

# 4 Mesh Control in SolidWorks Simulation

---

## 4.1 Introduction

You must plan ahead when building a solid model so that it can be used for realistic finite element load and/or restraint analysis cases. You often do that in the solid modeling phase by using lines or arcs to partition lines, curves, or surfaces. This is called using a “split line” in SolidWorks and SW Simulation. There is also a “split part” feature that is similar except that it cuts a part into multiple sub-parts. You sometimes need to do that for symmetry or anti-symmetry finite element analysis so that we can analyze the part more efficiently. Here, the concept of splitting surfaces for mesh control will be illustrated via stress analysis.

## 4.2 Example Initial Analysis

The split line concepts for mesh control will be illustrated via the first tutorial “Static Analysis of a Part”, using the SW Simulation example file Tutor1.sldprt which is shown in Figure 4-1. (Remember to save it with a new name by putting your initials at the front.) Consider that tutorial to be a preliminary analysis. You should recall that there was bending of the base plate near the loaded vertical post. However, the original solid mesh had only one element through the thickness in that region and would therefore underestimate the bending stresses there. You need to control the mesh there to form 4 to 5 layers to accurately capture the change in bending stress through the thickness.

The original deformed shape, in Figure 4-2, is shown relative to the undeformed part (in gray). You see bending at the end of the base and deflection of the part bottom back edge in the direction of the (unseen) supporting object below it. From the original effective stress plot in Figure 4-3 and Figure 4-4 you can see that large regions, within the red contours, have exceeded the material yield stress. Actually, the maximum value is over *133,000 psi*, or about 1.5 times the

yield stress, on the top and bottom of the base. Clearly, this part must have its material or dimensions changed and/or new support options must be utilized.

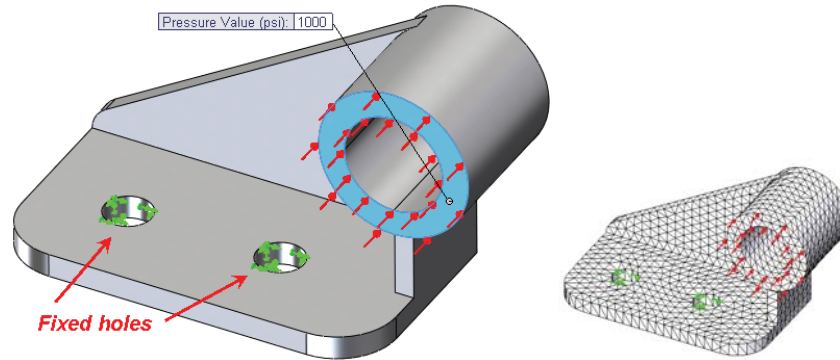


Figure 4-1 Original part, restraints, load, and mesh

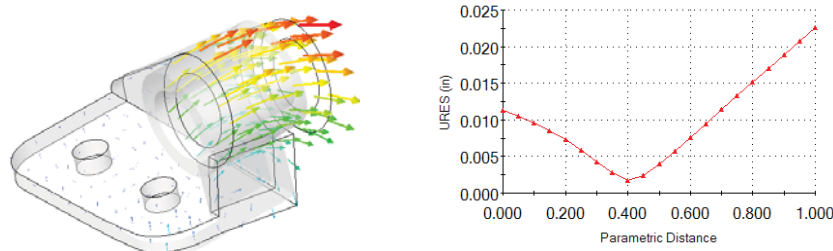


Figure 4-2 Original part vector deflections and bottom right edge graph

Begin a revision of the first study of this part by reconsidering the restraints utilized. In many problems the restraints are unclear or questionable and you need to consider other restraint cases. You previously restrained all the three translations on the two small cylindrical surfaces. That makes those surfaces perfectly rigid. That would be almost impossible to build. It is likely that the holes were intended to be bolt holes. Then the applied backward ( $-z$ ) pressure load would probably require the development of tension reaction forces along the back ( $-z$ ) half of the bolt cylinder. That would not happen because an air gap would open up. Also, a bolt usually applies a restraint to the surface under the bolt head (in addition to a bolt bearing load on its cylindrical part), and that is not in the original choice of restraints. Furthermore, the bracket seems to be attached to some rigid object under it and therefore you would expect the back edge of the bracket to be somehow supported by that object when that region of

the part deforms as seen in Figure 4-2. The original effective stresses are in Figure 4-3 and Figure 4-4.

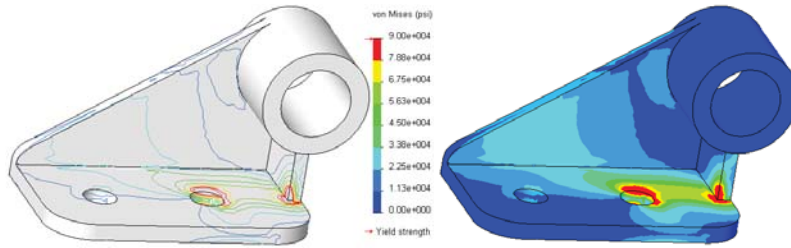


Figure 4-3 Original top surface effective stresses

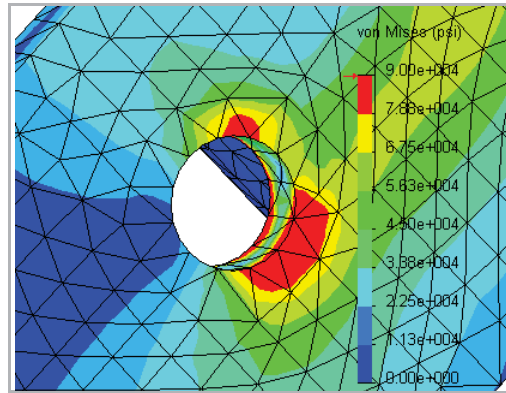


Figure 4-4 Original bottom surface effective stresses

Having reviewed an initial set of assumptions for this part, its mesh and its results a second preliminary study will be outlined. As a new possible restraint set, assume that the bolt heads are tight and act on a small surface ring around each hold. That could provide rigid body translation restraints in three directions. Each bolt would prevent three translations and the pair of them combines to also prevent three rigid body rotations. Thus they combine to prevent all six possible rigid body motions (RBM). If the bolts were not tight then only a normal displacement (along the bolt axis) would be restrained on the base top surface (and two RBM would remain).

Bearing loads on the bolt shaft can be found by an iterative process in SW Simulation, but for early studies you can assume a small cylindrical contact at the most positive  $z$  (front) location. The previously computed bending of the

part is also assumed to cause contact with the supporting object below, and thus a y- translational restraint, along the bottom back edge of the part. To accomplish those types of restraint controls you need to “split” the surfaces and transition the mesh in those regions to get better results. Thus, you need to form two smaller surface rings for the bolt heads, and split the cylinders into smaller bearing areas. That is done by adding split lines to selected regions of the part.

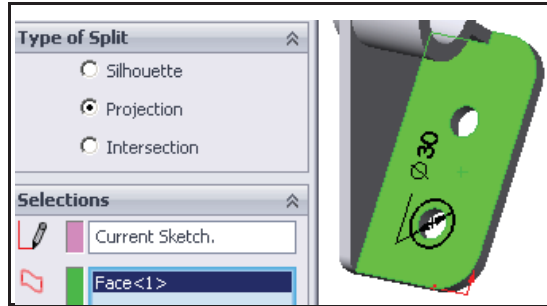
### 4.3 Splitting a Surface or Curve

To avoid changing the master tutorial file, open the part and then “save as” a new file name on the desktop. To introduce the required split lines:

1. Select the top surface of the base by moving the cursor over it until its boundary turns green and **Insert Sketch**. Pick the **Sketch** icon and pick the **Circle** option to form a bolt head washer area.
3. Place the cursor on the hole edge to “wake up” the center point. Draw a larger circle on the surface. Set its diameter to 30 mm and click **OK**. That does not change the surface; it just adds a circle to it.



4. To split the surface, go to the top and select **Insert → Curve → Split Line**. The **Type** will be a projection.
5. Next, pick the surface(s) to be cut by this curve. Here it will be the top of the base plate again so select it and click **OK**. Repeat the above two processes for the second hole.



Now you will see that moving the cursor around shows a new circle and a new ring of surface area that could be used to enforce restraints or loads. The new surface areas are shown in Figure 4-5. Later you will use these newly created ring areas as a bolt head (washer) restraint region.

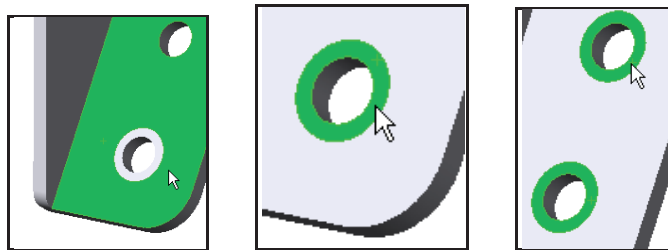
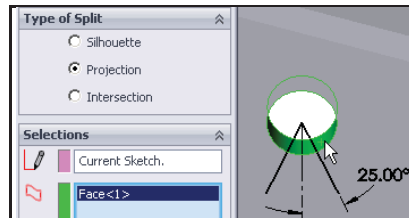


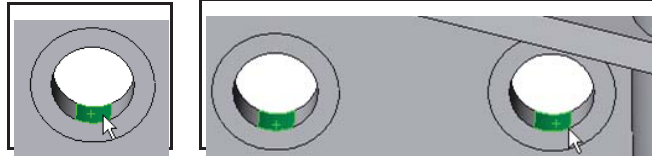
Figure 4-5 The original surface is now three surfaces

To form a vertical bearing region on the bolt hole sidewall:

1. Select the top surface and use **Sketch → Line**. “Wake up” the center point again. Use it to draw a construction line forward from the center, and two other radial lines offset by about 25°.



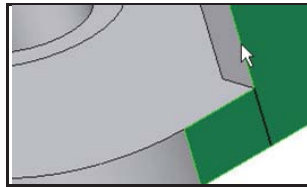
2. Select **Insert → Curve → Split Line** and pick the cylinder of the first bolt hole, click **OK**. That creates a new load bearing surface that could be used to restrain and/or control the mesh.



3. Repeat those operations for the second bolt hole.

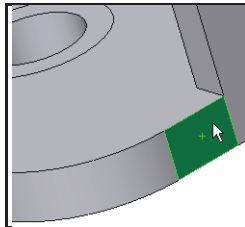
The above surface splits were constructed to give more flexibility in applying various displacement restraints. You will need another surface split to help exercise engineering control over the revised mesh. You want the “L” shaped side area to have the leg split off so you can control a bending mesh there:

1. Right click on the (+x) right side face, **Insert Sketch**→**Sketch**→**Line**.
2. “Wake up” the left vertical line to put in a short line that crosses the base.



3. Use **Insert** →**Curve**→ **Split Line** and select the “L” face, click **OK**.  
Now it has become two rectangles.

All of these new surface areas and lines can have different local mesh sizes prescribed to control our mesh generation. That is a standard feature of SW Simulation, but you must supply the engineering judgment as to where any part needs the extra line or surface divisions for applying loads (or heat sources) and restraints in an analysis. True point loads or point moments are unusual in practice. They should be replaced by reasonable loading areas (that require split curves) and pressure distributions.



## 4.4 Beginning SW Simulation Study

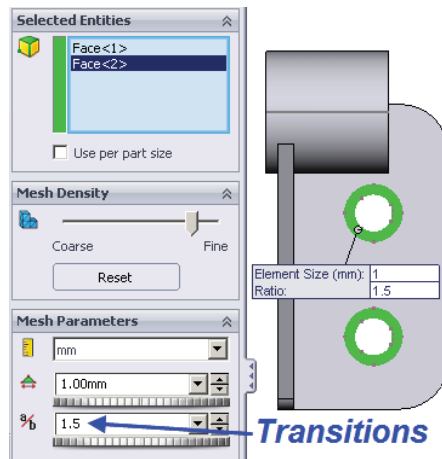
Activate the SW Simulation Manager by clicking on its icon:

1. Then start a static study. In this case the element type defaults to a quadratic solid. **Part name**→**Apply/Edit Material**. Pick alloy steel as the material.
2. Note that you have a choice of display units for the properties. The number of significant figures in the test data should be independent of the units selected. Any difference is due to using a higher number of digits in the units conversion. Do not be misled by a long string of digits in material values. You can usually trust the first three, or maybe four. Note for later use that the yield stress of this material is about *620 MPa* or *90 ksi*.

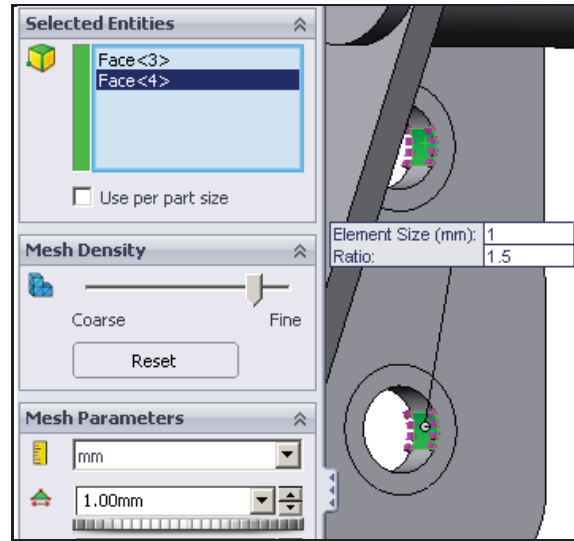
## 4.5 Mesh Control

Wherever translational displacement restraints are applied, reaction forces are developed and localized stress concentrations are likely. Therefore, you want to assure that small elements are created in such regions. This process is referred to as mesh control. It is required in almost every study. Invoke it with:

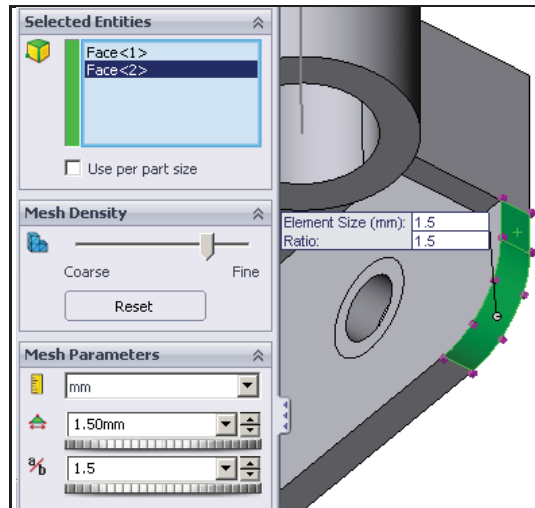
1. Right click on **Mesh**→**Apply Control** to bring up the **Mesh Control panel**. The default element size of about *3 mm* needs to be changed in the new regions created above. Select the region around the two bolt washers by picking those two surfaces, and set the desired size to 1mm.



- Likewise, for the two bearing surfaces within the smaller vertical cylindrical holes set the size to about 1.0 mm.



- In the final corner region you need several elements through the thickness of curved front corner to accurately model the local bending seen in the first study. Select the small split rectangle and the adjacent cylindrical corner and use an element size of about 1.5 mm.



## 4.6 Mesh Preview

Use **Mesh**→**Create** to generate the mesh. Then examine the mesh and increase or decrease the local sizes specified above so that it looks acceptable, as shown in Figure 4-6. Note that the mesh makes a smooth transition from the smaller element sizes to the larger default size in the far body regions. An additional refinement near the corner of the base and rectangular shaped leg (below the loaded tube) would also be wise. You should always preview the mesh before running the solution.

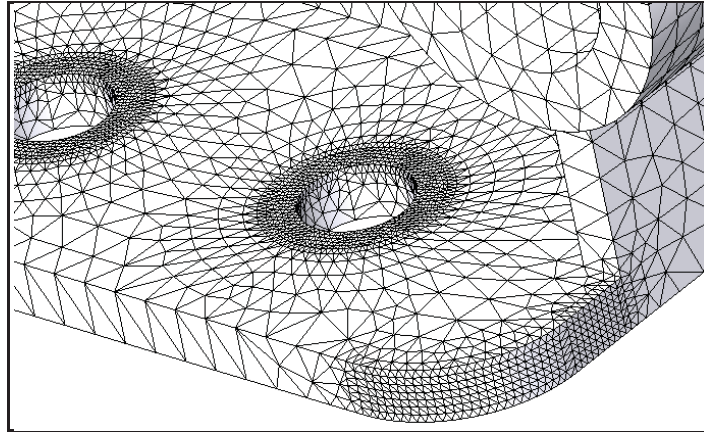


Figure 4-6 Controlled mesh sizes for the second model

## 4.7 Fixtures (Essential Boundary Conditions)

Various terms are used to describe the essential boundary conditions: fixtures, restraints, supports, etc. In SW Simulation the term is Fixtures. The part fixtures will be enforced by beginning with the new surface areas representing the bolt washer contact regions:

1. Select **Fixtures** → **Roller/Slider** and restrain the washer areas against y-translation perpendicular to the base (solid elements do not have rotational nodal DOF). This will also prevent rigid body motion (RBM) rotations about the x- and z-axes.
2. In the **Standard panel** pick the two concentric ring faces as the **Selected Entities** and chose **Roller/Slider** as the **Type**.