6. Thermal-Stress Analysis
Thermal-Stress Analysis

• In this chapter, we will briefly describe how to do a thermal-stress analysis.

• The purpose is two-fold:
  – To show you how to apply thermal loads in a stress analysis.
  – To introduce you to a coupled-field analysis.

• Topics covered:
  A. Overview
  B. Procedure
  C. Workshop
Thermal-Stress Analysis

A. Overview

Thermally Induced Stress

• When a structure is heated or cooled, it deforms by expanding or contracting.

• If the deformation is somehow restricted — by displacement constraints or an opposing pressure, for example — *thermal stresses* are induced in the structure.

• Another cause of thermal stresses is **non-uniform** deformation, due to different materials (i.e., different coefficients of thermal expansion).
Thermal-Stress Analysis

B. Procedure

- Thermal-stress analysis involves two sequential analyses:

  1. First do a steady-state (or transient) thermal analysis.
     - Model with thermal elements.
     - Apply thermal loading.
     - Solve and review results.

  2. Then do a static structural analysis.
     - Switch element types to structural.
     - Define structural material properties, including thermal expansion coefficient.
     - Apply structural loading, including temperatures from thermal analysis.
     - Solve and review results.
Thermal-Stress Analysis

...Procedure

1. Thermal Analysis
   - The procedure for this is described in Chapter 6.

2. Structural Analysis
   a) Move to PREP7 and switch element types from thermal to structural.
      • Preprocessor > Element Type > Switch Elem Type
      • Or ETCHG command
      
      Caution: Switching element types will reset all element options back to their default settings. For example, if you used 2-D axisymmetric elements in the thermal analysis, you may need to respecify the axisymmetric option after the switch. Therefore, be sure to verify and set the proper element options:
      • Preprocessor > Element Type > Add/Edit/Delete > [Options]
      • Or use ETLIST and KEYOPT commands
b) Define structural material properties (EX, etc.), including the coefficient of thermal expansion (ALPX). (If you use the ANSYS-supplied material library, both thermal and structural properties will be defined, so this step may not be needed.)

*Note:* If ALPX is not defined or set to zero, no thermal strains will be calculated. You can use this technique to “turn off” temperature effects!

(c) Specify static analysis type. This step is needed only if the thermal analysis was a transient.

- Solution > -Analysis Type- New Analysis
- *Or* ANTYPE command
Thermal-Stress Analysis

...Procedure

d) Apply structural loads and include temperatures as part of the loading.
   • Solution > -Loads- Apply > -Structural- Temperature > From Therm Analy
   • Or use the LDREAD command.

e) Solve.

f) Review stress results.
Thermal-Stress Analysis

C. Workshop

- Refer to your Workshop Supplement for instructions on:
  
  W4. Axisymmetric Pipe with Fins