

# Plate With a Hole - Verification & Validation

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

## Verification & Validation

Now that we have our results, it is important that we check to see that our computational simulation is accurate. One possible way of accomplishing this task is comparing to the pre-calculations, as we did in the results section. Another way to check our results is by refining the mesh further. The smaller the elements in the mesh, the more accurate our simulation will be, but the simulation will take longer. To refine the mesh, look to the outline tree and click **Mesh > Face Sizing** Change the element sizing to 0.025 in (half the size of the mesh we originally tried). The new mesh looks like [this](#) . It has twice as many elements as the original.

Now hit solve. Compare the values for your stresses with those we found for the original mesh. Are they very different? Or do they seem to approach a limit? If the latter, the mesh is refined enough and if you modeled the problem correctly, you are done! Below are the values from our original mesh, followed by the values for our refined mesh.

	Maximum Sigma_xx	Maximum Deformation
<b>Theory Values</b>	3.0033 x 10 <sup>6</sup> psi	0.1724
<b>Original Mesh</b>	3.0336 x 10 <sup>6</sup> psi	0.1761
<b>Refined Mesh</b>	3.0360 x 10 <sup>6</sup> psi	0.1759

As one can see from the table above the results do not change greatly as the mesh is refined. This means we don't need to refine the mesh further.

We're Done!

[Go to Exercises](#)

[Go to all ANSYS Learning Modules](#)