Cantilever Beam - Verification & Validation (OLD)

Author: John Singleton, Cornell University

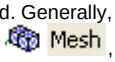
Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

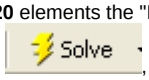
Verification & Validation

Verification of Maximum Bending Stress and Maximum Total Deformation

We have already noted that the ANSYS results compare well to our hand calculations obtained using Euler-Bernoulli beam theory in the pre-analysis. The ANSYS simulation gave 4.6352MPa for the maximum bending stress and the calculation in the pre-analysis yielded 4.635 MPa. The ANSYS simulation gave 0.005135m for the total deformation of the beam at x=4 while the calculation from the pre-analysis yielded 0.005103m. The ANSYS results closely match the hand calculations from the pre-analysis. This is one way to verify the solution.

Mesh Refinement

Another way to verify the solution of a numerical method is to examine the convergence of the solution as the mesh is refined. Generally, the numerical solution should converge to the exact solution as the mesh is refined. In order to refine the mesh, first click on the ***Mesh*** tab, 

, in the tree outline. Next, expand ***Sizing*** in *Details of Mesh*. The mesh will be refined by adjusting the ***Element Size***. The length of the beam is 4 m so if you want **n** elements then you will need to set the "Element Size" to (4m/**n**). For instance, if you wanted **20** elements the "Element Size" should be set to (4m/**20**)=0.2m . After you have changed the ***Element Size*** to your preference, click on the ***Solve*** button, 

, to recalculate the solution with the new mesh.

The table below displays the outputs of the ANSYS simulation for a mesh of 2 elements and a mesh of 10 elements.

	Total Deformation (m)	Maximum Bending Stress Pa
Theory Values	0.005103	4.635x10^6
2 Element FEA	0.0051352	4.6352x10^6
10 Element FEA	0.0051352	4.6352x10^6

As one can see from the table above the results do not change as the mesh is refined. The reason that the results do not change is as follows: the exact solution for cantilever beam deformation is cubic and for this setup ANSYS uses element BEAM 188 which also uses cubic interpolation. Thus, for the simple cantilever beam setup the numerical method converges very quickly.

Go to Exercises

Go to all ANSYS Learning Modules