

Thermal Stresses in a Bar - Numerical Solution

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 Solution

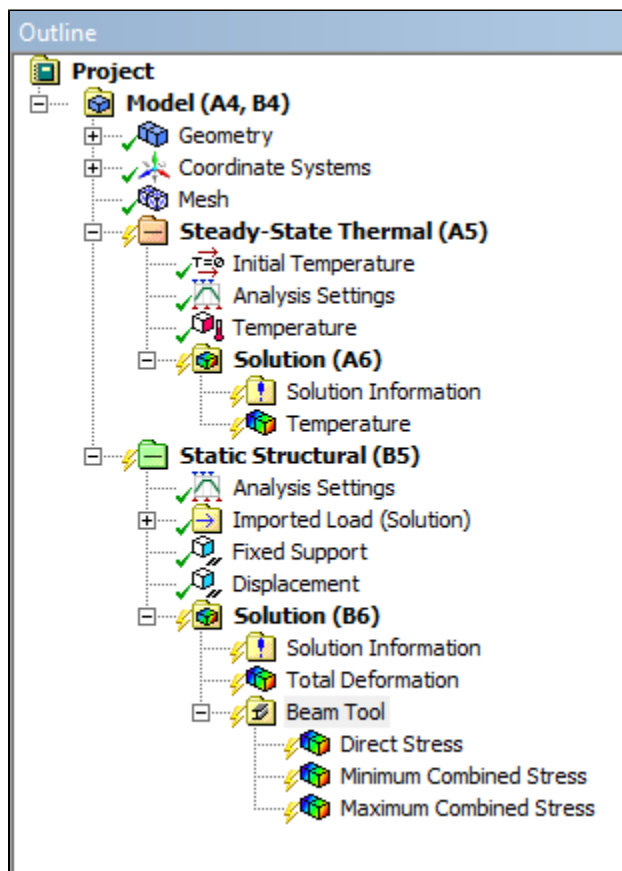
In the *Outline* window, again notice that there are two solution boxes: one for the temperature solution and one for the structural solution. We will first set up the solution for the temperature solution, to be followed by the structural solution.

Temperature Solution

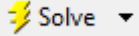
In the *Outline* window, look under  **Steady-State Thermal (A5)** and click  **Solution (A6)** to bring up the temperature solution sub menu in the menu bar. In the temperature solution sub menu, click **Thermal > Temperature** to create a temperature solution.

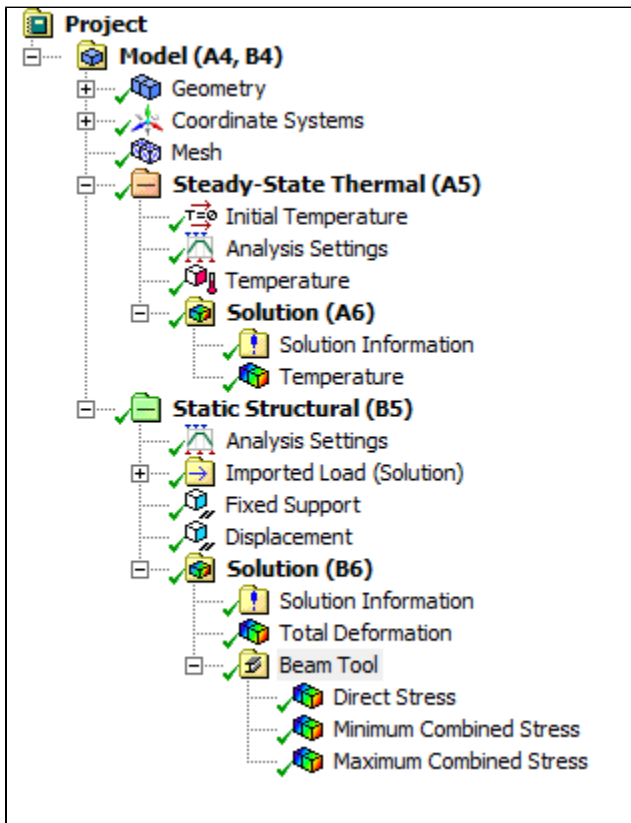
Structural Solution

In the *Outline* window under  **Static Structural (B5)** click  **Solution (B6)** to bring up the structural solution sub menu. In the structural solution sub menu, select **Deformation > Total**. This will create a deformation solution in under the structural solution. Next, select **Tools > Beam Tool** to create solutions relating to stress in the line body.



Solve

Finally, press  **Solve** to solve the simulation. Note: you may have to solve the problem twice: once to solve the thermal problem and once to solve the structural problem. You'll know if the solutions are solved once the lightning bolts in the above figure become green check-marks.



[Go to Step 6: Numerical Results](#)

[Go to all ANSYS Learning Modules](#)