Before attempting a finite element analysis you should check your solid model for fatal flaws that may not be noticed except at greatly magnified views. Within SolidWorks this is called a Geometric Analysis. To Utilise that feature:

1. **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 school down the list the result item location on the part illustrated by an arrow. Either the feature needs to be eliminated (best), or the mesh needs to be fine there (ok). To consider a potential mesh refinement you should determine the size of the small feature.

4. **Tools ➔ Measure** will open up the Measure panel. Select the **XYZ** option, click on a edge of the feature to see its length displayed.

5. Attempt to create a mesh: **Mesh ➔ Create Mesh** and select a default element size. 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 mesh 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 building them in a different order. Sometimes the surface can split by inserting split lines to make more manageable regions.
7. First try specifying small element size along the edges of the distorted region Mesh→Apply Mesh Control.
8. Specify a local element size that will assure that one or two elements will fit along the smallest edge.

Surprisingly, this executed but gave a distorted mesh. Ideally, the surface triangles 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. They are also badly distorted and not desirable for analysis.

9. Try to improve this mesh by removing the bad surface, or subdividing it into controllable regions. At the narrow region insert a split line that avoids very small intersection angles with both curves.
10. The small slender partition will need very small elements, but the larger partition can have larger ones. Use Mesh→Apply Mesh Control to specify element sides of 0.02 and 0.05, respectively in the Mesh Control panel.

This gives a much better mesh in this region.
Another part shows a similar mesh distortion that gives a very bad element aspect ratio and might have caused the mesh generation to fail. It is wise to carry out a geometry analysis at various stages of your solid construction.

You also want the corner angles of the element to be equal. While that is not likely to occur you must avoid corner angles less than 15 degrees or the element fails due to too much distortion (it develops a non-positive geometric Jacobian).
A common cause of failure in mesh generation is to have two solid regions or two joining surfaces meet at a zero angle. That often happens in practice and often requires intervention to be able to create a mesh for analysis.