24-650 Applied Finite Element Analysis Homework No 6 Thermal Transient Problem Ignacio Cordova

The objective of this assignment was to perform a thermal transient analysis of a concrete block that got stuck on the surface of a steel plate during a cold winter night at -5 C and then was heated using a hair dryer.



Figure 1: System dimensions

1. Setup

The first step was to create the geometry in SpaceClaim and then import it to Ansys Mechanical in a Transient Thermal module. There is only one contact region (shown in Figure A.1), which is the region where the concrete block got stuck to the surface of the steel plate. The initial and boundary conditions are shown below:

- Initial Temperature: -5 C
- Base of plate insulated
- Convection over all the external faces that are in contact with the air (Heat dryer blows hot air at 42 C, and the convection coefficient is estimate to be 98 W/m²-C)

The properties of the materials are shown below:

- Steel: *k*=43 W/m-C , *ρ*=8100 kg/m³ , *c* =490 J/kg-C
- Concrete: *k*=1 W/m-C , *ρ*=2300 kg/m³ , *c* =800 J/kg-C

Because our interest is in the temperatures on the contact region, a temperature plot was assigned to that region. For the selection of the Step End Time, I tried some random values until the minimum value of the contact region (center of the plate) was close to 0 C (the ice starts melting).

The mesh used was the default one with a refinement element set to 3. The mesh consisted of **13,673 nodes** and **8,002 elements**. This is shown in Figure A.2.

2. Results and Analysis

As can be seen in Figure 1, when the Step End Time is **279.19 s**, the center of the plate is above 0 C, so the whole contact region melted.



Figure 1: Minimum Temperature at the Contact Region for different Step End Times

To confirm that the mesh was adequate to obtain the value of 279.19 s for the Step End Time, I calculated the %Difference of the Maximum Temperature of the Contact Region for 4 different cases:

- Default Mesh with no Refinement
- Default Mesh with Refinement set to 1
- Default Mesh with Refinement set to 2
- Default Mesh with Refinement set to 3



Figure 2: %Difference of Max Temperature at the Contact Region for different refinements (End Time= 279.19 s)

As can be seen in Figure 2, for the Refinement=3, the %Difference is -0.55%, which confirms that the mesh is adequate. All the values obtained are shown from Figure A.6 to A.9.

Finally, a plot showing the evolution of the temperature is presented in Figure 3. As can be seen, at 279.19 s the minimum temperature at the center of the plate reaches 0 C, while the maximum temperature at the corners reaches around 5 C.





3. Appendix



Figure A.1: Contact Region



Figure A.2: Mesh

24-650 Applied Finite Element Analysis Homework No 6







24-650 Applied Finite Element Analysis Homework No 6



Figure A.5: Heat Flux at 279.19 s.



Figure A.6: Temperature at the Contact Region at 279.19 s (Default Mesh)



Figure A.7: Temperature at the Contact Region at 279.19 s (Refinement=1)



Figure A.8: Temperature at the Contact Region at 279.19 s (Refinement=2)



Figure A.9: Temperature at the Contact Region at 279.19 s (Refinement=3)