Conjugate Heat Transfer Analysis in a 3D printer nozzle

Kevin Mathew Thomas
Undergraduate Student
Department of Mechanical Engineering
National Institute of Technology Tiruchirappalli, Tamil Nadu, India

ABSTRACT

The objective of this project is to study chtMultiRegionFoam and highlight the temperature distribution on critical components of the nozzle geometry through the open source CFD package OpenFOAM. The nozzle consist of 2 cooling fans/inlets and one outlet. Geometry is sourced directly from CREALITY, meshed in Salome and exported to OpenFOAM. Details regarding Flow properties are listed in the table below.

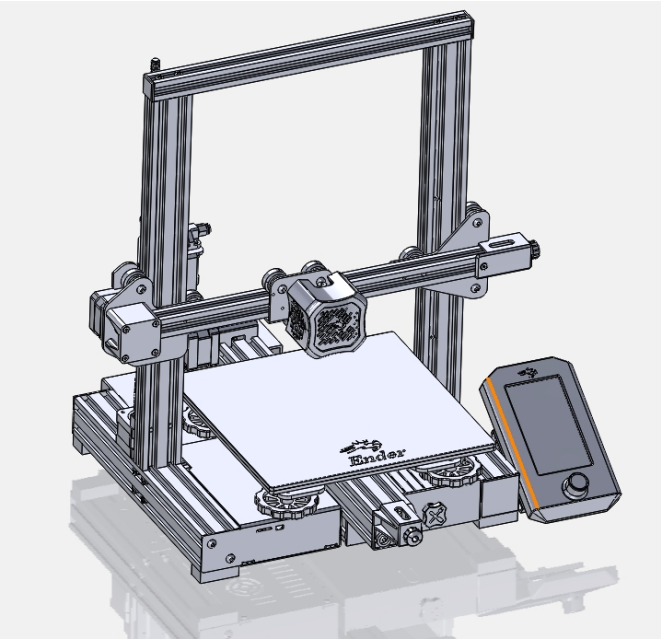

Figure 1: Ender 3 v2 3D Printer

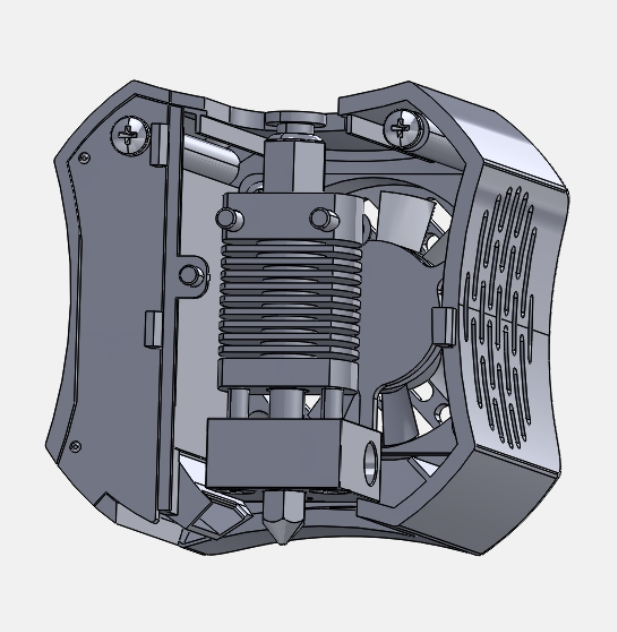

Figure 2: Nozzle Geometry (Close View)

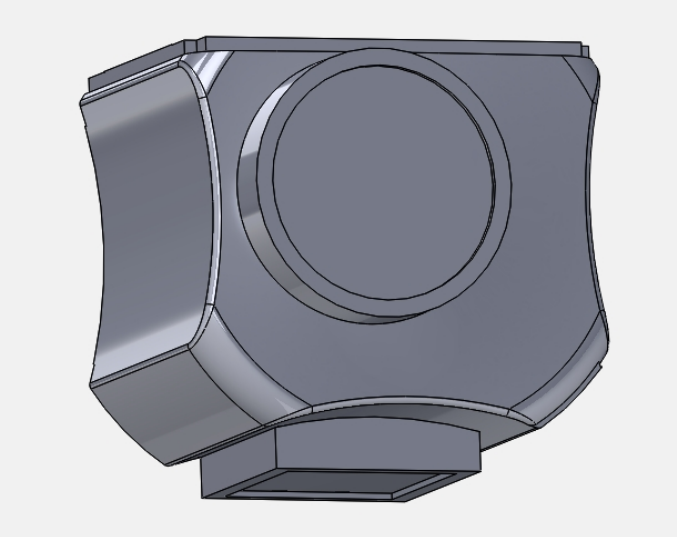

Figure 3: Cleaned up geometry (For simplicity and reduction in mesh generation time)

Fluid Property- Air	Kinematic Viscosity = 15.7 e-6 m^2/s
Flow Conditions	Inlet 1 flow rate: 8cfm
	Inlet 2 flow rate:8cfm
Nozzle Temperature	200 °C

Table 1: Tabulation of Flow conditions and other properties