CFD Analysis of Helical Coil Finned Tube Heat Exchanger for Helium Liquefaction Plant

Arpit D. Patel1 Dr.S.M.Mehta2

1P.G Student 2Professor

1,2L.D. College of Engineering, Ahmedabad, Gujarat, India

Abstract—Heat exchanger is the most crucial component in cryogenic processes, for improving efficiency and decreasing cost of the whole plant. This thesis focuses on the analysis of flow of fluid through a helical coil tube fin type heat exchanger through CFD software. Also on the enhancement in convective heat transfer in between the fluid and the surrounding surface in these helical coils which has been a major topic of study as reported by many researchers.

In this study, an attempt has been made to study the counter flow of inner hot fluid flow and outer cold fluid flow, which are separated by copper surface in a helical coil heat exchanger. The temperature contours, velocity vectors, total pressure contours, total heat dissipation rate from the wall of the tube were calculated and plotted using CFD. The materials for the study were decided to be copper for helical tube and stainless steel for shells due to their better conducting properties and fluid taken was helium.

Key words: Coil Finned Tube Heat Exchanger, CFD Analysis, Helical Tube Flow

I. INTRODUCTION

Helium is widely used in space research, superconducting magnets and medical fields. Helium gas is widely liquefied using Collins cycle [1]. And heat exchanger largely drives the performance of this liquefaction system. In fact, a cryogenic liquefier will produce no liquid if the heat exchanger effectiveness is less than approximately 85% [2]. So a heat exchanger should be designed in a manner to have optimum effectiveness with lower pressure drop. [7]. According to Gupta Prabat Kumar [4], pressure drops in finned tubes used in helical coil tube type heat exchangers are found to be higher than 1.8–3.5 times as compared to the smooth tubes and depend on the mean diameter of coiled finned tube heat exchanger. Heat transfer in helical coils has been experimentally investigated by Seban and McLaughlin (1963) both for laminar and turbulent flow regimes for flow of water for constant wall flux. The range of Reynolds number studied was 6000–65,500 and the Prandtl number variation was from 2.9 to 5.7. Fluid flows play the key role in the working process of many modern engineering devices. Designing of these devices for the required operational parameters is impossible without reliable prediction of characteristics of these flows.

Computational Fluid Dynamics (CFD) deal with simulation of fluid flows with heat and mass transfer in various engineering objects. Basically it simulate the numerical model. The primary objective of this thesis is to analyze the optimized dimensionalised coil finned-tube heat exchanger by CFD and predicting pressure, velocity and to determine the fluid flow pattern in helical coiled heat exchanger. And quantitative insight into the heat transfer process that occurs when a fluid flows in a helically coiled tube.

A. Coiled finned-tube heat exchangers

Coiled finned-tube heat exchangers have been used in small and medium helium refrigerators/liquefiers, miniature J–T refrigeration systems for many years. In coiled finned tube heat exchanger the finned tubes are helically wound on mandrel and shield is provided on the outside of the coil. Fins are generally used on the outside of tube, but they may be used on the inside of the tubes in some applications. They are attached to the tubes by a tight mechanical (press) fit, tension winding, adhesive bonding, soldering, brazing, welding, or extrusion. Commercially available typical fin densities for flat fins vary from 250 to 800 fins/m, fin thicknesses vary from 0.08 to 0.25mm. The working fluid such as helium, nitrogen, argon at high pressure flows inside the helically coiled finned tube, and returns over the fins after expansion through an orifice at the end of heat exchanger.

Tube-fin exchangers can withstand high pressures on the tube side. The highest temperature and pressure is again limited by the type of bonding, materials employed, and material thickness.

B. Friction factor and pressure drop

Major requirements of heat exchangers used for helium liquefaction plant are high effectiveness and low pressure drops to stipulate limits in both of fluid streams. To fulfill these design requirements, an accurate evaluation of heat transfer and friction factor coefficients are needed. In many heat exchanger designs, especially for low-density fluids such as gases, the frictional power expenditure is considerable. In addition to compressor power requirement, like in conventional heat exchanger, pressure drop design for cryogenic heat exchangers is extremely important from the point of view of overall performance of cryogenic systems.

Tube side pressure drop across the heat exchanger will reduce the amplitude of high pressure stream thereby reducing area of expansion space in PV diagram and the gross refrigeration produced by refrigerator/liquefier.

Pressure drop across heat exchanger directly affects the liquid yield obtain at the end of liquefaction cycle which is Collins cycle for helium liquefaction. Maximum enthalpy drop must be take place across joule Thomson valve in order to obtain maximum yield from cycle. Hence pressure drop across heat exchanger must have to minimum in order to deliver high pressure gas at valve inlet.

II. COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics abbreviated as CFD is predicting what will happen, quantitatively, when fluids flow, often with the complications of simultaneous flow of heat, mass transfer, phase change, chemical reaction,
mechanical movement, stresses in and displacement of immersed or surrounding solids.

Computational Fluid Dynamics, uses different numerical methods and a number of computerized algorithms in order to solve and analyze problems that involve the flow of fluids. It discretizes the spatial domain into small cells to form a volume mesh or grid, and then apply suitable algorithms to solve the equation of motion.

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modeling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Analysis starts with the defining of the boundary and initial conditions for the dominion and leads to modeling of the entire system. Finally, it is followed by the analysis of the results, conclusions and discussions.

A. Turbulence Modeling

Turbulence arises due to the instability in the flow. This happens when the viscous dampening of the velocity fluctuations is slower than the convective transport, i.e. the fluid element can rotate before it comes in contact with wall that stops the rotation. For high Reynold numbers the velocity fluctuations cannot be dampened by the viscous forces and the flow becomes turbulent.

Turbulent flows contain a wide range of length, velocity and time scales and solving all of them makes the costs of simulations large. Therefore, several turbulence models have been developed with different degrees of resolution. All turbulence models have made approximations simplifying the Navier-Stokes equations. There are several turbulence models available in CFD-software including the Large Eddy Simulation (LES) and Reynolds Average Navier-Stokes (RANS). There are several RANS models available depending on the characteristic of flow, e.g., Standard k-ε model, k-ε RNG model, Realizable k-ε, and RSM (Reynolds Stress Model) models. For current work RSM model is used.

B. Reynolds stress model

RSM closes the Reynolds-Averaged Navier-Stokes equations by solving additional transport equations for the six independent Reynolds stresses. It utilizes transport equations derived by Reynolds averaging the product of the momentum equations with a fluctuating property.

In present work fluctuation of velocity and thermodynamic properties must have to be taken care hence RSM is more suitable.

RSM is good for accurately predicting complex flows. It is generally used for streamline curvature, swirl, rotation and high strain rates. Physically most complete model i.e. history, transport, and anisotropy of turbulent stresses are all taken into account by this method.

C. Wall Functions

Wall functions are a set of empirical formulas which connects the different variables such as velocity, temperature and pressure at the wall to the near wall region (Turbulence boundary layer). Wall functions are applied by using the law of wall for the variables near the wall region. Then they formulate the turbulence variables such as turbulent kinetic energy and turbulent dissipation energy. These formulations depend upon the respective turbulence model.

There are following types of wall functions mostly used:

- Standard Wall Functions
- Non-Equilibrium Wall Functions
- Enhanced Wall Functions.

D. Flow Governing Equations

The flow is governed by the continuity equation, the energy equation and Navier-Stokes momentum equations. Transport of mass, energy and momentum occur through convective flow and diffusion of molecules and turbulent eddies. All equations are set up over a control volume where x, y, z = 1; 2; 3 correspond to the three dimensions.

1) Continuity Equation

The continuity equation describes the conservation of mass and is written as in equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_1}{\partial x_1} + \frac{\partial \rho U_2}{\partial x_2} + \frac{\partial \rho U_3}{\partial x_3} = 0$$

The continuity equation describes the conservation of mass and defines the rate of increase of mass in a control volume as equal to the difference in amount through passing between its inlet and outlet faces.

2) Momentum Equations (Navier-Stokes Equations)

The momentum balance, also known as the Navier-Stokes equations, follows Newton’s second law. The change in momentum in all directions equals the sum of forces acting in those directions. There are two different kinds of forces acting on a finite volume element, surface forces and body forces. Surface forces include pressure and viscous forces and body forces include gravity, centrifugal and electromagnetic forces.

The momentum equation in tensor notation for a Newtonian fluid can be written as in equation

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = \frac{1}{\rho} \frac{\partial P}{\partial x_j} + \nu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + g_i$$

The continuity equation is difficult to solve numerically. In CFD programs, the continuity equation is often combined with momentum equation to form Poisson equation. For constant density and viscosity the new equation can be written as below:

$$\frac{\partial}{\partial x_i} \left( \rho \frac{\partial P}{\partial x_i} \right) = -\frac{\partial}{\partial x_j} \left( \rho U_i U_j \right)$$

This equation has more suitable numerical properties and can be solved by proper iteration methods.

3) Conduction Heat Transfer

Thermal conduction is a mechanism of heat propagation from a region of higher temperature to a region of low temperature within a medium or between different mediums in direct physical contact.

$$\frac{a}{a_x} (k_x \frac{\partial T}{\partial x}) + \frac{a}{a_y} (k_y \frac{\partial T}{\partial y}) + \frac{a}{a_z} (k_z \frac{\partial T}{\partial z}) + q = \rho c \frac{\partial T}{\partial t}$$

4) Model for analysis

From literature work dimensions optimized through numerical techniques are taken for CFD analysis. The modelling was done in Next generation (NX) software.

- Helical tube is created using helix and wound around cylindrical shell. Fins are created using extrude...
command in plane lies perpendicular to helix curve. Fins are replicate throughout helical tube using pattern features.

- Outer and inner cylindrical shell created using shell command. Inlet and outlet are extended for imposing boundary condition carefully without error. Fins are united with helical tube and whole united part chosen as tool part for analysis.
- Fluid domain of hot fluid created by choosing inner tube diameter of tube. Cold fluid domain is created by subtracting helical tube part with fins from portion between cylindrical shells. Intersecting portion are clearly joined. Initially the full scale model of the heat exchanger was made. Due to computing limitations the model than reduced up to 5 turns of coil and respective shell height of 400mm.

**Fig. 2.1: Three dimension model of heat exchanger**

### E. Analysis

Analysis is done using thermal-coupled flow solver of NX software. Number of iterations to be carried out is kept as 500.

Time step is reduced to 0.0005 instead of default value specified by software in order to predict minute variations. Property of flowing fluid helium is clearly defined and some values are defined manually for low temperature range. Material properties for steel and copper are assigned from table defined by software.

1) **Meshing or Grid Generation**

Generated mesh contains tetrahedral elements cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries.

For mashing helical tubes 3-D tetrahedral are used. For simplification and accuracy model is divided into 4 main domain as follow:

1) Helical tube made of copper material.
2) Fluid domain of hot helium gas inside helical tube meshed with 3d elements.
3) Fluid domain of cold side helium gas through shell side.
4) Outer and inner cylindrical shell covering helical tube.

It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region, hence it meshed with fine elements of element size of 1 unit. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

**Fig. 2.2: Meshed model**

2) **Effect of Numerical Mesh Grid Refinement**

From previous literature it is confirmed that fine mesh always results in sensitivity of friction factor and heat transfer coefficient on curved section of pipes. The refinement of the numerical mesh grid has a small effect on values of the friction factor.

Initially results are plotted using coarser mesh which later on modified and made fine particularly near wall section.

3) **Boundary Conditions Imposed For Analysis**

<table>
<thead>
<tr>
<th>Sr.no.</th>
<th>Position</th>
<th>Hot fluid inlet</th>
<th>Hot fluid outlet</th>
<th>Cold fluid inlet</th>
<th>Cold fluid outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>Temperature</td>
<td>300 K</td>
<td>92 K</td>
<td>85 K</td>
<td>297 K</td>
</tr>
<tr>
<td>02</td>
<td>Pressure</td>
<td>14 bar</td>
<td>13.78 bar</td>
<td>1.013 bar</td>
<td>1.012 bar</td>
</tr>
<tr>
<td>03</td>
<td>Mass flow rate</td>
<td>0.003 kg/sec</td>
<td>0.003 kg/sec</td>
<td>0.003 kg/sec</td>
<td>0.003 kg/sec</td>
</tr>
</tbody>
</table>

Table 1:

### III. RESULTS AND DISCUSSION

In first step of analysis results are plotted for steady state analysis. Analysis is performed using thermal-flow coupled solver in NX Software. Results are plotted primarily for temperature and pressure drop for hot and cold fluid flowing across helical tube and shell side respectively.

A. **Temperature Distribution**

**Fig. 3.1: Hot fluid at inlet temperature**
Hot fluid inlet temperature is at 300 K as per input conditions assigned to it. Temperature at the end of helical tube decreases to 125 K with simultaneous increase in temperature of cold fluid to 265 K. With the observed temperature variation overall heat transfer coefficient between fluid can be roughly predicted around 74 W/m\(^2\)K.

Pressure drop of 0.6 bar takes place while hot fluid flow across helical tube. Pressure decreases gradually as fluid flows along helical tube.

Maximum velocity of hot fluid flowing through helical tube is found to be 3.5 m/s. Velocity across shell side ranges from 0.6 to 1.2 m/s.

### C. Effect of Mass Flow Variation on Pressure Drop

Results are plotted for pressure drop by varying mass flow rate values at inlet boundary conditions. Variations are calculated for 145 mean shell diameter, 900 fins/meter and 8.8 mm inner tube diameter.

<table>
<thead>
<tr>
<th>Sr.no.</th>
<th>Mass flow rate (g/sec)</th>
<th>ΔP tube side (At 14 bar inlet)</th>
<th>ΔP shell side (At 1.2 bar inlet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>2.5</td>
<td>0.2 bar</td>
<td>0.2 bar</td>
</tr>
<tr>
<td>02</td>
<td>3.5</td>
<td>0.6 bar</td>
<td>0.25 bar</td>
</tr>
<tr>
<td>03</td>
<td>4.5</td>
<td>0.8 bar</td>
<td>0.28 bar</td>
</tr>
</tbody>
</table>

Table 2:

### D. Effect of Tube Size Variation on Temperature Drop

In order to achieve higher heat transfer there must be maximum temperature drop of hot fluid against minimum rise in cold fluid temperature as inlet temperature of both fluids are fixed. Following results are obtained for temperature drop by taking alteration of helical tube size and all other parameters as it as by analyzing segmented portion of heat exchanger.

<table>
<thead>
<tr>
<th>Sr.no.</th>
<th>Tube diameter</th>
<th>ΔT hot fluid side</th>
<th>ΔT shell side cold fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>6.5mm</td>
<td>72</td>
<td>98</td>
</tr>
<tr>
<td>02</td>
<td>8.5mm</td>
<td>75</td>
<td>100</td>
</tr>
<tr>
<td>03</td>
<td>11.5mm</td>
<td>77</td>
<td>103</td>
</tr>
</tbody>
</table>

Table 3:

From above results it can be concluded heat transfer increase as tube diameter increase, but simultaneously it can also increase overall size of heat exchanger.

### E. Validation

Results for pressure drop are validate against previous literature in which pressure drop had been experimentally calculated for various Reynolds number tube flow. In the range of 27000 Re number pressure drop experimentally calculated is 160 mbar at 2 bar. CFD results shows it 155 mbar for full length model.

### IV. CONCLUSIONS

The heat transfer and flow distribution is discussed in detail and proposed model is compared with the experimental results as well. After predicting temperature and pressure drop of cold and hot fluid follow conclusion can be made by for optimization of dimensions and heat transfer phenomenon.

- Nusselt numbers which directly influence heat transfer at various points along the length of the pipe was estimated. Nusselt number on the outer side of the coil is found to be the highest among all other points at a specified cross-section, while that at the inner side of the coil is the lowest.
− Tube side pressure drop decreases drastically as the tube diameter increases, but shell side pressure drop and surface area required for the given heat duty increases.
− Inner heat transfer coefficient decreases as the tube diameter increases and hence reduces the overall heat transfer coefficient. Shell side heat transfer coefficient reduces due to increase in shell diameter of a heat exchanger and the requirement of the length of finned tube will be more in this case for given heat duty.
− Increase in mass flow rate results in higher pressure drop of hot fluid while it is not much pronounced for cold side flow, as hot inlet pressure is much more than cold inlet.

V. ACKNOWLEDGMENT

The authors are thankful to the Department of Science and Technology, Government of India, (No. SB/FTP/ETA-0014/2014) for funding the research project under which present work is carried out.

REFERENCES
