SCM ENGINEERING SERVICES

Technical Report on

CFD SIMULATION OF A CALANDRIA FLOW AND TEMPERATURE DISTRIBUTION INSIDE A NUCLEAR REACTOR

Executed By:

S Sunil Krishna
sunilkrishna.s@scmengg.com

S Sampath Kumar
sampath.s@scmengg.com

Technical Director

Dr. Subhash Deo
subhash@scmengg.com

SCM Engineering Services
Plot #156, First Floor,
Motilal Nagar,
Begumpet,
Hyderabad 500019,
India.
Ph: 040 – 65449094
Email: projects@scmengg.com
URL: http://www.scmengg.com

CONTENTS

1. INTRODUCTION

The present study involves, CFD simulation of calandria flow and analysis of temperature distribution inside a nuclear reactor. Basically it is a shell and tube heat exchanger in which the fluid having higher temperature is flowing through tube and that has lower temperature flows through the shell. In our case we are considering the shell fluid only.

The study was carried out in the procedure given below,

1. CAD cleaning
2. Meshing
3. Fluid flow Analysis

2. SOFTWARE AND HARDWARE USED

2.1 Software

The analysis consists of Pre–processing, analysis and post processing followed by the result synthesis. Mesh generation being critical to the analysis was done using Star-CD. Boundary conditions definition and post processing results analysis were carried out using Star-CD.

2.2 Hardware

The hardware used for mesh generation, pre processing, post processing and the Star-CD solver was Intel based windows platform. The work was carried out on P4 processor, with 2 GB RAM.

3. GEOMETRY AND MESH GENERATION

3.1 CAD clean up and mesh generation

3.1.1 CAD Clean Up

In order to simplify and making the geometry ready for meshing, a CAD clean up procedure was done using CATIA and steps indicated below:

- Removing the unwanted surfaces and holes from the original igs file supplied.
- Surface trimming and using other techniques to get a closed surface
- Surface generation by closing edges

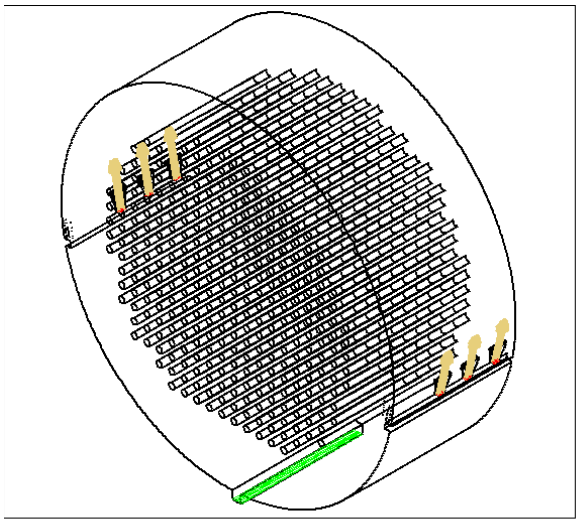

Fig 1. Schematic view of heat exchanger model

3.1.2 Mesh generation

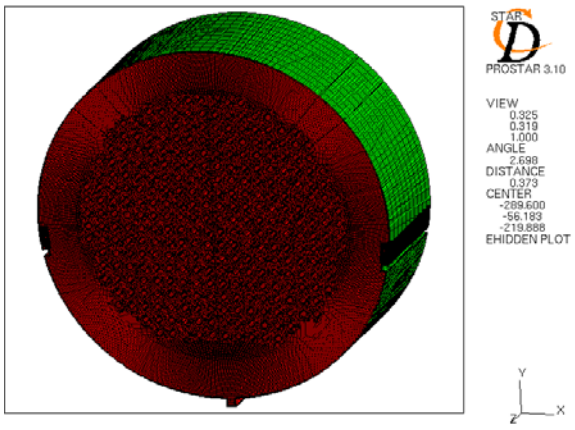

Fig 2. Finished mesh used for analysis

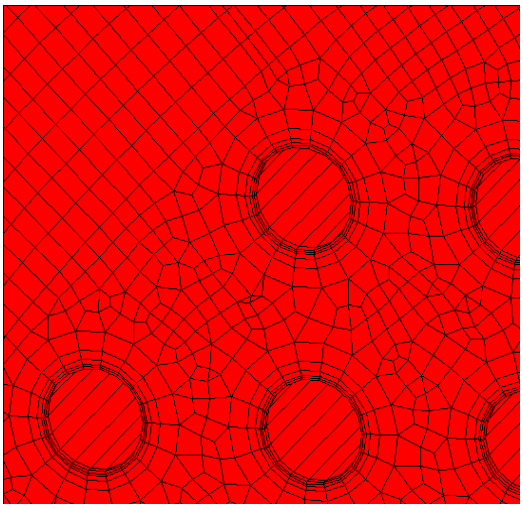

Fig 2. Mesh showing surface layers around the tubes.

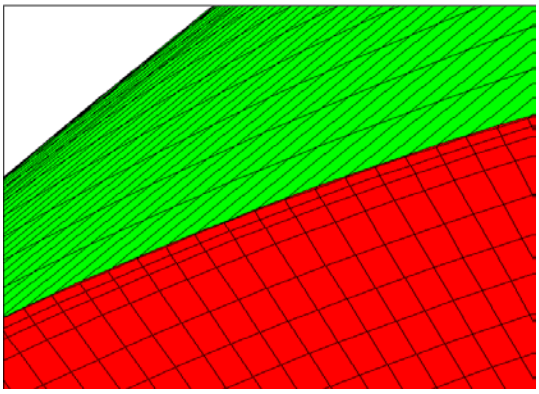

Fig 3. Mesh showing surface layers near to shell surface.

4. PROBLEM SET UP

4.1 Complexities Involved

- Large Vol. of Heat Generation
- Mixed Convection
- Flow Obstruction
- Buoyancy Forces

4.2 Assumptions

- The Calandria is symmetry about the horizontal plane
- The properties of the moderator fluids are constant and there is no change of phase.
- Boussinesq approximation is applicable for modeling Free Convection

4.3 Model Specifications

Archimedes No.	:	0.06
Reynolds No.	:	1580
Flow Rate	:	26 lpm
Inlet Temperature	:	32.6^{o}C
Outlet Temperature	:	34.8^{o}C
Volumetric Heat Gen.:		32957.76 w/m3
Turbulence Model	:	Low Re No. k-ε model

5. RESULT OVERVIEW

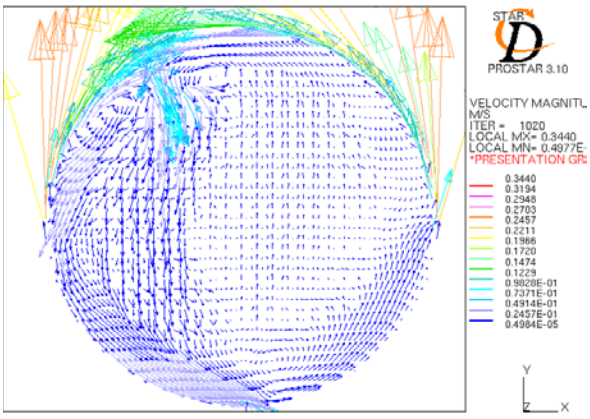

Fig. Velocity vector shows the fluid flowing direction in the heat exchanger.

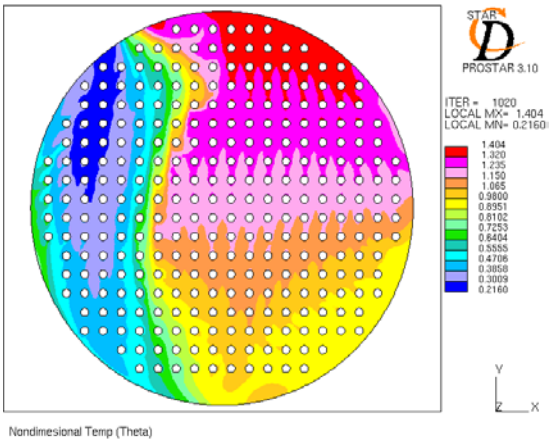

Fig. Contour plot showing temperature distribution across the heat exchanger

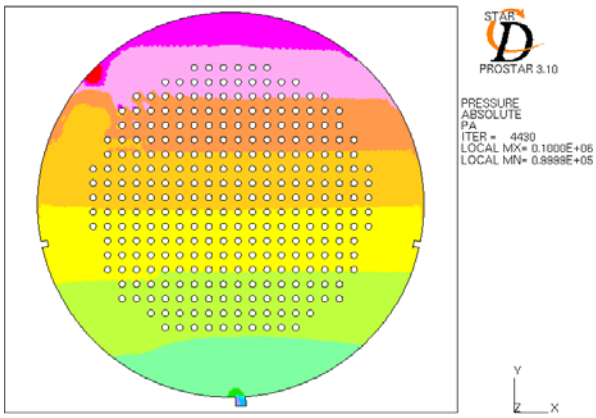

Fig. Contour plot showing pressure distribution

6. CONCLUSION

- The predicted results stands good from the heat transfer point view.
- The maximum flow velocities and the maximum temperatures reached are quite comparable with those of original flow configuration.
- This shows the usage of CFD in handling complex problems in terms of physics as well as geometry.