EIC Beampipe Section Test Stand Thermal Simulation

Pablo Campero
2023-03

With the aim of validating some of the results obtained on the thermal simulation performed in Ansys-Fluent for the Electron Ion Collider (EIC) beampipe central section, specifically the thermal effect on the first silicon sensor tracking layer, the Detector Support Group members assembled a test bench to attempt to replicate, available and on-the-shelf components were used with similar dimensions to the EIC beampipe. Material for the pipes, separation between beampipe and pipe used to simulate the first silicon layer, and air flow velocity were different.

Since there were some considerable variables between the test stand and the simulated beampipe section model in Ansys, I decided to replicate the test stand with the same exact dimensions and materials in Ansys Fluent to later compare its results with the measurements done in the test stand.

The model consists on a heater immersed in mineral oil within an aluminum pipe, an aluminum pipe to represent the beryllium beampipe, and another aluminum pipe with bigger diameter to represent the silicon sensor layer 1, see fig.1. Additionally, because of the geometry of the model, I created a volume between the heater pipe and inner volume (empty space) of the beampipe to ensure heat transfer in more than one tangential point

The Ansys-Fluent simulated model dimensions were based on the actual components used in the test stand and are show in table 1.

Part Name	ID [in]	OD [in]	Thickness [in]	Length [in]
Heater pipe	N/A	1.9	N/A	12.00
B`eampipe	2.43	2.50	0.07	
Si Sensor Layer 1	3.50	3.06	0.44	

Table 1. Model dimensions

- **Modeled EIC-Beampipe central section test stand setup by DSG**
- **Implemented material properties, thermal boundary conditions, and cell zone conditions**
- **Ran multiple simulations for different air flow velocities**
- **Compared Ansys-Fluent results with previous measurements done in the EIC test stand model**

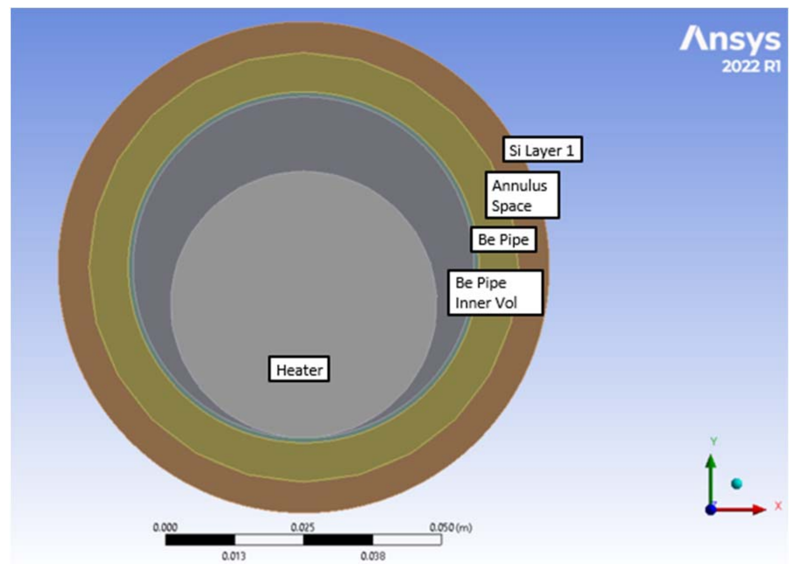

Fig. 1. EIC Beampipe central section test stand model

EIC Beampipe Section Test Stand Thermal Simulation

As noted from table 1 there is a separation of 0.280 in (~7.11 mm) between the represented beampipe and silicon sensor layer 1 pipe.

There was a variation between the test stand and the simulated model on the way of flowing the air through the annulus space formed between the beampipe and silicon sensor layer 1; for the test stand the air was introduced from four ntp push lock connectors located radially with respect to the concentric center, for the Ansys-Fluent simulated model the air was flowed in the axial direction from the inlet end. Also, for the test stand model the atmospheric air had contact at the outlet end while the simulated model was set with no interaction with the atmosphere.

Once the geometry was completed using Ansys-SpaceClaim software, I proceeded with the meshing of the model, for which I encounter issues due to the geometry presented between the heater pipe and inner volume of the beampipe, I solve issues by sharing topology and removing previously set contact regions. I used sweep, inflation, and face sizing meshing methods to get a optimum mesh of the model.

I implemented the model to Ansys-Fluent and set the set boundary conditions and model configurations as show in table 2. In the generated contour temperature plot, I placed the temperature probes approximately where the RTD temperature where located in the test stand. Fig.2 shows the comparison between the simulation resulted temperatures and the measured temperature at the silicon sensor layer 1, I observed that the temperature simulated results are close to the measure values when the air flow velocity is greater than 0.5 m/s.

I plan to add the four inlets in radial direction, a cap to the inlet, and allow interaction with the atmosphere to see changes on the temperature. Also a test stand model with aerogel included is planed

I completed the Ansy-Fluent thermal simulation of EIC silicon sensor layer 1 test stand obtaining expected and similar results to the measurements done previously in the lab.

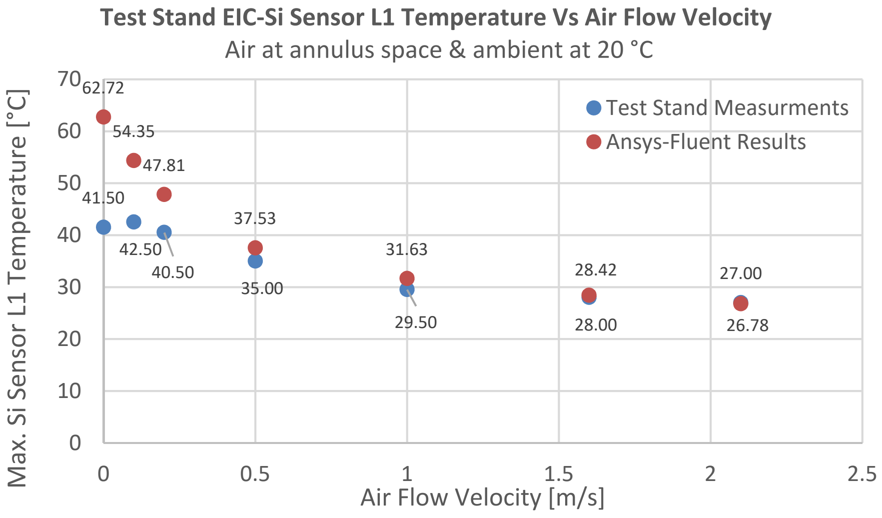

Fig.2. Comparison between Ansys-Fluent results and measured values for the maximum temperatures for the silicon sensor layer 1 test stand model

Solver	Fluid Flow Fluent , pressure-based
Model	k-omega, Shear Stress Transport (SST)
Heat Transfer	Convection and radiation
Precision	Double
Simulation Iterations	1000
Heat source for system	83229.9 [W/m^3]
Air temperature	20 °C
Air flow velocity	0 to 2.1 m/s

Table 2. Boundary conditions