EIC Beryllium Pipe Section - CFD Thermal Analysis

Pablo Campero
2022-04

EIC Beryllium Pipe Section - CFD Thermal Analysis

I continued with the thermal analysis of the Electron Ion Collider (EIC) beam pipe central section, which is to be made of beryllium and will be heated to 100°C to ensure a good vacuum.

I used Computational Fluid Dynamics (CFD) tools such as *Ansys Fluid Flow Fluent* software. I imported the simplified model of the beryllium pipe (Be pipe), the silicon sensor layer 1 (Si sensor L1) and the ambient enclosure. The model was based on the actual dimension and component locations provided bythe original design (Table 1) but with the length of the Be pipe (actual length 1450 mm) shortened to 320.5 mm.

Part Name	ID [mm]	OD [mm]	Thickness [mm]	Length [mm]
Beryllium pipe	62.00	63.52	0.76	1470.00
Silicon sensor L1	66.00	66.08	0.04	320.50

Table 1. Model dimensions

I created various models with different separation between Be pipe and Si sensor L1: 1.24 mm (Base model), 2 mm, and 3 mm. Once the geometry was completed, I imported the model to Ansys Fluent and proceeded with the assignment of the domains for each component of the model. I set fluid domain for the annulus space and enclosure volumes, and solid domain for Be pipe and Si sensor L1. I meshed the model with a 3 mm element size; meshing presented issues due to the irregular geometry of the model. I setup Si sensor L1 component with only one meshing cell layer due to its thickness (0.04 mm).

- **Used *Ansys Fluid Flow Fluent* software to to find the temperature of the silicon layer.**
- **Ran simulation with different separation between Be pipe and Si sensor L1 at different temperatures and velocities for the air in the annulus space and enclosure.**

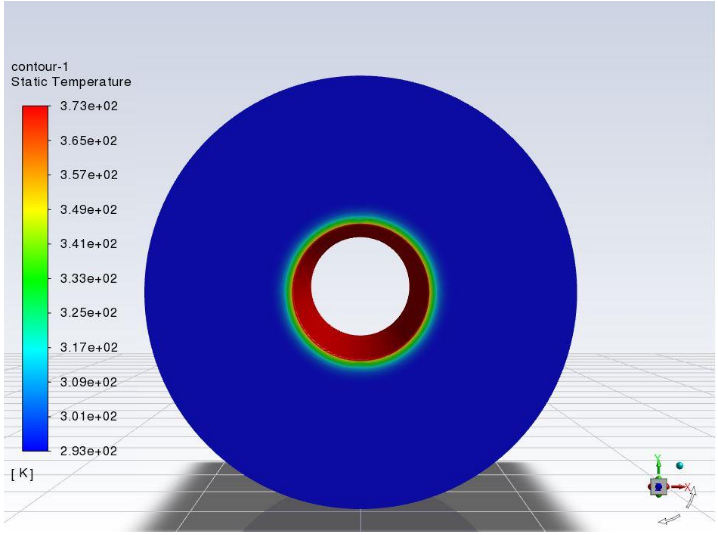

Fig. 1. Temperature profile for model with 2 mm of separation between Be pipe and Si sensor L1 with air at 15°C

EIC Beryllium Pipe Section Steady State Thermal Analysis

I had problems setting up the material for the model components, especially for the solids since the library available in the *Ansys Fluid Flow Fluent* doesn't have beryllium or silicon. To solve this issue, I created costume material and entered the thermal properties and density manually.

I configured a viscous model K-omega, shear stress transport model, recommended for turbulent flows. In the boundary conditions the annulus space and enclosure were defined as the inlet and outlet of the system. I selected a double precision for the calculations, forced convection thermal heat transfer with air in the annulus space and enclosure at 20°C, the temperature at the inner face of the Be pipe was 100°C .

I ran simulations for each generated model at different temperatures for the air in the annulus space and enclosure, 20, 18, 16, and 14°C, see fig. 1 and 2. Table 2 below shows an example for the results of simulation.

Ambient Temp [°C]	Annulus Temp [°C]	Separation between Be pipe and Si sensor L1 [mm]	Air flow Velocity [m/s]	Si sensor L1 Max Temp [°C]
18	18	1.24	5	57.99
		2	5	51.39
		3	5	41.64

Table 2. Maximum temperature for Si sensor L1 for model for various separation between Be pipe and Si sensor L1. Air temperature in the annulus space and in the enclosure (ambient) at 18°C at 5 m/s.

Observed about 10°C difference between each model. Results for simulation at the other temperatures followed the same trend.

I plan to continue with the thermal analysis for a model with 5 mm of separation between Be pipe and Si sensor L1. Improve meshing and add additional details for the model work on the post processing to visualize data are some of the next goals.

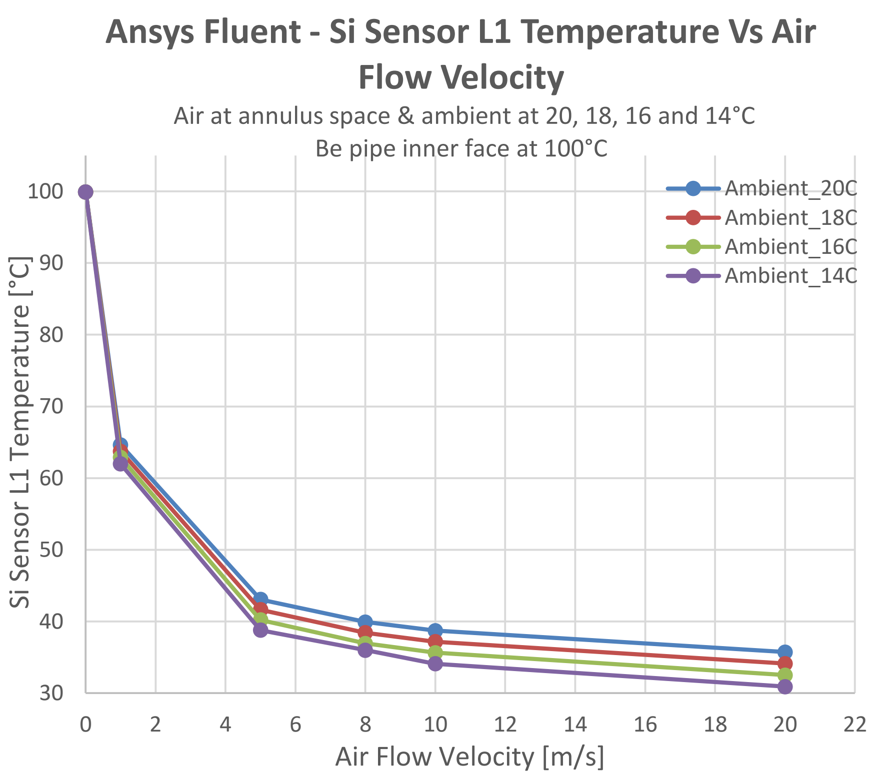

Fig.2. Maximum temperature for Si sensor L1 for model with 3 mm of separation between Be pipe and Si Sensor L1.