

A Skill Based Mini Project Report

On

**Simulation of Shell & Tube parallel flow heat exchanger using
ANSYS fluent**

Submitted by

Ayush Tiwari (0901ME201050)

Under the guidance of

Dr Manoj Kumar Gaur, HOD

Department of Mechanical Engineering,

MITS, Gwalior

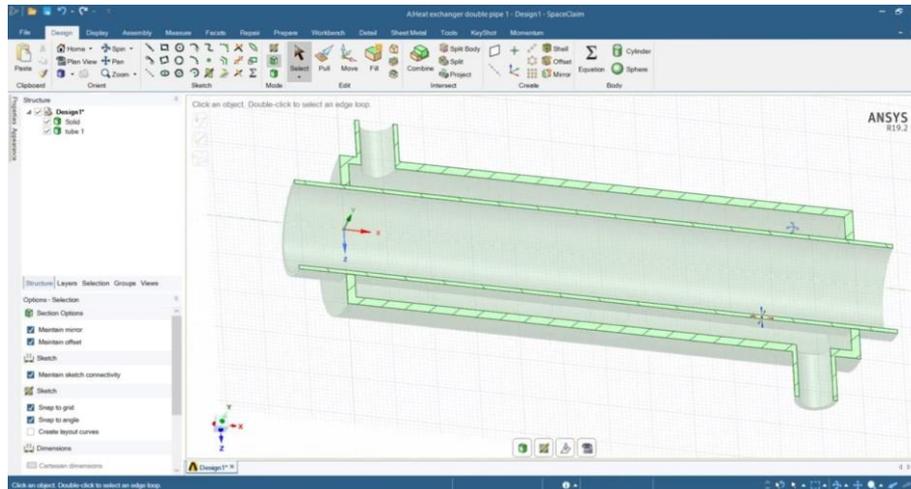


**DEPARTMENT OF MECHANICAL ENGINEERING
MADHAV INSTITUTE OF TECHNOLOGY & SCIENCE
GWALIOR-474005 (M.P)**

Introduction

A counter flow concentric tube heat exchanger is a type of heat exchanger that consists of an inner tube surrounded by an outer shell. The hot and cold fluids flow in opposite directions through the inner and outer tubes, respectively. This type of heat exchanger is often used in various industrial and HVAC (heating, ventilation, and air conditioning) applications.

Model Description



To analyse the performance of a counter flow concentric tube heat exchanger using ANSYS Fluent, you will need to set up the geometry of the heat exchanger in the software and specify the appropriate boundary conditions and fluid properties. You will also need to define the flow and heat transfer models that will be used to simulate the heat exchange process. The dimensions of the model are as follows:

- Outer shell diameter: 40 mm
- Inner shell diameter: 36 mm
- Outer tube diameter: 26 mm
- Inner tube diameter: 22 mm
- Outer Shell length: 120 mm
- Inner Shell length: 140 mm
- Shell thickness: 2 mm
- Process inlet and outlet diameter: 8 mm

Boundary Conditions

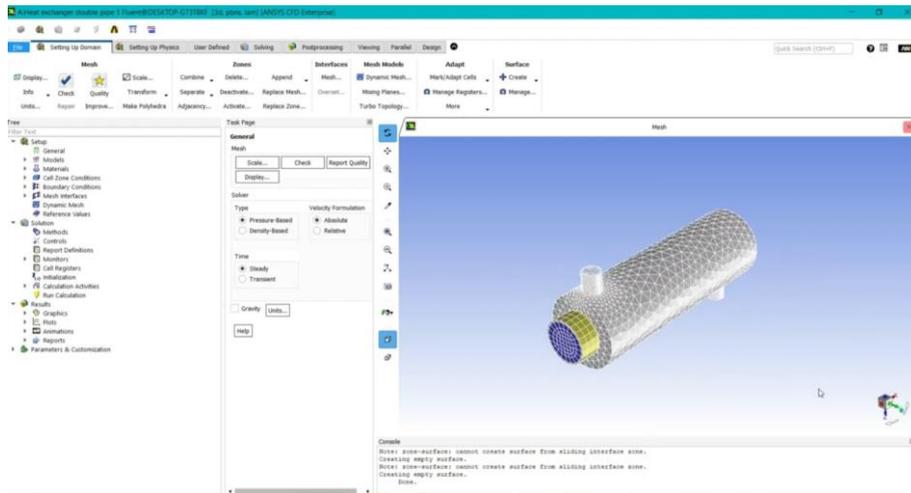
- Shell inlet velocity: 2 m/s
- Shell inlet temperature: 303 K
- Tube inlet velocity: 4 m/s
- Tube inlet temperature: 330 K

Once you have set up the simulation, you can run it and analyse the results. Some of the key parameters that you may want to analyse include the heat transfer coefficient, the overall heat transfer rate, and the temperature distribution within the heat exchanger. You can also plot graphs or

AT's

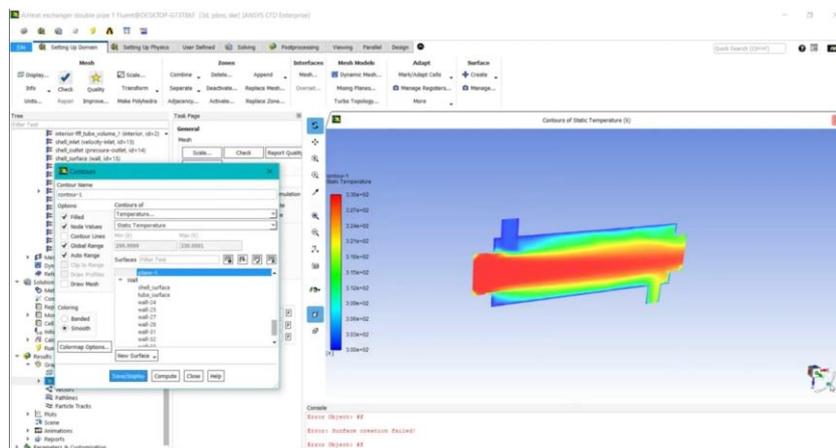
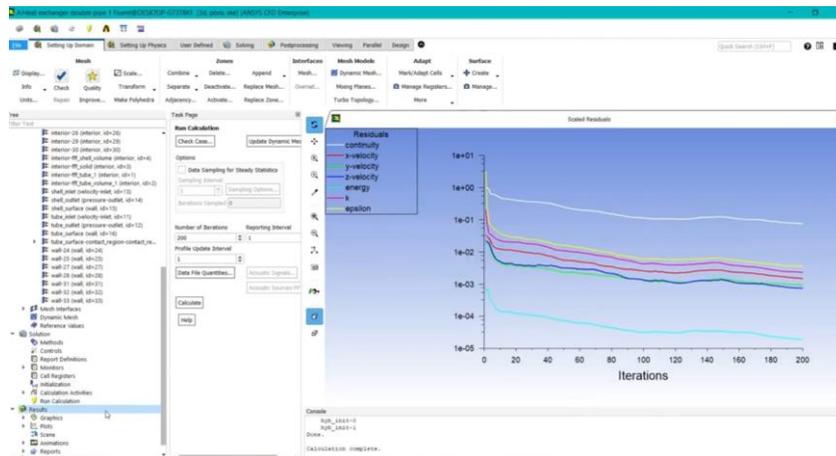


create contour plots to visualize the heat exchange process and compare the results with experimental data or design calculations.



It is important to note that the accuracy of the simulation results will depend on the quality of the input data and the appropriateness of the models and assumptions used. It may be necessary to iterate the simulation to refine the results and ensure that they are accurate and reliable. Hence the following simulation is iterated 200 times.

Results



ATC's

