

AIM Plate with Hole - Verification

Author(s): Sebastian Vecchi, ANSYS Inc.

[Problem Specification](#)

- [1. Pre-Analysis & Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Physics Setup](#)
- [5. Results](#)
- [6. Verification & Validation](#)

Verification

In the pre-analysis, the maximum stress was calculated. In order to verify that our simulation was accurate, a comparison must be made. To calculate the maximum stress of the simulation, [return to the Results panel](#), click on [Add](#) next to [Results](#), and select [Calculated Value](#). In the [Function](#) drop down menu, select [Maximum](#). In the [Variable](#) drop down menu, select [Stress XX](#) in the [Stress](#) category. Next, press the [Evaluate](#) button. Very quickly, the maximum stress of the plate with a hole is calculated and displayed.

The table below compares the calculated and simulated values for maximum stress in the plate with a hole. With a difference of less than 5%, we can consider our simulation to be accurate. This small difference in results may be caused by not using enough mesh refinement.

Simulated Value	Calculated Value	Difference
3.089E6 psi	3.033E6 psi	1.81%

[Go to all ANSYS AIM Learning Modules](#)