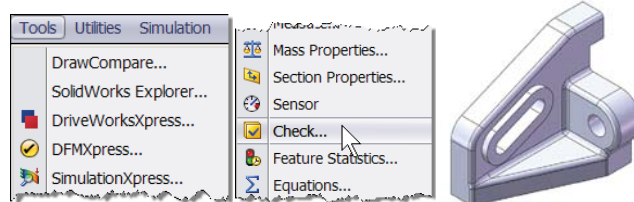


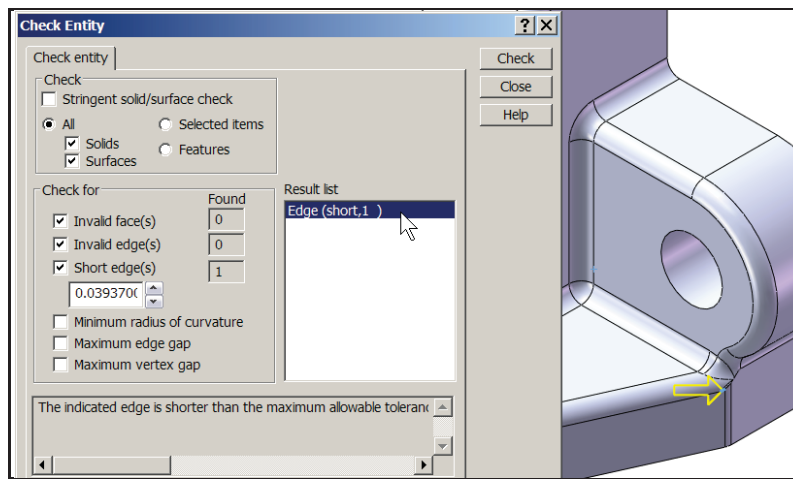
1.6 Part Geometric Analysis and Meshing Failures

Before attempting meshing your part, for a finite element analysis, you should check your solid model for potentially fatal geometric flaws that may not be noticed except at greatly magnified views. Within SolidWorks this is called a **Geometric Analysis**. To utilise that feature, a geometric analysis check the Angle_Connector part will be outlined:

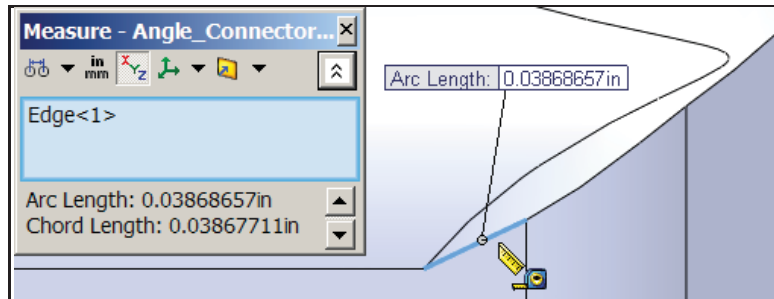
1. Select **Tools**→ **Check** will open the **Check Entity** panel.



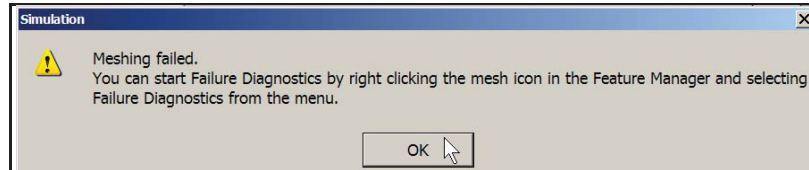
2. In that panel check the boxes for most entities, select **Check**.
3. Highlight each item in the **Result List**. As you scroll down the Result list the short edge location on the part is illustrated by a yellow arrow. Either the feature needs to be eliminated, or the mesh will need to be finer.



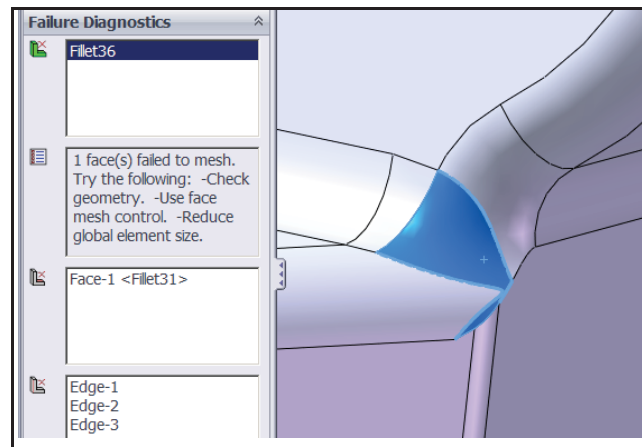
4. To consider a potential mesh refinement you should determine the size of the small feature. Use **Tools**→**Measure** to open up the **Measure** panel. Select the **XYZ** option, click on a edge of the feature to see its length.



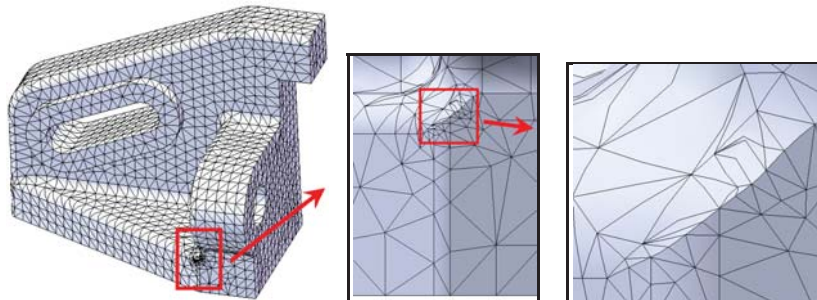
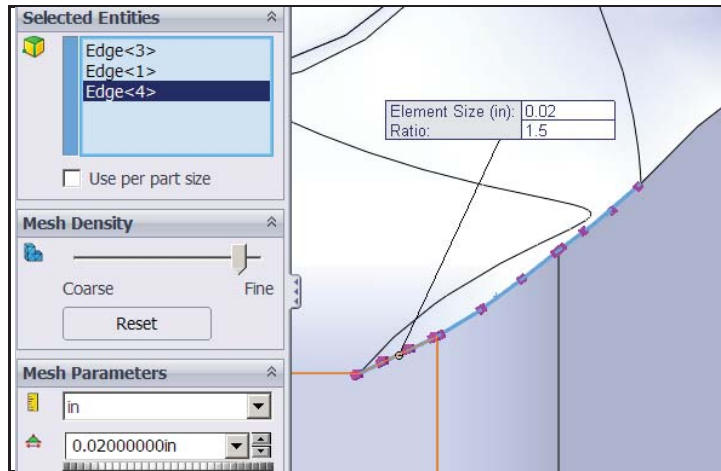
5. Attempt to create a mesh: **Mesh→Create Mesh**. As expected, that process fails and a failure diagnostic message appears:



6. Right click on **Mesh** to open the **Failure Diagnostics** panel. Scroll down the lists of faces or edges that caused the meshing failure. In this case, there is a highly distorted surface that formed with the fillets. Sometimes this type of surface can be removed by suppressing the fillets, or by simply building the fillets in a different order. Sometimes the surface can split by inserting split lines to make more manageable regions. Repairing the surface is better than having to *try* to control the mesh.



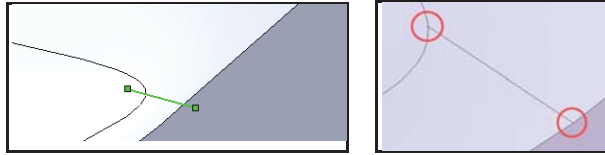
7. First, try to get some type of mesh output by specifying a small element size along the edges of the distorted region **Mesh→Apply Mesh Control**. Specify a local element size that will assure that one or two elements will fit along the smallest edge. Surprisingly, this worked. But it yielded a distorted mesh in the region of the small edge. Ideally, the surface triangles (one face of the tetrahedron) would be isosceles. That gives an element “aspect ratio” (say the ratio of the long side divided by the short one) of unity. Here the triangles are curved. A few are also badly distorted and not desirable for analysis if they are in an expected high stress region.



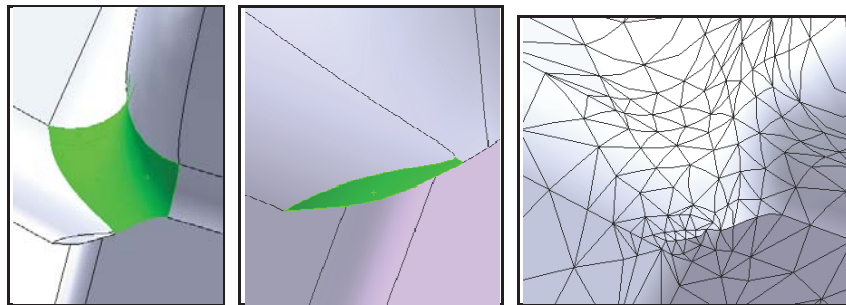
One measure of the quality of an element is its aspect ratio. Think of that as the ratio of the diameter of the enclosing sphere to the diameter of the enclosed sphere. Alternately, use the ratio of the longest element edge length to its

shortest. An ideal aspect ratio should be near unity. Check the mesh quality by looking at a plot of the aspect ratio of the elements. Select **Mesh → Create Mesh Plot → Aspect Ratio**.

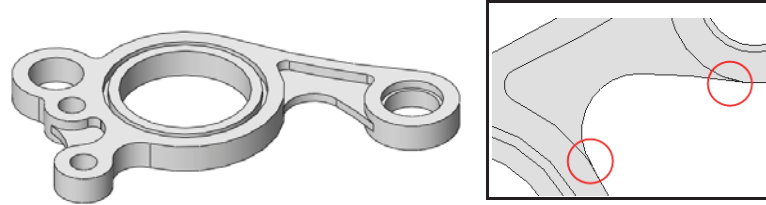
Try to improve this mesh by removing the bad surface, or subdividing it into two regions. At the narrow region, insert a **split line** that avoids very small intersection angles with both curves.



The small slender partition will need very small elements, but the larger partition can have larger ones. Especially if you use the transition control ratio to give five or more growth layers at an enlargement ratio of about 1.2 instead of the default value of 1.5. Use **Mesh → Apply Mesh Control** to specify element sides of *0.02* and *0.05 inches*, respectively in the **Mesh Control panel**. They give a much better mesh in this region.



Another part, the *Five_Hole_Link*, shows a tangency that gives very bad element aspect ratios. A common cause of failure in mesh generation is to have two solid regions or two joining surfaces meet at a near zero angle. That often happens in practice and requires intervention to be able to create a mesh for analysis. If a tangency condition is really required in the part, then you must force smaller element sizes there via the **Mesh Control** option.



If the part can be modified to avoid the tangency, then meshing becomes much easier. That is illustrated below where it was feasible to avoid a tangency requirement in this application.

