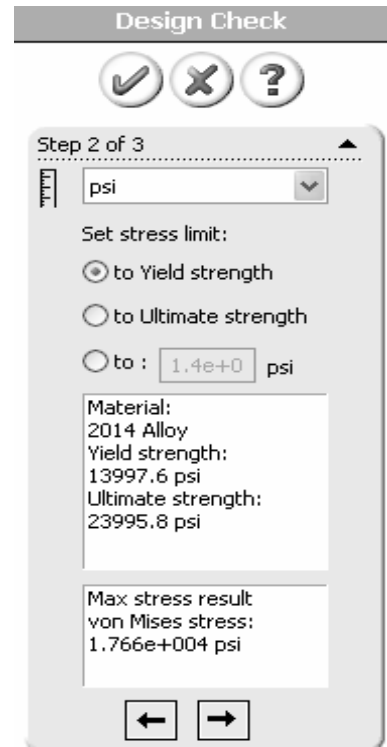


Figure 22 – Step 2 of 3 in the **Design Check** process.

8. 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 **0.792507** (approximately 0.79) 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 from those shown*).

Also note that this value of safety factor matches that listed at the bottom of the FOS color-scale on the current screen image, Fig. 20. Finally, observe that this value is the reciprocal of the equation appearing in the first **Design Check** window. That is:

$$\frac{\sigma_{Limit}}{\sigma_{vonMises}} = \frac{13997}{17660} = 0.7925$$

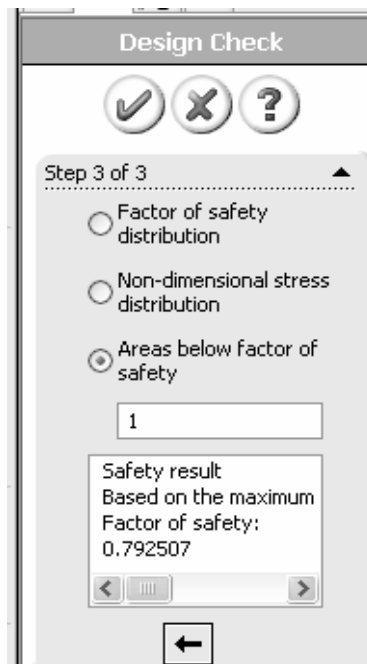


Figure 23 – Redefinition of *Safety Factor* and values to be displayed on the new plot.

- Click **[OK]** to close the **Design Check** property manager and a new plot, **Plot2 (-Criterion: Max von Mises Stress-)**, appears in the graphics window and is listed beneath the **Design Check** folder.

The figure now on the screen should correspond to Fig. 24. 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. Observe that only a localized region along the left vertical edge and extending into the concave region has a safety factor less than one.

NOTE: This solution is extremely sensitive to slight variations in calculated stress values. In part this is due to discretization errors that are part of the finite element process. Thus, a smaller region where $FOS < 1$ may occur on the concave surface only.

The line of text, circled near the top-left in Fig. 24, provides a “key” to interpret safe and unsafe regions on the model.

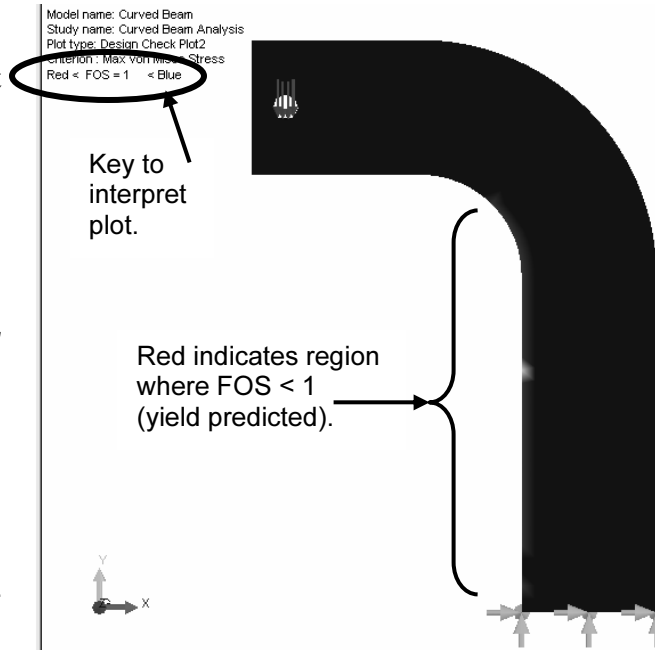


Figure 24 – Curved beam model showing areas where $FOS > 1$ (safety predicted) and where $FOS < 1$ (yield predicted).

- Repeat steps 3 through 9 above, but this time set the **Areas below factor of safety** to **2** instead of **1**, in step 8. 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 *value* to the magnitude of yield strength. This topic is aligned with theories of failure in a machine design course.

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 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 given information. One simple way to confirm that results “make sense” is to check whether or not reaction forces are consistent with external forces 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 Analysis Manager, right-click the **Displacement** folder.
2. From the pull-down menu, select **Reaction Force...** The **Reaction Force** property manager opens as shown in Fig. 25.
3. Adjacent to the **Units:** field, select **lb**, if not already selected. (Do *not* close the **Reaction Force** window.)

At the bottom of this window, reaction names are listed in the **Component** column and reaction forces corresponding to the **Entire Model** are listed in the right-hand column. These values reveal the following:

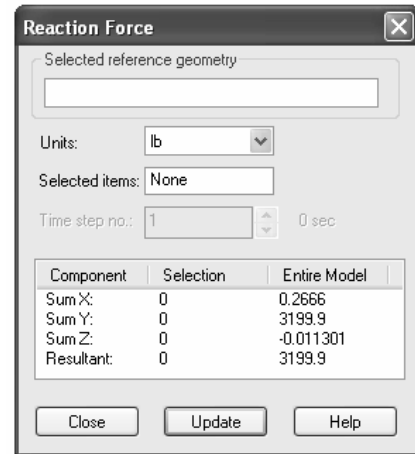


Figure 25 – Data appearing in the **Reaction Force** property manager.

SumX: 0.2666 (essentially zero)
SumY: 3199.9 (essentially 3200 = the applied force)
SumZ: -0.0113 (essentially zero)
Resultant: 3199.9 (essentially 2300)

It should be noted that a *moment* reaction at the base of the curved beam is missing from the **Reaction Force** window. This outcome is unfortunate, but is consistent with our understanding of **Immovable** restraints applied to three-dimensional, solid, tetrahedral elements. That restraint only restricts translations in the X, Y, Z directions at each restrained node. Thus, only three force reactions in these directions result in Fig. 25.

The default results above are valid for the **Entire Model**. However, in many instances a model is supported (i.e., restrained) at more than one location. A good example would be a simply supported beam that is supported at two locations. Although not the case in this example, it is often necessary to determine reaction forces at other locations on a model. Since only the bottom of the curved beam is restrained, it will serve to demonstrate how to determine reactions at a specific location on a model. Proceed as follows.

4. Rotate the model so that its bottom (restrained) surface is visible.

Analysis of Machine Elements using COSMOSWorks Professional

- Next, click to select the bottom *surface*. If an edge or vertex (corner) of the model is incorrectly selected, click again to select the bottom *surface* before proceeding.
- Within the **Reaction Force** window, click to select the **[Update]** button. The updated **Reaction Force** window appears in Fig. 26.

Notice that reaction forces now also appear in the **Selection** column and that **1 Faces** appears in the **Selected items:** field.

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.

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 the model.

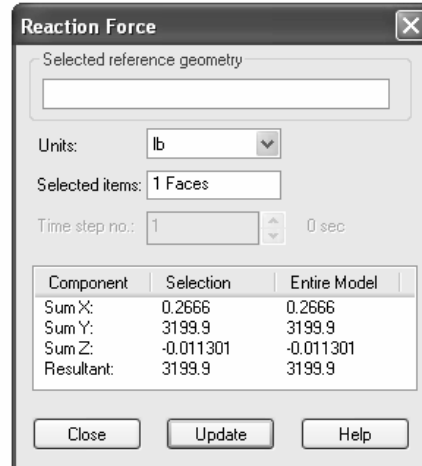


Figure 26 – **Reaction Force** window listing reactions on a selected surface of the model.

Although a surface was selected in the above example, it should be evident that other geometric features, such as edges, vertices, or additional surfaces can be selected at other locations that might be restrained on a particular model.

Logging Out and Saving the Current Analysis

This concludes an introduction to analysis of the curved beam model. It is suggested that this file be saved as follows.

- On the Main menu, click **File**. Then from the pull-down menu select, **Close**.
- The **SolidWorks** window, Fig. 27, opens and asks, “**Save changes to Curved Beam Analysis?**” Select the **[Yes]** button. This action saves all results files along with other items specified during this solution.

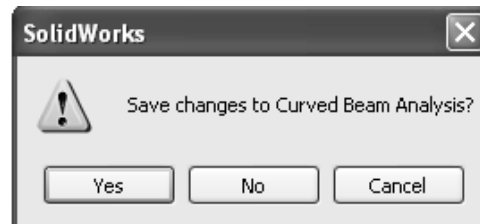


Figure 27 – **SolidWorks** window requests user confirmation whether or not to save changes to the current analysis.

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 federal test criteria is that the movable (lower) jaw be extended a certain percentage of the distance of the fully open state to ensure that column failure is an integral part of the test. Presuming that the movable jaw of the clamp satisfies 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
- Finite Element Solver: **FFEPlus**
- Restraint: **Immovable** applied to top gripping surface.
- Force: 850 N applied normal to lower gripping surface.

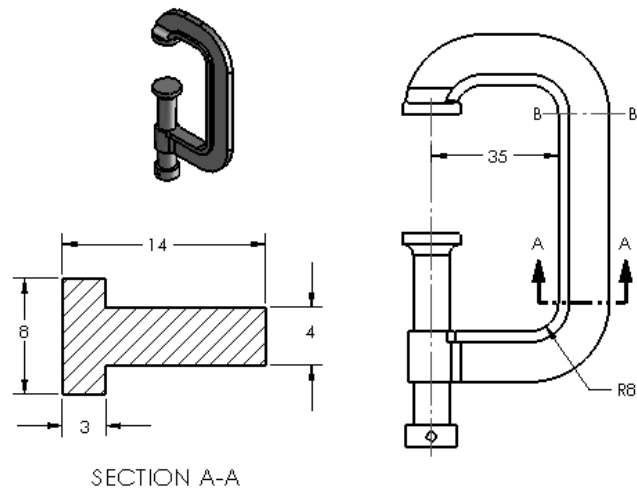


Figure E2-1 – C-clamp frame and cross-section dimensions. Stress to be determined at Section A-A.

Determine the following:

- a. Develop a finite element model, which includes: material specification, restraints, applied force, mesh the model, and a solution.
- b. 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.
- c. 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.

Analysis of Machine Elements using COSMOSWorks Professional

- d. Use the **Probe** feature to produce a graph of the *most appropriate stress* across section A-A; begin at the inside surface and continuing to the outside of the “T” cross-section. In other words, because values from this plot are to be compared with manual calculations of part (b), it is necessary to choose the corresponding stress from those available within the finite element software.
 - e. If Yield Strength of the material is exceeded, produce a plot showing regions of the clamp where safety factor < 1.0 . If Yield Strength is not exceeded, produce a plot showing where safety factor > 1.0 .
 - f. Question: Is there justification for using **High** quality elements for analysis of the C-clamp? Provide reasons either “for” or “against” using **High** quality elements for this model.
2. A simplified view of a common metal-cutting “hacksaw” is shown in the figure below. Further simplifications are made to the solid model available as file: **Hacksaw 2-2**. Those simplifications include two round holes that pass through the “backbone” of the saw on its left and right ends. For purposes of analysis the inside surface of the left-hand hole is to be considered **Immovable**. Use split lines to create a small “patch” 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
 - Units: **English (IPS)**
 - Finite Element Solver: **FFEPlus**
 - Restraint: **Immovable** applied to inside of left hole.
 - Force: 50 lb applied to inner surface of right-hand hole (split lines needed).

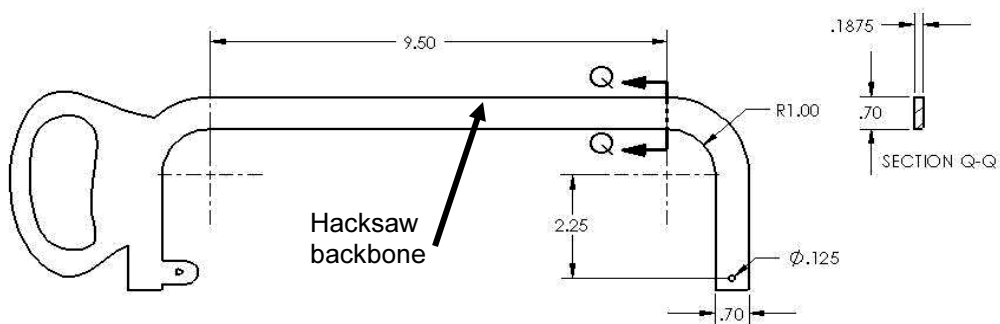


Figure E2-2 – Basic geometry of a hacksaw frame. Stress to be determined at Section Q-Q.

Determine the following:

- a. Develop a finite element model, which includes: material specification, restraints, applied force, mesh the model, and a solution.
- b. Use classical equations to compute stress at the inside and outside surfaces of the saw backbone (frame) at section Q-Q. Section Q-Q is located where the straight and curved sections are tangent.
- c. 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*.
- d. Create a stress contour plot of von Mises stress in the saw backbone. Include automatic labeling of maximum von Mises stress on the plot.
- e. 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 of the backbone cross-section. Because values from this plot are to be compared with manual calculations of part (b) above, it is necessary to choose the corresponding stress from those available within **Stress Plot** property manager.
- f. If Yield Strength of the material is exceeded, produce a plot showing regions of the hacksaw backbone where safety factor < 1.0 .
- g. Question: Is there justification for using **High** quality elements for analysis of the hacksaw backbone? Provide reasons either “for” or “against” using **High** quality elements for this model.

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 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.