DSG Ansys R&D Meeting Minutes

Date: October 19, 2023
Time: 2:00 PM – 3:00 PM

Attendees: Aaron Brown, Peter Bonneau, Pablo Campero, Brian Eng, Tyler Lemon, and Marc McMullen

1. NPS thermal analysis with Ansys Mechanical

Aaron Brown and Pablo Campero

1. Added crystal array, dividers, and cooling plate to model
   - Created name selections for air region of the crystals
   - Completed thermal conditions and materials setup
   - Mesh of the model in progress
2. Modified model of one crystal
   - Divided crystal into five regions—front, back, air, mu-metal, and carbon fiber
   - Kept thin slice volume attached to the rear face of the crystal to enable the setup of internal heat generation
3. Ran simulations of the one crystal model with different thermal conditions
   - Set up model with internal heat generation of 7.5e5 W/m3 and then same model with heat flow of 0.3 W
   - For both cases, applied convection on six walls—front, back and air region (four walls)
   - Generated temperature contour plots
   - Noted maximum temperature of ~52°C for both cases

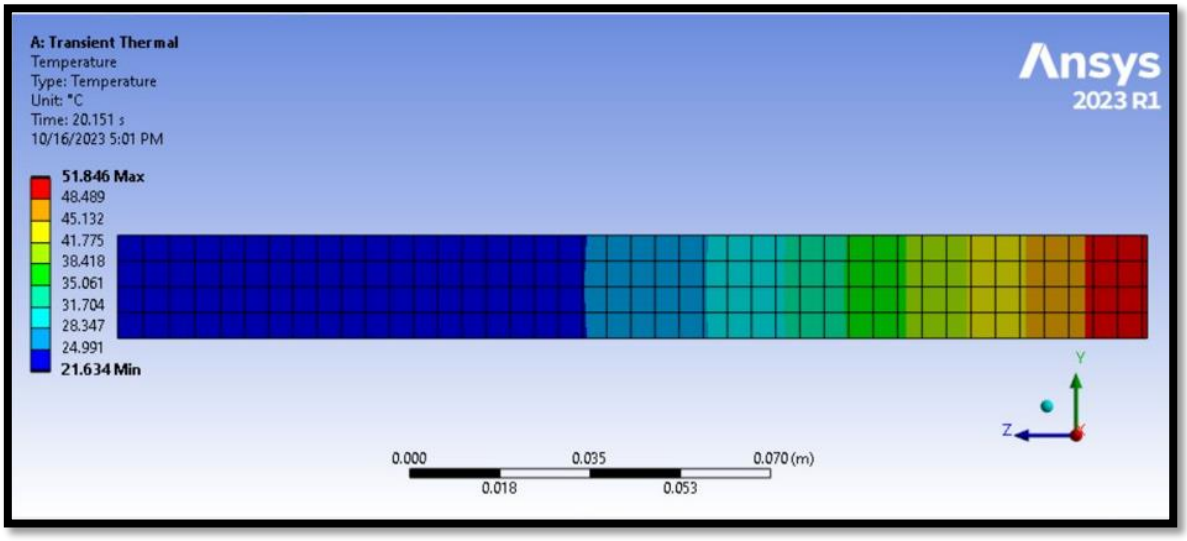

FIG. 1. Right side view of crystal temperature contour plot with internal heat generation of 7.5e5 W/m3 and convection at six walls. Maximum temperature was 51.8°C.

4. Created model with one crystal and dividers
    - Model keeps the five regions, the thin slice volume attached to the rear face of the crystal, carbon fiber divider, and mu-metal divider
    - Applied Share Topology feature to ensure contact between the surfaces of dividers and crystal
5. Ran thermal simulation for model with one crystal and dividers
    - Set up model with internal heat generation of 7.5e5 W/m3
    - Applied convection on six walls—front, back and air region (four walls)
    - Generated temperature contour plots
    - Noted maximum temperature of ~48°C

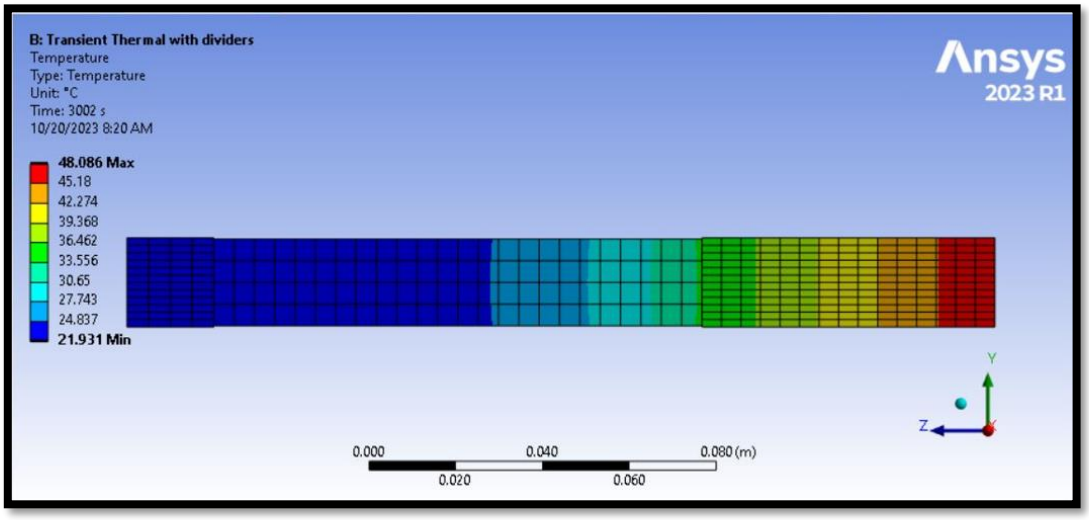

FIG. 2. Right side view, temperature contour plot of crystal and dividers with internal heat generation of 7.5e5 W/m3 and convection at six walls. Maximum temperature was 48.08°C

2. EIC beampipe Ansys Fluent thermal analysis

Pablo Campero

1. Created a beampipe model with three regions
    - Central region made of beryllium and the next left and right regions made of aluminum
    - Dimensions were based on the design drawings and CAD model provided by EIC designers
    - Created internal volume for the air thermal effect simulations
    - Created inlet and outlet regions for the air volume inside the beampipe; inlet left side and outlet at right side

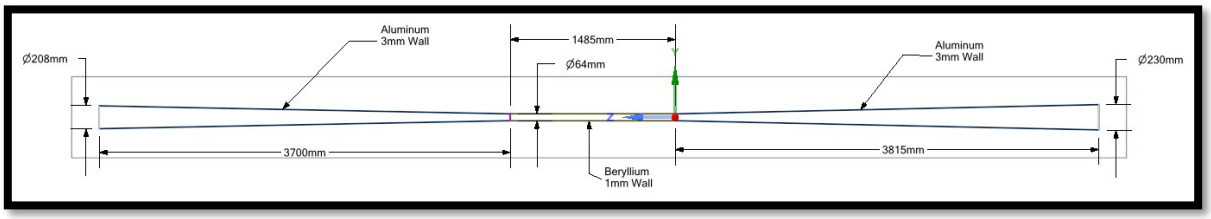

FIG. 3. Beampipe simplified model with beryllium and aluminum sections

2. Completed mesh for model; generated two layers for the beryllium section
    - Total number of cells ~9 million
3. Setting up model materials and boundary conditions in Fluent