Problem Description

The thin plate (70×35) shown below is exposed to a temperature of 25 degree. When the temperature reaches 150 degree, the plate will have expansion. A fixed boundary condition of the top plate will cause changes in stress field. The thermal expansion coefficient is \(1.35 \times 10^{-5}/\degree\text{C}\). Plot the stress field at 150 degree.
ABAQUS Analysis Steps

1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)
3. In the Create Part dialog box name the part and
   a. Select “2D Planar”
   b. Select “Deformable”
   c. Select “Shell”
   d. Set approximate size = 100
   e. Click “Continue…”
4. Create the geometry shown below (not discussed here)
5. Create material property---steel
   a. Mechanical---Elasticity---Elastic, E=2E9, v=0.3.
   b. Mechanical---Expansion, coff=1.35E-5

6. Create section
   a. Create section as below

   ![Material property creation](image1)
   ![Expansion parameters](image2)

   b. Assign section

   ![Section creation](image3)
7. Assembly
   a. Create Instance part

8. Create Step
   a. Create loading step, accept default setting
9. Under Load Module,
Create initial temperature field:

a. Create predefined field to define the initial temperature 25 degree, select “Predefined field” in main menu (as shown below), select “Create”

![Image of predefined field creation](image1)

b. Create Predefined field 1, select “initial” in step, “other” in category, “temperature” in Types for selected step. Click continue…

c. Select the whole plate, click “Done”

![Image of predefined field application](image2)

d. Define magnitude of temperature field: put 25 in magnitude
10. Define the temperature increases to 150 degree
   a. Open Predefined field Manager

   b. Select “Propagated”, click “Edit”, Select “Modified” in “Status”, Change the magnitude to 150 degree. Click “OK”.

   ![Predefined field Manager window](image)

   ![Edit Predefined field window](image)
11. Create Boundary Condition
a. Define fixed boundary condition on the top: Select ENCASTRE
b. Define roller boundary condition on the left and right edge:
12. Mesh the part
a. Create seed size: Assign global size as 2.

b. Define mesh controls as below:

c. Assign element type as below:
d. Mesh the part

13. Submit the job
a. Create job: “Heat_plate”

b. Submit the job
14. View results