

EIC: Thermal Analysis of Beampipe

Brian Eng
2022-02

Ansys Computational Fluid Dynamics (CFD) Analysis

To better understand the interaction between the heated beampipe (mainly done to remove any residual contaminants) and the first layer of the silicon pixel detector, the analysis solution is being switched from a steady state to fluid.

Per vacuum experts, the inside wall of the beryllium beampipe must reach at least 100°C at a minimum. The first layer of silicon in the current detector model is separated from the beam pipe by ~1.25 mm radially.

Ansys Workbench is the main program where the analysis systems for the simulation can be chosen for a project, e.g. steady-state thermal, transient thermal, harmonic response, fluid flow, etc. Within the fluid flow analysis, there are several choices available for the analysis system, but this depends on the options selected during Ansys installation. Initially only CFX was available; after modifying the installation, Fluent and Polyflow became available, Fig. 1. The focus on this paper will be between CFX and Fluent, namely their workflow.

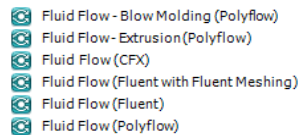


FIG. 1. Current Ansys Workbench standalone analysis systems for fluid flow.

I found that both CFX and Fluent projects have the same steps, Fig. 2, for a fluid flow simulation, namely: Geometry, Mesh, Setup, Solution, Results.

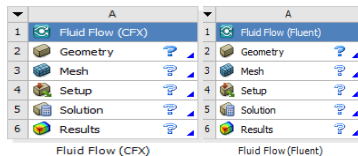


FIG. 2. Ansys Workbench standalone project steps for CFX (left) and Fluent (right).

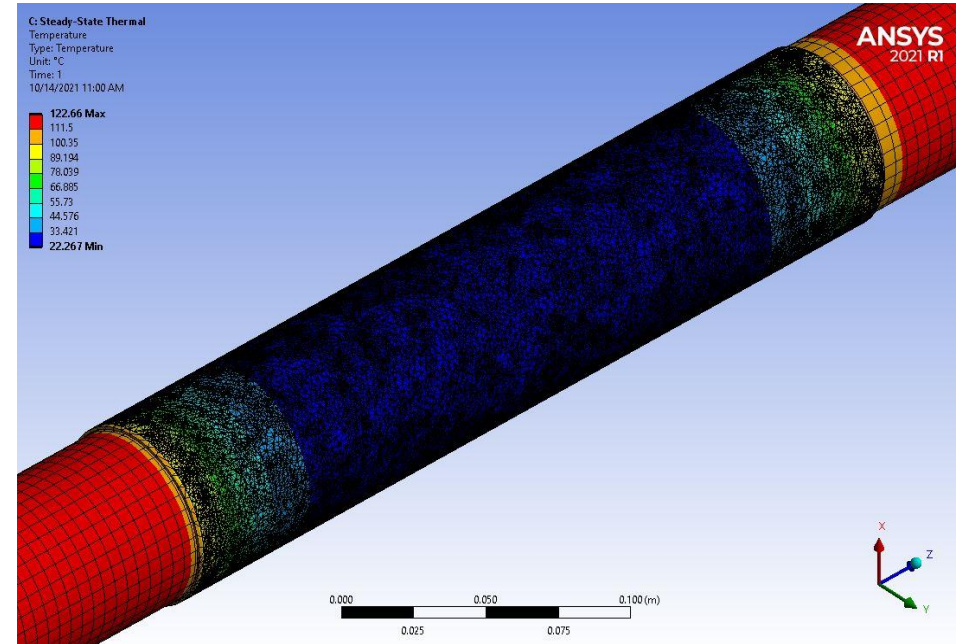
- **Setting up simulation, using CFD, for thermal interaction between heated beampipe and first silicon layer**
- **Found license checkout process depends on which CFD software is selected**
- **Procured additional license as Jlab has only a single floating CFD solver license and it is in use continually**

EIC: Ansys Thermal Analysis for Beampipe

The functions for CFX and Fluent are the same but how they ended up being used in practice presented some issues. I determined that all of the steps involve configuring various options, which must be completed successfully prior to the next step being able to execute, e.g. if the geometry step is incomplete or fails the mesh step will not run. The geometry step allows one to either import or design a new geometry for the 3D model being simulated. The mesh step applies a mesh to geometry from the previous step. The first two steps are similar between CFX and Fluent. The setup step is where all the details for the simulation are added: what materials are being used, the boundary conditions between bodies, initial parameters (flow, temperature, etc.), among some of the similar parameters, but as CFX and Fluent perform their calculations slightly differently (CFX being vertex-centered and Fluent being cell-centered), their setup is slightly different. The solution step allows for changes to the simulation itself, number of runs, convergence, etc. The final result step takes the output from the solution step and allows for various plots and visualizations to be set up.

Aside from the calculation method and slightly different setup steps, I found that the biggest difference between the CFX and Fluent analysis is when a license is checked out for the software to run. Both software requires the following licenses to run: `cfdbase`, `cfdsolve_level1`, and `cfdsolve_level2`. For CFX these are checked out only when the solution step is actually running, meaning it is possible to set up the initial conditions and the solution itself without any licenses. For Fluent the licenses are checked out at the start of the setup step, meaning if there are no licenses available no work can be done.

As there is really only a single available CFD license, this has made progress very slow, as very often all available licenses have been in use. This makes iteration and refinement on the analysis slower than it could be. Currently another CFD license package is being procured, which should help alleviate this issue.



L1 silicon layer around beampipe (red)