

Thermal Stresses in a Bar - Numerical Results

Author: Ben 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](#)

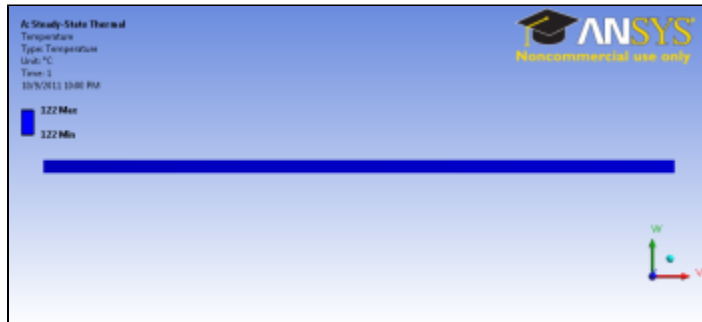
[Comments](#)

Numerical Results

Now, we will look at the results of the simulation.

Temperature

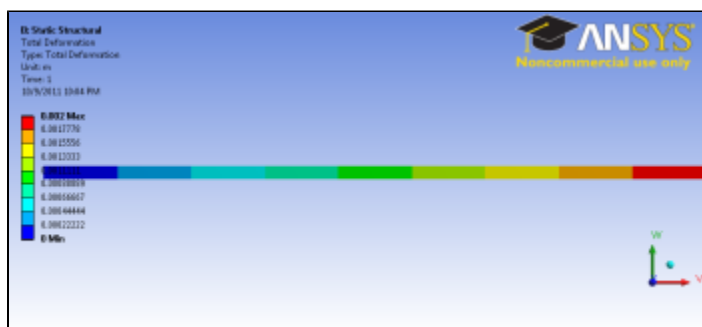
In the *Outline* window, click the temperature solution we created  **Temperature** . you should see the following



The temperature is 122 degrees Celsius throughout, which is to be expected.

Total Deformation

In the *Outline* window, click the deformation solution we created  **Total Deformation** . you should see the following



The maximum deformation is .002 m, which matches our boundary condition.

Normal Stress

In the *Outline* window, expand the beam tool folder, and click on **Direct Stress**. You should see the following



The stress is constant throughout.

[Go to Step 7: Verification & Validation](#)

[Go to all ANSYS Learning Modules](#)