Homework 8

Presented: Anshul Yadav

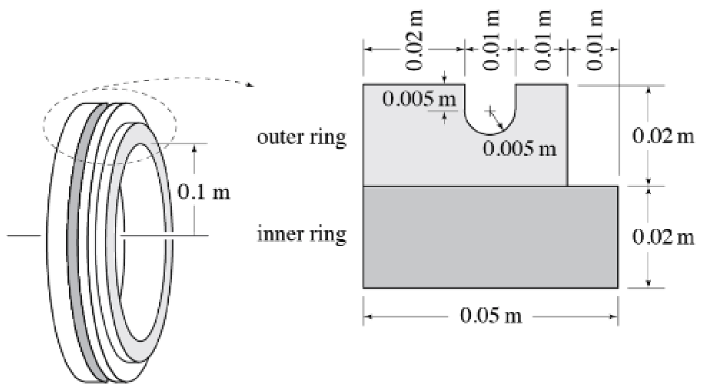
Date: December 13 September 2021

Introduction

This report is written in response to homework 8. One quarter of both rings were modelled in solidworks and then assembled. The assembly was then used in a transient analysis followed by a static structural analysis in Ansys workbench. An analytical check was also performed using hand calculations.

The Method:

The Problem:



Two circular steel rings are assembled by shrink-fitting. When the inner ring is at 20 °C and the outer ring is at 100 °C, the rings have the dimensions shown at the right and they fit together perfectly with no interference or gap. To provide sufficient clearance between the rings during assembly, the outer ring is heated to 150 °C while the inner ring is at 20 °C, the inner ring is placed within the outer ring, and then they are cooled to 20 °C. ($E = 200$ GPa, $\nu = 0.3$, $\alpha = 12 \times 10^{-6}$ /°C, $k = 40$ W/m·°C.)

Use FEA with either 4-node quad elements or 8-node quad elements (your choice) to determine the final minimum inside radius of the assembly. Assume the interface between the two rings is perfectly bonded (i.e., they are in contact and there is no slip).

Figure 1: The Problem

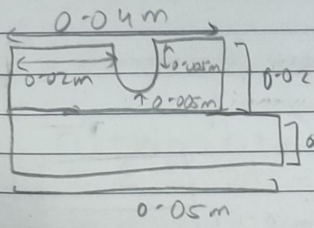
Analytical Solution:

Initially the outer ring is at 100 C and the inner ring is at 20 C. For the analytical solution, I presume that no heat is lost to the surroundings (no convection) and conduction brings both bodies to an average temperature such that the thermal energy after conduction is the same as thermal energy before conduction. After the two rings reach the average temperature (56.41 C), the entire assembly is cooled to 20 C, which results in some thermal strain. Strain can also be calculated as change in length divided by original length. Thus the new value of length is calculated. The values for the volumes were obtained from solidworks.

Date ....

APPENDIX

SHRINK FIT WITH HEAT



Initially the outer ring is at 150°C and is cooled down to 100°C when a contact is made with 20°C inner ring.

Here, I calculate the average temperature assuming no heat losses and instantaneous conduction of heat between the two rings. Then the assembly is cooled down from the average temperature to 20°C and the inner diameter is presented. Reference temperature is 20°C

$$m_1 C_v \Delta T = m_2 C_v (T_{avg} - 20°C)$$

$$m_1 80 = m_2 (T_{avg} - 20°C)$$

$$\rho_{steel} V_{outer\ ring} 80 = \rho_{steel} V_{outer + inner} (T_{avg} - 20°C)$$

$$(144.37)\, 80 = (144.37 + 172.79)(T_{avg} - 20°C)$$

$$T_{avg} = 56.4157°C = 329.566K$$

$$\varepsilon = \alpha \Delta T = -12 \times 10^{-6} \times 36.4157$$

$$= -0.000437$$

$$\varepsilon = \frac{\Delta l}{l} \Rightarrow \Delta l = -0.000437 \times 0.1$$

$$= -4.36988 \times 10^{-5} m$$

$$l_f - 0.1 = -4.36988 \times 10^{-5} m$$

$$l_f = 0.099956 m$$

Thus the inner radius is 9.9956 cm after the analysis.

Figure 2: The analytical solution.

The inner radius should be 9.9956 cm after the assembly is cooled to 20 C.

ANSYS Solution:

A Finite element model was constructed using the modelled assembly from solidworks. The material used structural steel with E = 200 GPa, poisson ratio of 0.3, $\alpha = 1.2 * 10^{-5} C^{-1}$, k = 40 W/(mC). Contact element (conta 174) and target element (targe170) were used to simulate a contact pair between, (Figure 3). Otherwise the model was entirely composed of solid 279 elements. Body sizing was used to create a fine mesh (Figure 4).

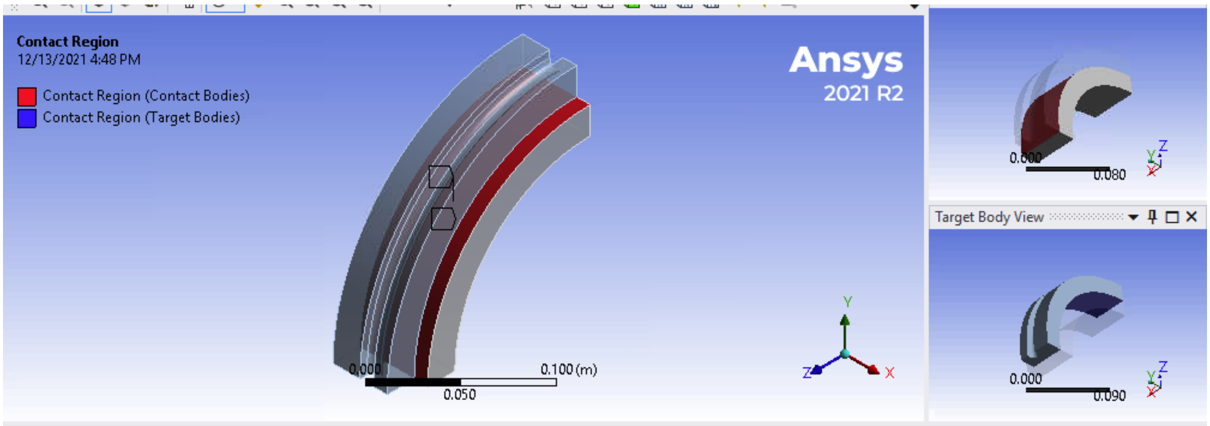


Figure 3: Contact Region

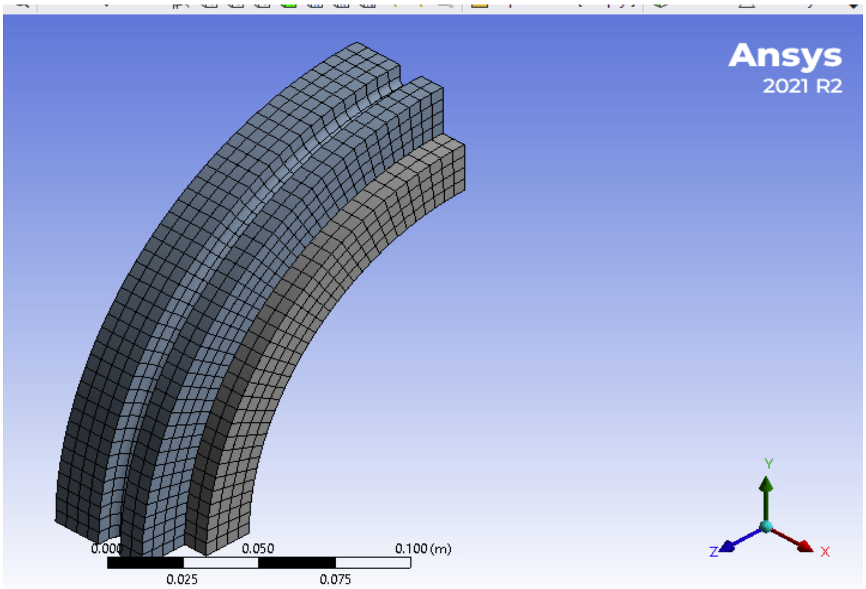


Figure 4: The mesh

Transient Thermal Analysis:
A thermal analysis module was used for thermal analysis. The results of thermal analysis were exported to a static structural analysis to figure out the final inner radius. Initial temperature (surrounding temp) was set to 20C. Reference temperature for the inner ring was set to 20 C and 100 C for the outer ring. A temperature load of 100 C was then applied on the outer ring. Since the two rings are in contact, conduction takes place of its own accord in ansys. The symmetric faces were insulated as there is no heat flow to the surroundings from the symmetric faces as I only modelled a quarter of the total rings. Temperature was applied using 2 load steps. In the first step the outer ring cooled from 100 C to 20 C and in the second step nothing happened. The temperature in figure 4 is at the end of first load step.

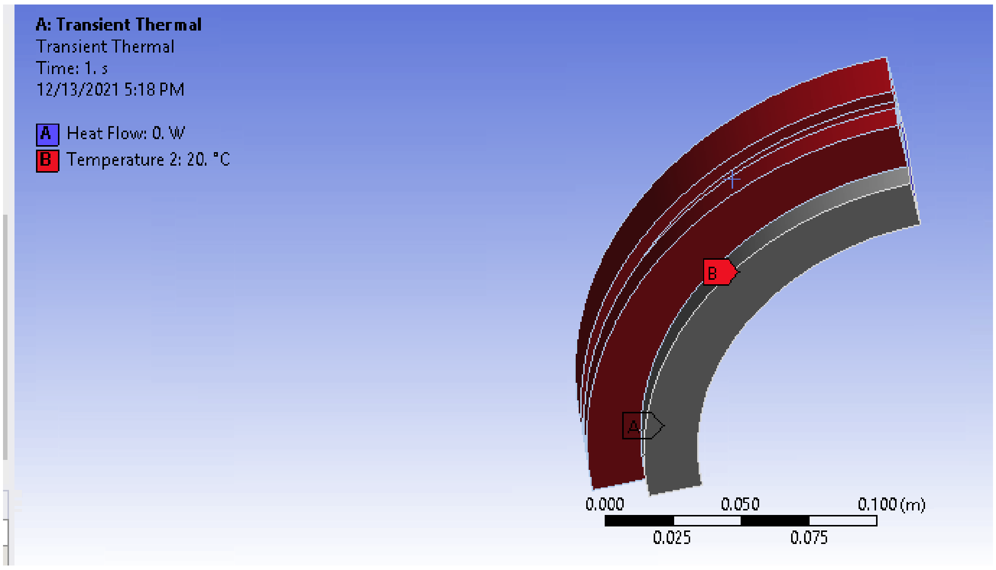


Figure 4: The Transient Thermal Modal

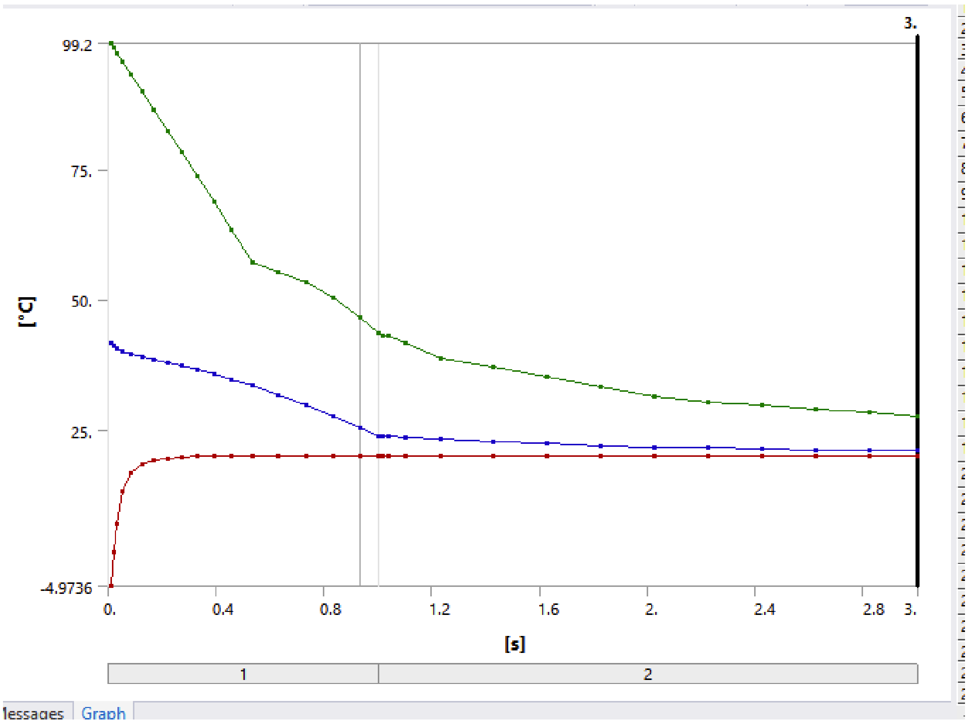


Figure 5: Results of thermal analysis

Figure 5 shows the variation of temperature for both rings. As seen in the graph, the average temperature approaches 25C at the end of the second load step. This is due to the presence of residual heat in the interior of the outer ring as the entire body of the outer ring was heated to 100 C.

Static Structural Analysis:

The results of the thermal analysis were imported in the static structural analysis. A frictionless boundary condition was applied to simulate symmetry and displacement boundary condition was applied to constrain rigid body motion. The element use was solid 186 for body bodies and conta 174 and targe 170 to simulate the contact surface.

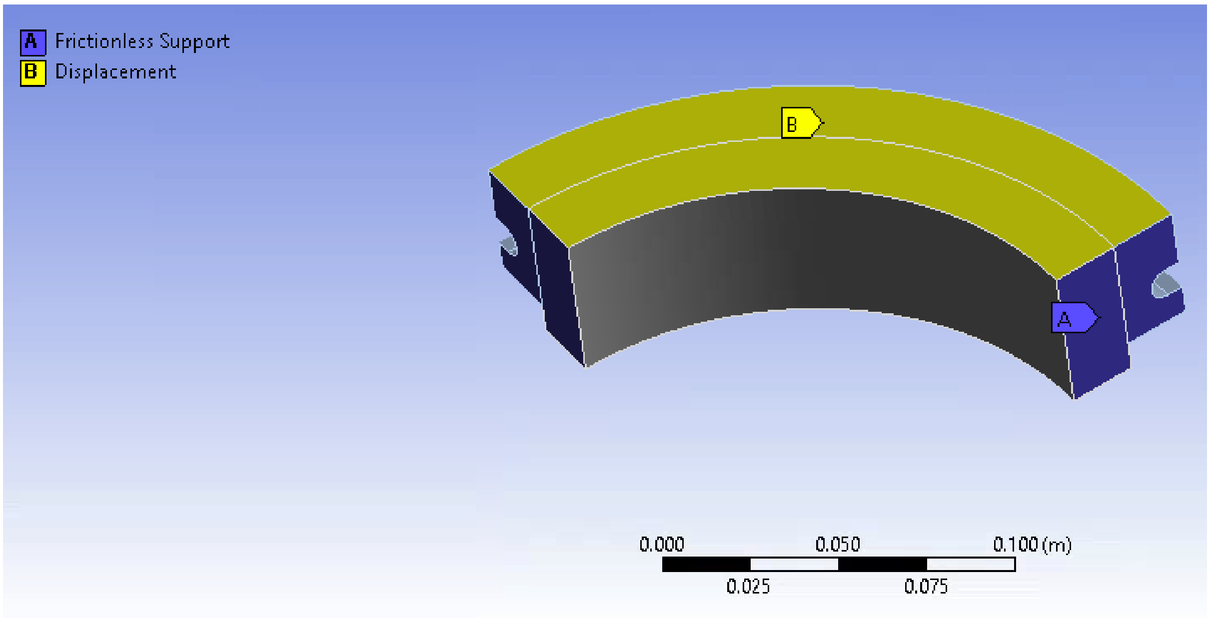


Figure 6: The static structural model.

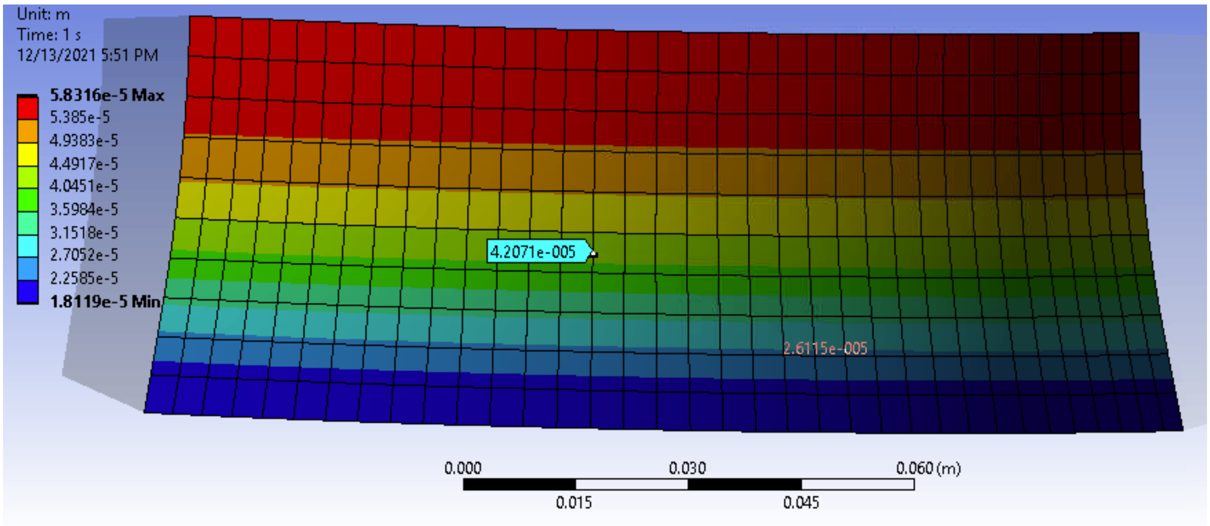


Figure 7: Results of Static structural model.

Figure 7 shows the results of static structural analysis. The max deformation observed was 5.8316e-5 m and minimum was 1.8119e-5 m. To calculate the average radial deformation, I eyeballed the average which is around 4.2071e-5 m (Figure 7). This corresponds to the radius of 0.99958m (9.9958cm) which is very close to the theoretical value of 9.9956 cm which gives credence to the ansys solution.