CFD Simulation of Convective Heat Transfer in Actively Heated Tube Bundle

Pawar Karan Hiralal[1] Dr. Pravin V Honguntikar[2]
[1]M.Tech Student [2]Professor & Head of Depatment
[1,2]Department of Mechanical Engineering
[1,2]PDACEG, Kalaburagi, India

Abstract— Active heating is used as a flow assurance technique to prevent the huge temperature drop of crude oil while transporting from wells to facilities, which creates a friendly environment for hydrate formation and solid depositions. In the present study, convective heat transfer in tube bundle with three internal tubes was investigated through computational fluid dynamics simulation. A three dimensional model for existing bundle was developed. The model described the heat transfer occurring inside the bundle, aiming to heat up or at least maintain the production fluid temperature. The heat transfer from heating pipe to the whole bundle can be visualized through the temperature contours and the temperature gradient at the outlets. The effect of heating medium temperature and flow velocity are studied in the developed model.

Keywords: Active Heating; Tube Bundle; Computational Fluid Dynamics

I. INTRODUCTION

In offshore oil and gas production systems, the hydrocarbon emerges from the reservoir at a relatively high temperature and must then be passed up the well, through the flow lines to a production riser base from whence it flows to the processing facility (a platform or a Floating Production Storage Offloading-FPSO). As the flow proceeds from the well to the processing facility, the temperature of the hydrocarbon decreases due to the heat losses to the surroundings (the rock surrounding the well, the sea water surrounding the flow lines and riser and, finally, the atmosphere).

As the exploration and development of oil and gas fields in deep water environments continues to grow, more complex multiphase hydrocarbons are produced and transported for long distances to the processing facility. These include the Gulf of Mexico (GOM), offshore Brazil, offshore West Africa, the North Sea and the North Atlantic Regions. Based on the “Deepwater Gulf of Mexico 2009: Interim Report of 2008 Highlights” published by the U.S. Department of the Interior, Minerals Management Service, 57 percent of all GOM leases were located in deepwater at the end of 2008. There were 141 producing projects in the deepwater GOM at the end of 2008, up from 130 at the end of 2007.

Unfortunately, cooling of the production fluid during this flow process can often lead to the formation of solid phases known as Waxes, Hydrates, Asphaltenes and Scales, which may deposit on the surface of the pipelines or risers giving rise to a restriction in the production and, in the worst case, total blockage of the flow system.

A. Waxes

Waxes are high molecular weight, highly saturated organic substances, which are slightly soluble in the liquid phase of black oils and condensates. Wax in crude oil is primarily composed of paraffinic hydrocarbons usually between $C_{18}H_{38}$ and $C_{70}H_{142}$ (Sadeghazad, 2000). Solid wax can form at temperatures lower than 60°C (140°F). Wax deposits have been found on surfaces such as those of the well-bore and the flowline and export line. Wax deposition starts with the deposition of small wax particles followed by a partial restriction in the flow. As shown in Figure 1-1 below; this may ultimately result in complete blockage of the pipeline.

Fig. 1.1: Wax deposition (Kaczmarski and Lorimer, 2001)

Wax deposits restrict flow and reduce throughput onshore as well as offshore. Offshore operations in deeper water encounter cold temperatures, especially in the case of subsea flowlines and tiebacks. Temporary shutdowns are a concern in deepwater operations with paraffinic crude because of the chance that the oil will gel in a subsea flowline or pipeline. The problems with offshore systems are the increased remediation costs and the increased financial impact of flow restrictions in offshore production, where wells typically have higher production rates than those of onshore wells. The restart of a plugged subsea pipeline may be operationally or economically impossible.

B. Hydrates

Hydrates are ice-like crystals that consist of water and light hydrocarbons or other gas molecules (e.g. methane, ethane, propane, nitrogen, carbon dioxide, hydrogen sulphide, etc.). Hydrates contain about 85 mol% water and 15 mol% hydrocarbon at low temperature (0°C - 40°C) and high pressure conditions.

Hydrates are a great concern to the industry especially in deepwater oil developments since ambient temperatures are approximately 39°F (4°C), which is well within the hydrate formation region at typical operating pressures. Hydrates can form during normal operation but transient operations are often more vulnerable. Hydrates can also form in the wellbore, in the flowline, at valves and at meter discharges. Hydrates have been found in wet gases, condensates and black oils. Hydrate is a flow assurance problem and is mostly encountered during restart. Hydrates can cause pipeline blockage (as illustrated in Figure 1-2) and it can take up to weeks and months to dissociate the hydrate plugs. In the worst case scenario, the hydrate plug may cause the pipeline rupture.

C. Asphaltenes

Asphaltenes are a class of polymeric aromatic compounds in crude oil that are not soluble in n-heptane. Asphaltenes carry the bulk of the inorganic component of crude oil, including sulphur and nitrogen, and metals such as nickel and vanadium. All oils contain a certain amount of asphaltenes. Asphaltenes only become a problem during production when they are unstable.

Asphaltene stability is a function of the ratio of asphaltenes to stabilizing factors in the crude such as aromatics and resins. The factor having the biggest impact on asphaltene stability is pressure. Asphaltenes may also be destabilized by the addition of acid or certain types of completion fluids and by the high temperatures seen in the crude oil refining process. In general asphaltenes cause few operational problems since the majority of asphaltenic crude oils have stable asphaltenes. Typically problems only occur downstream due to blending or heating (as in the preheat train of a crude distillation unit). Crude oils with unstable asphaltenes suffer from some severe operational problems, most of which are fouling related and affect valves, chokes, filters, and tubing.

The properties and characteristics of waxes, hydrates, asphaltenes and scales are explained in detail in Section 2.1. In order to prevent formation of these solids, various techniques, being employed in the industry, include improved equipment design and selection, operational methodologies and chemical treatments or some combination of these methods. Some of the techniques are briefly summarised here, however, it should be noted that further details can be found in Section 2.2.

The methods for dealing with the problems of solids handling in hydrocarbon production may be classified as follows:

- Thermodynamic Inhibition (methanol, glycol, etc)
- Thermal Insulation
- Active Heating
- Mechanical Methods

The most important types of method are those aimed at preventing heat loss from the produced hydrocarbon. However, each of the methods has drawbacks due to either its working mechanism or to the high cost associated with the technology.

An example of such drawbacks is found in the case of hydrate formation; hydrate formation has been a continuing concern in onshore and offshore shallow water gas production flowlines requiring the operator to implement methanol or glycol injection, dehydration, or the use of line heaters. As operations moved further offshore to deepwater and ultra-deepwater, the issues of distance and accessibility compound the problem. The deepwater environment is cold (approximately 40°F or 4.4°C) and the pipeline operating pressure is higher due to a longer production riser requiring high hydrostatic pressure; the low local temperatures can cause flow problems in the riser and export pipeline since heat losses from the production tubing may lead to the formation and deposition of gas hydrate, paraffin and asphaltene materials. (Wang et al, 2006) Kaczmarski et al (2001) discusses other drivers for solid deposition in deepwater production, one of which is the much greater variability in fluid properties in deepwater reservoirs than in shallow water reservoirs. Fluids produced in deepwater reservoirs are subject to a greater shift in conditions of temperature and pressure as they travel from the reservoir to the processing facility. In addition, the nature of the produced hydrocarbons and the associated production problems in deepwater oil and gas reservoirs are less well known in general. Furthermoredeepwater oil and gas productions are very costly and difficult to access during repair and remediation efforts.

Kaczmarski et al (2001) quoted that a paraffin cutting job, which is to remove paraffin deposited inside a pipeline on a shallow water platform-well may cost $10,000-$20,000 with an associated production loss of maximum one day to perform the work. As for a Deepwater subsea well, the same operation may cost $1,000,000 or more since a vessel will have to be brought on location to perform the work. Hence the avoidance of temperatures at which the precipitation and deposition of hydrocarbon solids occur is very important for all oil and gas production systems.

Active heating techniques combined with thermal insulation are becoming increasingly popular, especially in Deepwater production systems, due to their greater adaptability in shutdown and start-up operations as well as their greater operational flexibility. The active heating method can be used to warm-up (or prevent the cooling down) of a non flowing system (for example during shutdowns) and to maintain or increase the temperature of the flowing product and reduce chemical injection requirements. Active heating is also capable of efficiently maintaining the temperature of hydrocarbons above the wax and hydrates formation temperature over long distances compared to a passively insulated pipeline.

Generally, active heating system can be categorized into two classes based on the means of heating, namely hot fluid circulation heating and electrical heating. The active heating system using hot fluid circulation requires a bundled pipeline in which consists of several pipes used for production fluid, gas or water injection lines, gas lift, heating fluid lines, control signal and electrical cables etc.

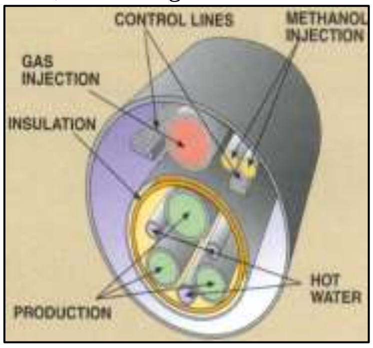


Fig. 1.3: Gullfaks South flowline bundle (Knudsen, 1999)

Figure 1.3 shows the flowline bundle installed at the Gullfaks South satellite platform, which includes two 6 in. and one 8 in. production flowlines a gas injection line, a line for heated water and two for water return, all of which are contained in a common insulated sleeve pipe inside the bundle. Figure 1-4 shows some of the different types of bundle configuration used in the industry. The bundle configuration depends on the type of produced fluid, and on the hydrate and wax mitigation philosophy. The space between the pipes in the bundle is usually filled with either water or nitrogen gas depending on the type of application being used for.

Fig. 1.4: Different types of bundle configuration (Subsea7, UK)

A bundle is claimed to provide an effective means of thermal management of flowlines using low-cost materials especially for Deepwater production systems. However, it is important to estimate the heat transfer performance correctly to avoid expensive over-heating and to prevent serious operational problems which may arise from failure to maintain an adequately high product temperature. Rigorous analyses of the heat transfer in bundled flowline systems has not been described in the published literature nor has such an analysis been offered by the vendors of such systems. The information available in the open literature for the heat transfer analysis of bundled pipeline systems is limited and is confined to the specific bundle configurations. Furthermore, these analysis methods are not being properly validated against the experiments.

The heat transfer mechanism taking place in these kinds of bundle systems are mainly of convection heat transfer and it is very difficult to accurately predict heat transfer behaviour quantitatively. It is therefore important to fully understand the heat transfer mechanisms inside a bundle pipeline and to develop adequate prediction methods for quantitative prediction of bundle thermal behaviour.

II. LITERATURE REVIEW

Sadeghipour and Asheghi (1994) carried out experiments on two to eight horizontal isothermal cylinders in vertical arrays to study steady state free convection heat transfer at low Rayleigh numbers. The authors aimed to establish the average Nusselt number for the individual cylinders and arrays as a function of Rayleigh number and separation distance between the cylinders.

Corcione (2005) studied free convection heat transfer from horizontal isothermal cylinders set in a vertical array by developing a specially developed computer code based on the SIMPLEC algorithm. In his numerical study, simulations are performed for the arrays of 2 to 6 circular cylinders with the spacing between them ranging from 2 to 50 cylinder diameters. The corresponding Rayleigh number ranges between 5×10^2 and 5×10^5 in air.

Ashjaee and Yousefi (2007) studied laminar free convection heat transfer from five aluminium horizontal isothermal cylinders of 10mm diameter arranged in vertical and inclined arrays by carrying out experiments using a Mach-Zehnder interferometer. This experimental study covered the vertical cylinder vertical spacing range from 2 to 5 cylinder diameters (centre-to-centre) for both vertical and inclined arrays and the horizontal spacing varied from 0 to 2 cylinder diameters for the inclined array. The Rayleigh number based on the cylinder diameter ranged between 1×10^3 and 3×10^3 in air.

Reymond et al (2008) report an experimental investigation of natural convection heat transfer from a single horizontal cylinder and a pair of vertically aligned horizontal cylinders. The cylinders were 30mm in diameter and were immersed in water; the inter-cylinder spacings studied were 1.5, 2 and 3 diameters. Their experimental study considered Rayleigh numbers of 2×10^6, 4×10^6 and 6×10^6, with the following three heating configurations:

- Lower cylinder heated with upper cylinder heated
- Lower cylinder heated with upper cylinder unheated
- Upper cylinder heated with lower cylinder unheated

From their experiments with the three spacings mentioned above, the heat transfer from the lower cylinder is unaffected by an unheated upper cylinder and similarly for the upper cylinder with an unheated lower cylinder. Therefore, their attentions were given to the case of heat transfer from the upper cylinder with both cylinders heated. For all three different spacings, the heat transfer distributions are broadly similar for each spacing, but the relative magnitude of the surface heat transfer is strongly dependent on the spacing between cylinders. The authors agreed with Sparrow and Niethammer (1981) that the optimum spacing is Rayleigh number dependent and the spacing of 2 diameters appears to be the optimum spacing for enhanced heat transfer at the bottom of the upper cylinder.

Zhang et al. (2002) introduced another thermal analysis method developed for both direct and indirect heated bundle designs as shown in Figure 2-12; this model facilitates the determination of the transient heat transfer between the heating water and the heated fluids in the presence of complex insulation configurations. In their analysis, heating is only used during shutdown and start-up and the authors studied warm-up time, pressure requirement, flowrate requirement and hot water temperature requirement. The authors claimed that currently available commercial pipeline simulation software packages do not have the accuracy needed to determine heat transfer in complex bundle configurations.

Chin et al (2000a) investigated the thermal interaction among the flowlines and its effect on the overall thermal performance of an insulated multiple flowline bundle; the bundle had multiphase flow in the production line and single phase flow in the two heating lines. A computer model was developed to simulate multiphase flow in a multiple flowline bundle, which was solved by using the “Bundle” package of the OLGA simulator. Chin et al (2000a) noted that the “Bundle” module in OLGA is a forced convection model, which was originally designed for the presence of the carrier fluid instead of insulation around the flowlines. It has difficulty in modelling multiple pipelines encased by insulation and hence Chin et al (2000a) modified the "Bundle" module of the OLGA simulator with a modified conduction model. From their steady state simulation study, the authors compared the numerical results against the actual production data and concluded that the model for steady state is accurate. From their steady state analysis, the authors discovered that there are thermal interactions among the pipelines inside the multiple flowline bundle.

Duret et al. (2000) made attempts to take into account models for bundles in an overall transient multiphase flow model (TACITE). The TACITE compositional code is able to simulate the behaviour of transient multiphase flow for the design and the control of oil and gas production pipelines and well. Duret et al developed a transient pipeline bundle module to analyze the thermal interactions between several single phase lines and the main production line.

A simplified bundle geometry is shown in Figure 2-1.

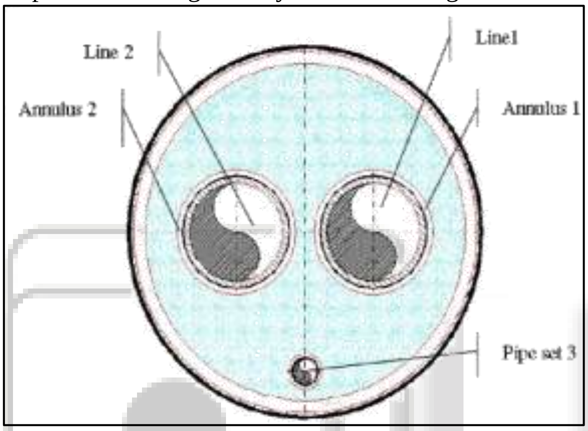


Fig. 2.1: Simplified bundle geometry validation (Duret et al, 2000)

Fig. 2.13: Integrated Production Bundle (IPB) used by Secher and Henry (2002)

Khlefa et al (2003) described the flow assurance challenges faced during the development of a multi-well field located approximately 27 km from the host facility and how those challenges have been solved using PIP active heating. The King Field development consists of a subsea production system tied back to the Marlin TLP. Marlin is located in Viosca Knoll Block 915 at a water depth of 990 m. The subsea wells are tied back to the Marlin TLP using two flowlines. Flow assurance was one of the key factors in the development of the field with hydrate and wax deposition as the primary design parameters. Several development options were considered including insulation, chemical mitigation and active heating. Active heating was selected as the preferable option for its cost effectiveness, avoidance of pigging for wax control, and ability to maintain the flowlines above the hydrate formation and wax deposition temperatures during shut downs and prior to start-ups after prolonged shut downs.

Fouad et al (2004) evaluated insulation and active heating options for transportation of waxy crude oil in an 8-inch flowline from a platform ten miles offshore Africa. Two bundled pipeline configurations are studied and the results are compared with conventional PIP and a non-jacketed insulated pipe. The authors concluded that the non-jacketed pipe option is not thermally efficient; the PIP option can keep the crude oil warm during normal flow, although high storage capacity is required at the platform which makes the solution impractical. The bundled pipeline configurations maintain the crude oil temperature above the WAT thus avoiding waxing and gelling during both steady state flow and restart operation.

III. METHODOLOGY

A. Computational Fluid Dynamics

CFD is the science of computer based simulation which defines fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena into a set of mathematical equations and solves these mathematical equations numerically by using computer programmes. The result of CFD analyses is relevant engineering data used in conceptual studies of new designs, detailed product development, troubleshooting, testing and experimentation of the product and equipment. The CFD analyses reduce the total effort required in the laboratory. The application of CFD started with the aerospace, automotive and nuclear industries (Bakker et al, 2001). With the advances in computing technology and the availability of fast inexpensive computing facilities, the application of CFD have propelled into industries including but not limited to process, petroleum, power, metallurgical, biomedical, pharmaceutical and food industries.

The applications of CFD are numerous and some examples of the application in the petroleum and process industry include, flow and heat transfer in process equipments such as boilers, heat exchangers, furnace, combustion equipment, pumps, blowers, piping, etc. CFD has also been applied to the study of aerodynamics of ground vehicles, aircraft, missiles for aerospace industry, film coating and thermoforming in material processing applications, flow and heat transfer in propulsion and power generation systems, ventilation, heating, and cooling flows in buildings and many more applications.

B. Advantages of CFD

Together with the advancement of computer technology, CFD enjoys a number of advantages namely cost, speed, ability to simulate real and ideal conditions. Using physical experiments and tests to get essential engineering data for design can be expensive and hence CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful.

In many of the engineering problems, CFD simulations can be executed in a short period of time compared to designing and setting up of the physical experiments. Therefore, engineering data can be introduced early in the design process. Experiments only permit data to be extracted at a limited number of locations in the system (e.g. pressure and temperature probes, heat flux gauges, LDV, etc.). CFD allows the analyst to examine a large number of locations in the region of interest, and yields a comprehensive set of flow parameters for examination.

One of the major advantages of CFD is its ability to theoretically simulate any physical and real conditions in the computer environment such as hypersonic flow which normally cannot be easily tested. Furthermore, CFD also allows great control over the physical process and provides the ability to isolate specific phenomena for study. For example, a heat transfer process can be idealized with adiabatic, constant heat flux, or constant temperature boundaries.

1) Limitations of CFD

Though CFD has numerous advantages, it has some limitations. CFD solutions rely upon physical models of real world processes such as turbulence, chemistry of the fluid and multiphase flow. Therefore, CFD solutions can only be as accurate as the physical models on which they are based. As with physical models, the accuracy of the CFD solution is only as good as the initial and boundary conditions provided to the numerical model.

As with solving equations on a computer, CFD invariably introduces numerical errors in the forms of round-off error and truncation error. Round-off errors will always exist even though they can be small in most cases due to finite word size available on the computer. Truncation errors will go to zero as the grid is refined due to approximations in the numerical models.

2) CFD Analysis Procedures

The basis principles and governing equations of CFD can be referred to Mohamad Taufik (2012) and other workers related to CFD. In short, the fluid domain is discretized into a finite set of control volume or cells, on which general conservation equations for mass, momentum, energy and species are, solved numerically using computer programmes. A typical CFD analysis procedure is shown in Figure 2-17 and discussed briefly in this section.

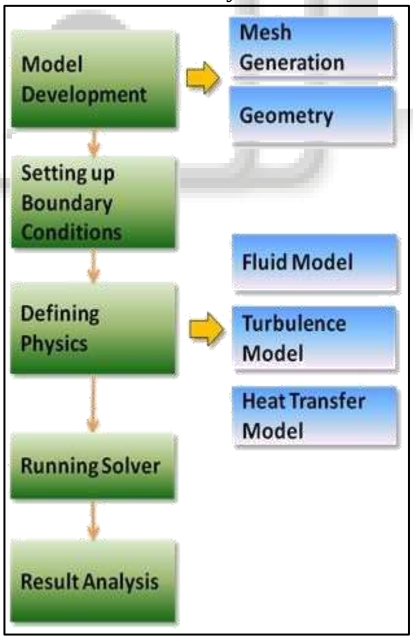


Fig. 3.1: A flowchart of the CFD analysis procedure

Firstly, it is very important to understand the problem being to be simulated using CFD because the problem will be built in the computer environment and require the data or information including geometry details, fluid properties such as density, single phase or multiphase, flow specifications, boundary and initial conditions. After completing data collection and verification for the problem, the geometry development shall be started for the given problem.

C. *Geometry Development and Mesh Generation*

The geometry of the flow domain can be constructed using any Computer Aided Drawing (CAD) software such as AutoCAD, SolidWorks, Pro/Engineer or the geometry development software provided by the CFD provider.

After completing the geometry of the fluid domain, it has to be divided into very small discrete cells, in which the flow variables will be calculated. This part is normally termed as mesh generation, and the number of mesh for a given fluid domain is important for the accuracy of the results. A finer mesh will generate better result compared to a coarse mesh but requires more computational power time. Therefore, for an optimum mesh quality, it is necessary to carry out mesh sensitivity analysis by carrying out simulations with different mesh sizes until results were no more dependent on the mesh size.

Figure 2-18 shows some of the generic term used in simple 2D and 3D mesh generation. Depending on the CFD provider, different types of mesh are available such as triangle, quadrilateral, tetrahedron, hexahedron, polyhedron, pyramid and prism.

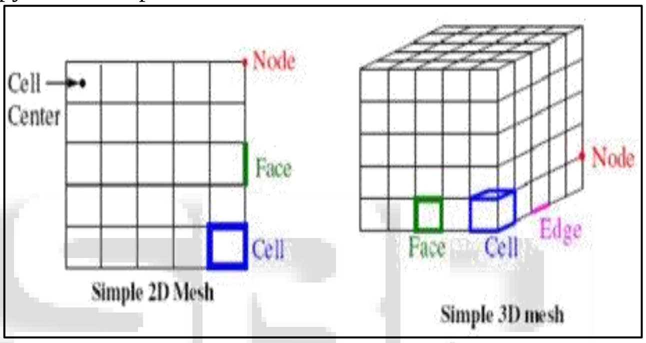


Fig. 3.2: Mesh components (ANSYS User Guide)

D. *Defining Physics of the Model*

The next stage after completing mesh generation is to define physics of the model. In this stage, the meshed fluid domain will be assigned with boundary conditions, which are a set of properties or conditions on the surfaces of a domain that are required to fully define the flow simulation. In general, there are four types of boundary as shown in Figure 2-19.

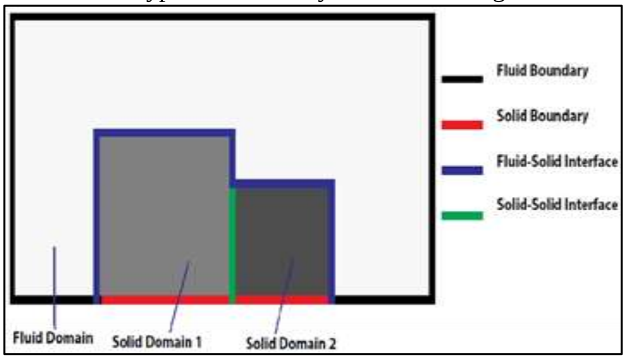


Fig. 3.3: Different types of boundary conditions (ANSYS User Guide)

For example, in ANSYS CFX, the fluid domain can be further divided into inlet, outlet, opening and wall as illustrated in Figure 2-20.

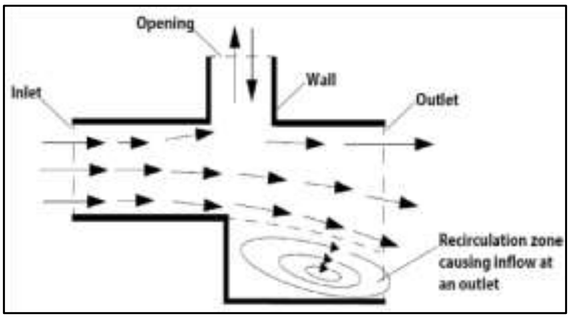


Fig. 3.4: Different types of fluid domain (ANSYS User Guide)

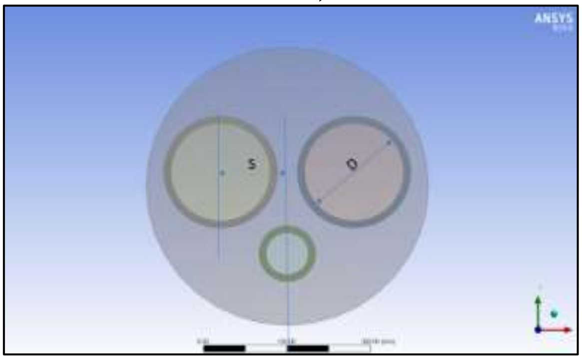


Fig. 3.5: 3 Pipe Bundle with S/D =0.58

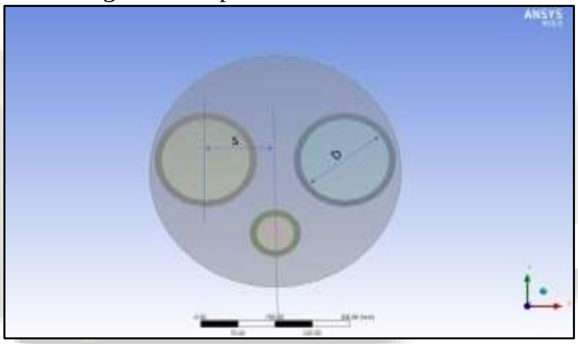


Fig. 3.6: 3 Pipe Bundle with S/D =0.63

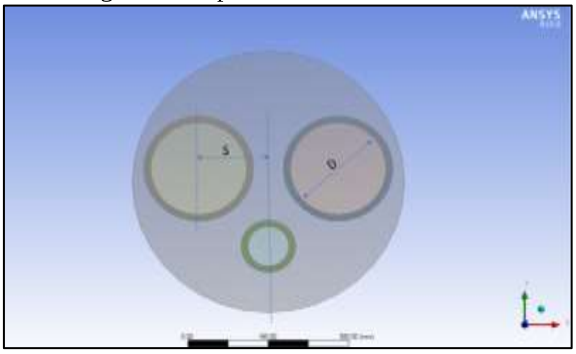


Fig. 3.7: 3 Pipe Bundle with S/D =0.68

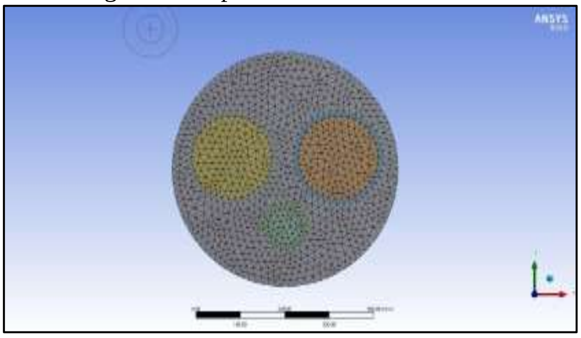


Fig. 3.8: Mesh Generated for 3-Pipe Bundle

The properties of the domain boundary either fluid or solid are defined in this stage. For a fluid domain, properties such as viscosity and thermal conductivity as well as fluid models which define the various aspects of the flow, such as turbulence, mass and heat transfer, and multiphase models. Depending on the nature of the problem, the other necessary phenomena such as chemical reactions, particle tracking, erosion rate modelling and heat transfer models are set up. An initial guess of the solutions are provided for a steady state simulation and an initial state of conditions for a transient simulations.

1) Calculating the Numerical Solutions

After specifying all relevant information required for the simulations, the CFD numerical solver will perform iterative calculations based on the solver control parameters required to control the behaviour of the numerical solution process. These parameters are referred in the ANSYS CFX as the time step (the maximum number of iterations to be performed) and convergence control (the normalised residual error in the final iteration).

2) Results Analysis

The results analysis part is very useful part of CFD, where one can do the analysis using various types of plots, some of which are X-Y charts, contour plots, streamlines, particle tracking, 2D and 3D animations. If the solution is not satisfactory to determine the required flow data, possible source of error needs to be identified and re-run the analysis until the result become satisfactory.

3) Commercial CFD Software

With the rapid development in technology and computation power, several commercial CFD software are available with ever more sophisticated flow modelling capabilities which can successfully solve the physics and complex nature of many industrial processes and applications. Some of the general-purpose commercial CFD software available are as follows;

- CFX by ANSYS Inc.
- FLUENT by ANSYS Inc.
- STAR-CCM+ and STAR-CD by CD-adapco
- CFdesign by Autodesk
- Fluidyn
- TransAT by ASCOMP
- FLOW-3D by Flow Science Inc.
- CFDExpert by Zeus Numerix

All the above mentioned software packages typically consist of three components namely PreProcessor, Solver and Post-Processor. The software package used in the current research work is ANSYS CFX, which consists of CFX-Pre for flow specifications, CFX-Solver for solving the numerical equations and CFX-Post for processing and analysing of the results. ICEM CFD was used for geometry development and mesh generation purposes. The main reason for the choice of ANSYS CFX was the availability of the code at Imperial College London.

ANSYS CFX, developed by ANSYS Inc., has been one of the leading CFD code in the industry, designed to fit easily within an integrated Computer-Aided Engineering (CAE) environment. The solver provides one of the most effective numerical methodologies available in industrial CFD code with the high level of accuracy required for complex unstructured meshes. The code's adaptive central bounded numeric scheme locally adjusts the discretization to be as close to second order as possible while ensuring stable simulation.

For turbulence modeling, the code provides numerous common two-equation models and Reynolds-stress models as well as shear stress transport (SST) model. In addition, the software provides large and detached-eddy simulation (LES and DES) models, as well as scaleadaptive simulation (SAS) model. The software provides complete mesh flexibility and includes mesh types such as triangular, quadrilateral, tetrahedral, hexahedral, pyramid and prism.

E. Flow Circuit

The basic principle underlying the design of the experiment is maintaining each of the surfaces (production pipe, test pipe, heat flow pipe, heat return pipe and sleeve) at nearly constant temperature. This was achieved by flowing water stream at the required temperature through the inner pipes and through the coil around the outside of the sleeve pipe.

The heat loss or gain through the respective surfaces is calculated by measuring the small temperature difference between the inlet and outlet of each water stream. Of course, from the heat balance, the amount of heat received by the cold water must equal the amount of heat released by the hot water and confirming this is a good check on the accuracy of the measurement. Heat transfer in the air-filled space between the simulated production pipe, test pipe, the heating pipes and the sleeve pipe is by natural convection and radiation. The following schematic diagram shows the flow circuit used in the experiments.

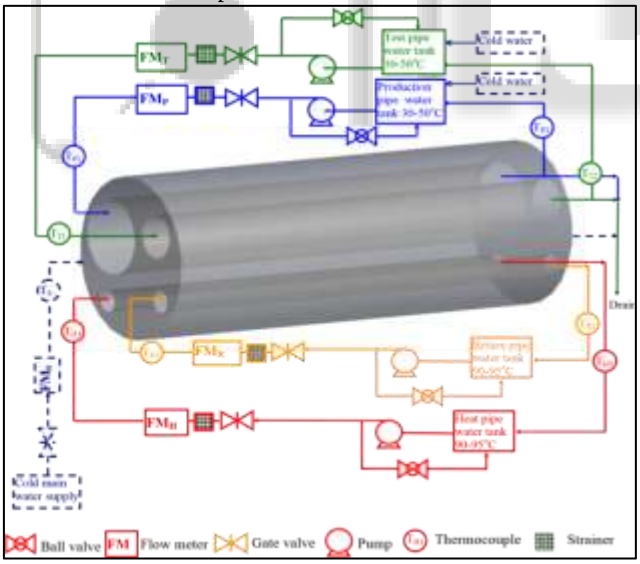


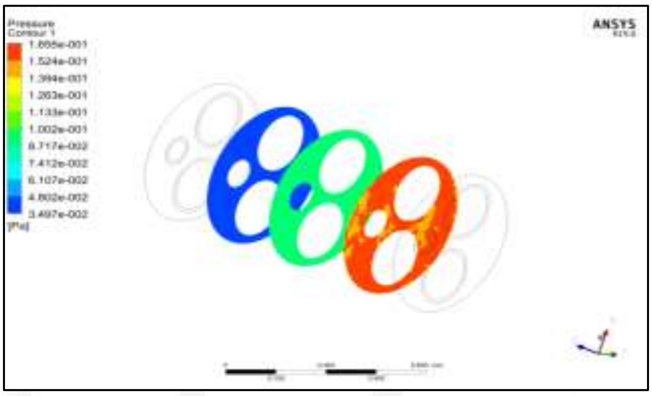
Fig. 3.9: Diagram of pipe bundle system

IV. RESULTS AND DISCUSSION

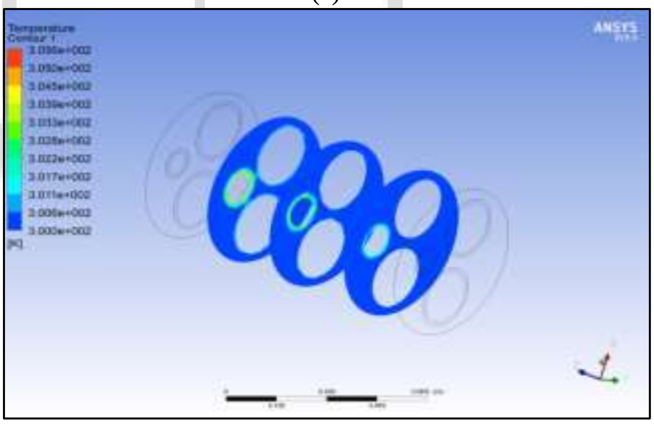
CFD simulations predict the heat transfer inside the tube bundle and the outlet temperature of all the streams were obtained. Convective heat transfer takes place where the hot water in heating pipe loses heat to the annulus fluid and further heat or at least maintain the production fluid temperature. Figure 4 shows the temperature contours for the annulus in between the tubes in 4 planes starting from inlets till the end of the bundle. Results were used to calculate the overall heat transfer rate inside the tubes to further investigate the effect of the heating medium temperature and flow velocity on the total heat gained by the production pipe.

Following Cases are are simulated to investigate the heat transfer effect in 3 Bundle Pipes

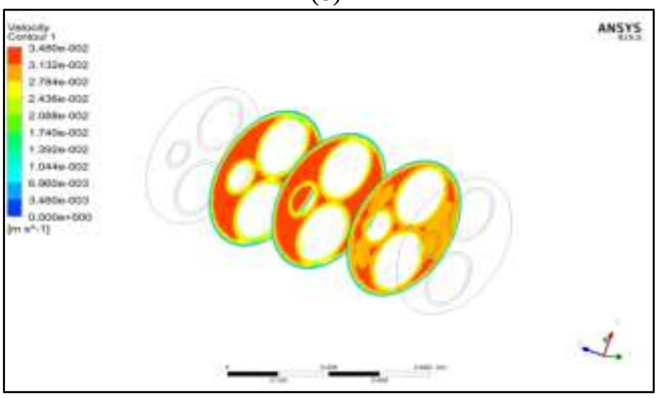
1) Thermal Analysis of 3 Bundle Pipe with S/D ratio of 0.58 with Heating medium Velocity from 0.03 m/s to 0.07 m/s
2) Thermal Analysis of 3 Bundle Pipe with S/D ratio of 0.63 with Heating medium Velocity from 0.03 m/s to 0.07 m/s
3) Thermal Analysis of 3 Bundle Pipe with S/D ratio of 0.68 with Heating medium Velocity from 0.03 m/s to 0.07 m/s



(a)

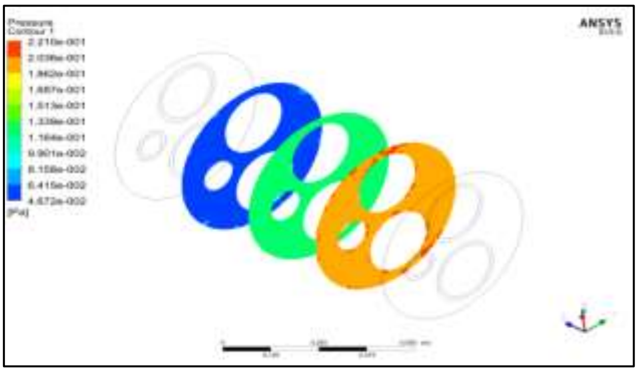


(b)

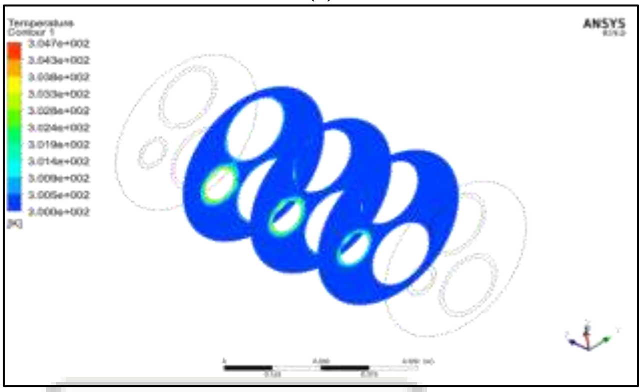


(c)

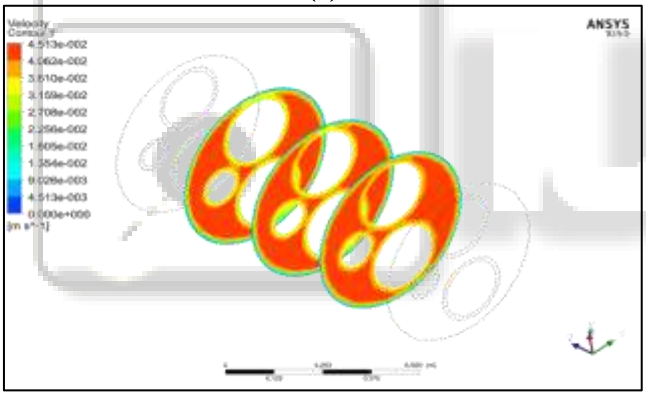
Fig. 4.1: Flow Contours with Heating Medium Velocity of 0.03m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes



(a)

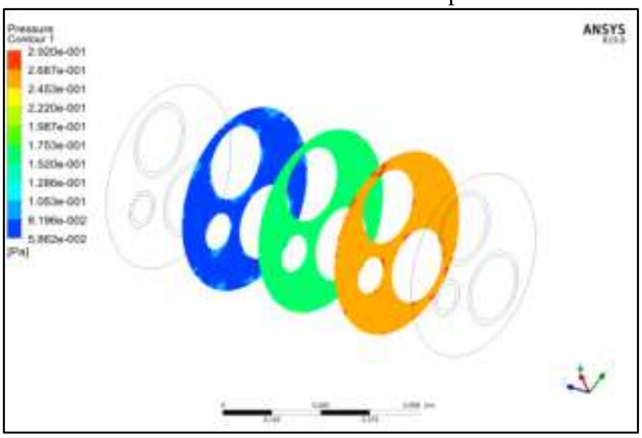


(b)

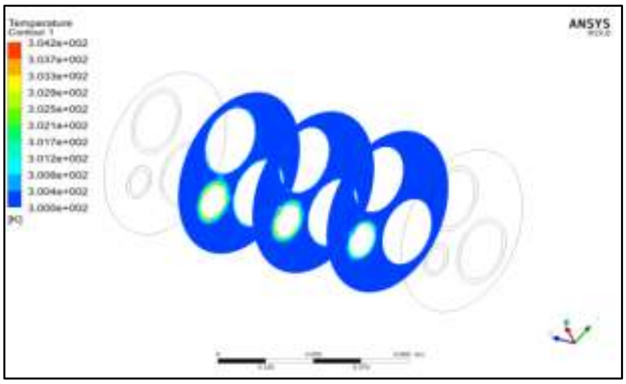


(c)

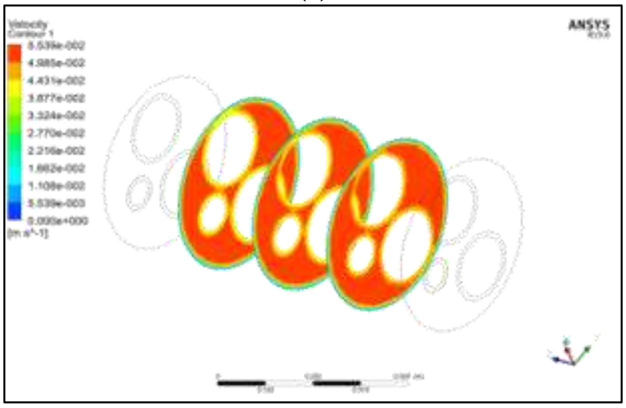
Fig. 4.2: Flow Contours with Heating Medium Velocity of 0.04m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes



(a)

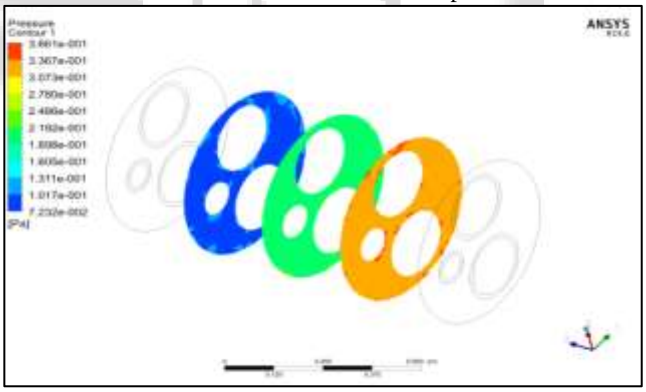


(b)

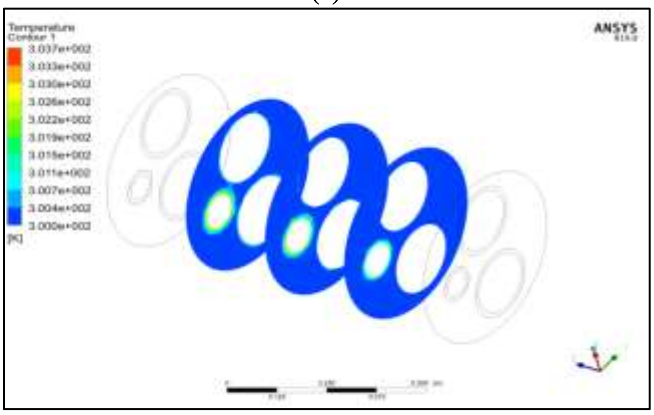


(c)

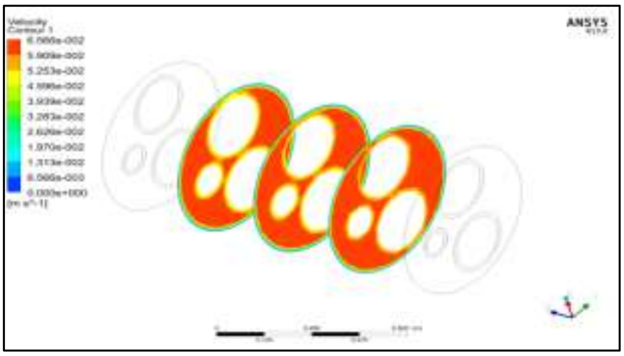
Fig. 4.3: Flow Contours with Heating Medium Velocity of 0.05m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes



(a)

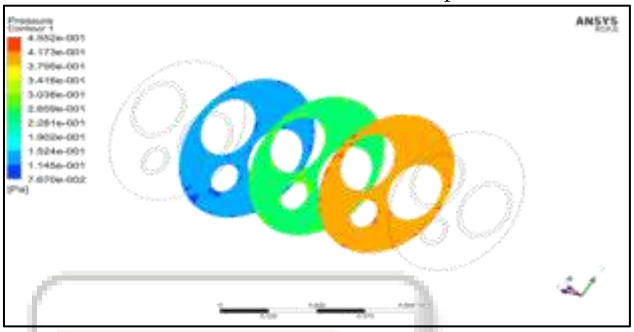


(b)

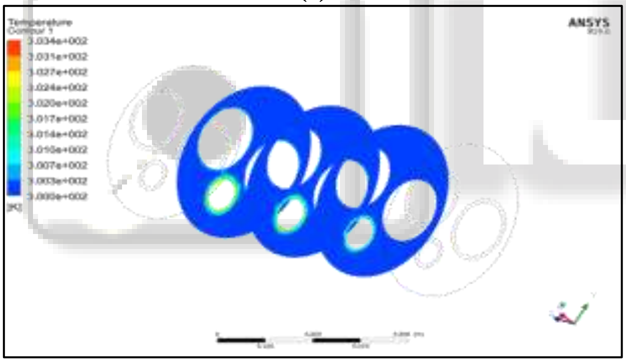


(c)

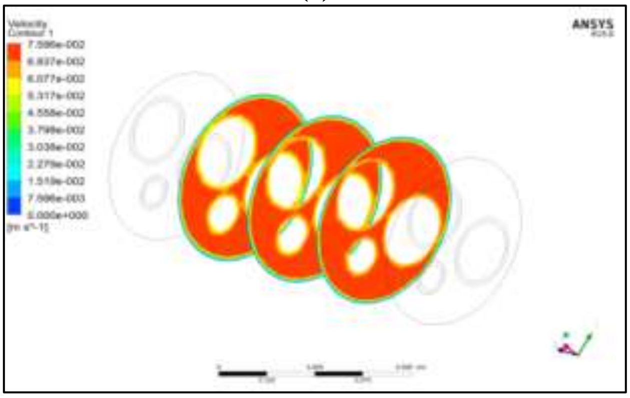
Fig. 4.4: Flow Contours with Heating Medium Velocity of 0.06m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes



(a)

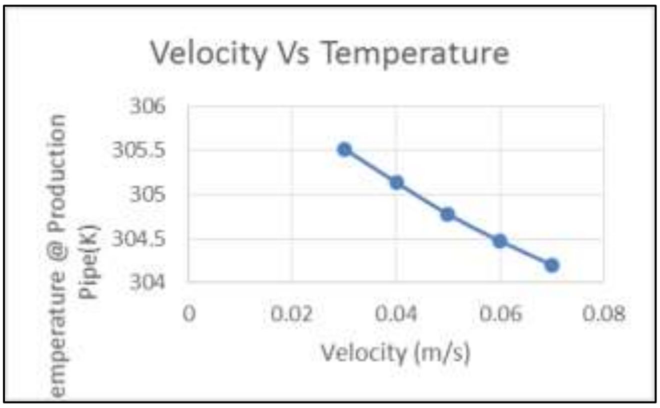


(b)

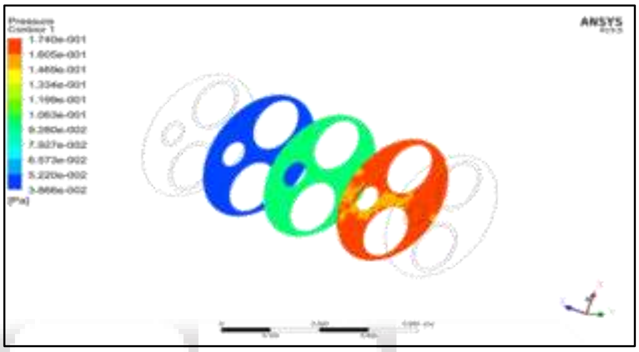


(c)

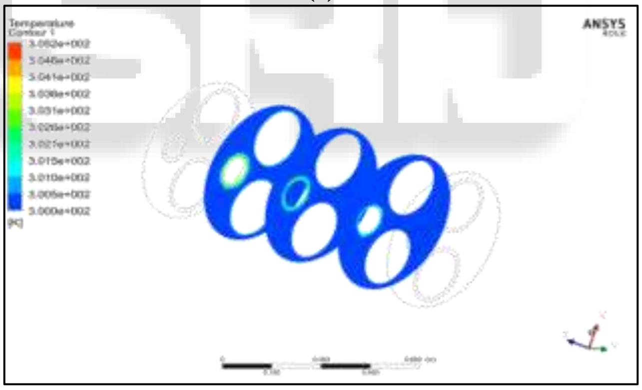
Fig. 4.5: Flow Contours with Heating Medium Velocity of 0.07m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes



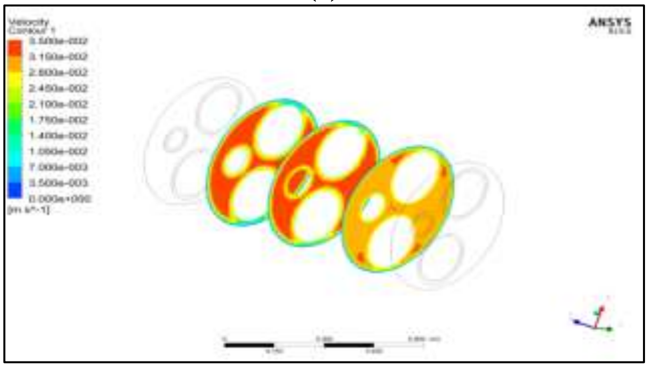
Plot 1: Velocity Vs Temperature of the Production Pipe Case 2 Thermal Analysis of 3 Bundle Pipe with S/D ratio of 0.63 with Heating medium Velocity from 0.03 m/s to 0.07



(a)

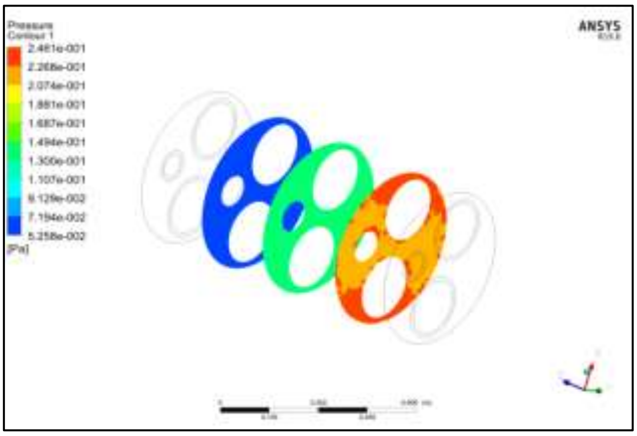


(b)

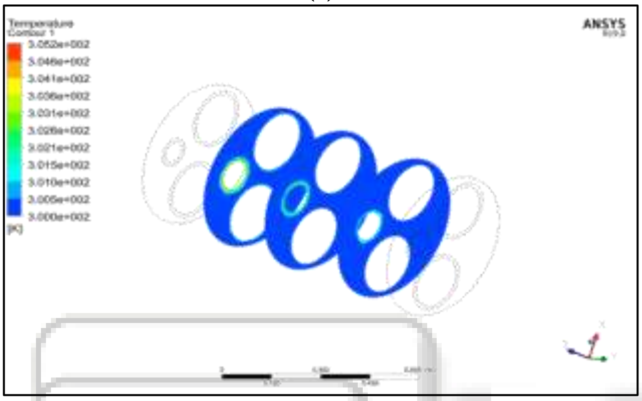


(c)

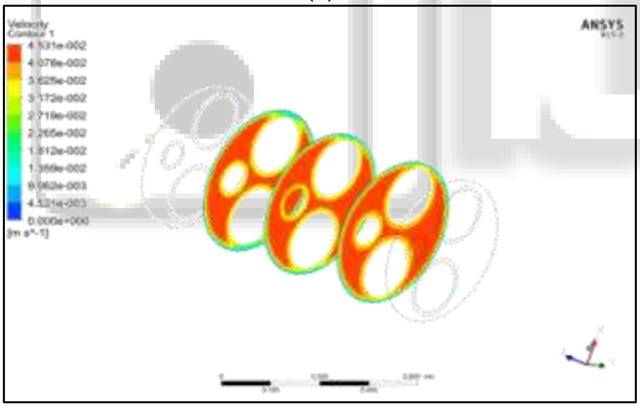
Fig. 4.6: Flow Contours with Heating Medium Velocity of 0.03m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.63.



(a)

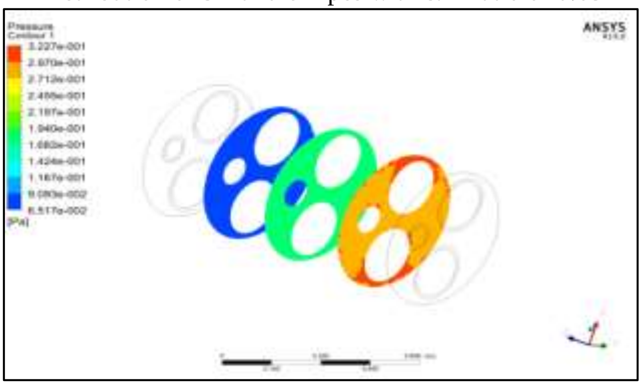


(b)

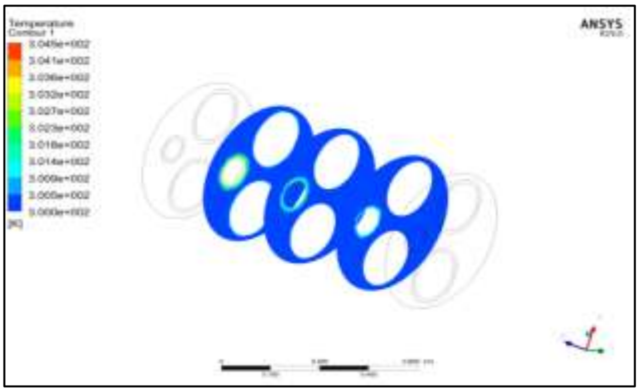


(c)

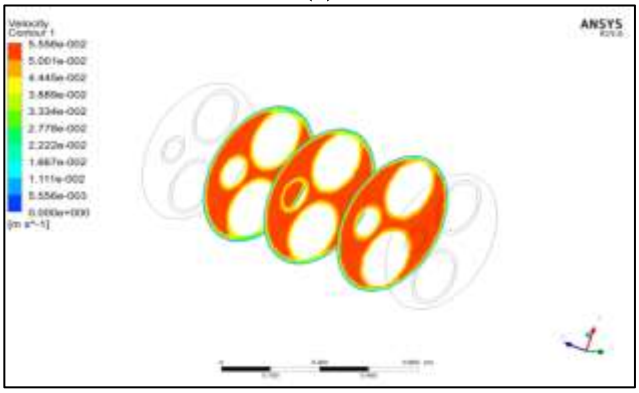
Fig. 4.7: Flow Contours with Heating Medium Velocity of 0.04m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.63



(a)

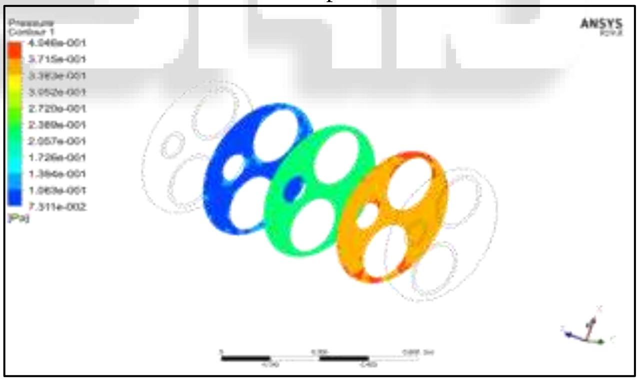


(b)

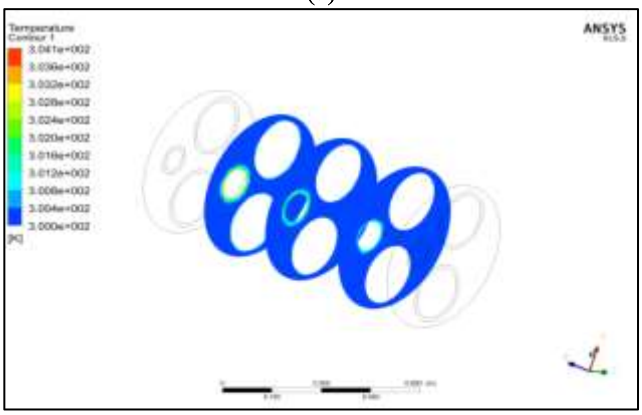


(c)

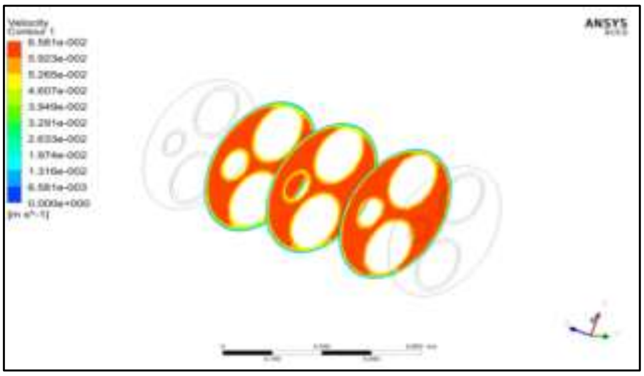
Fig. 4.8: Flow Contours with Heating Medium Velocity of 0.05m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.63



(a)

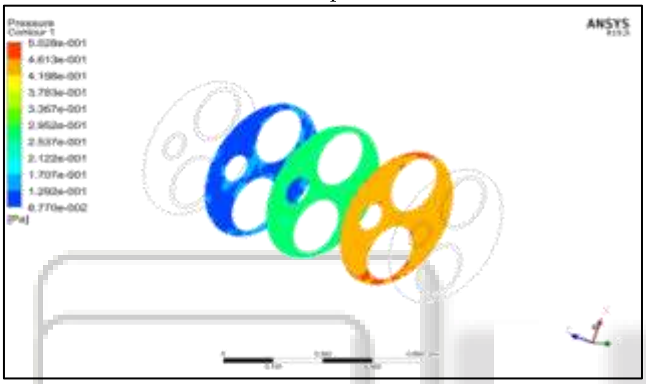


(b)

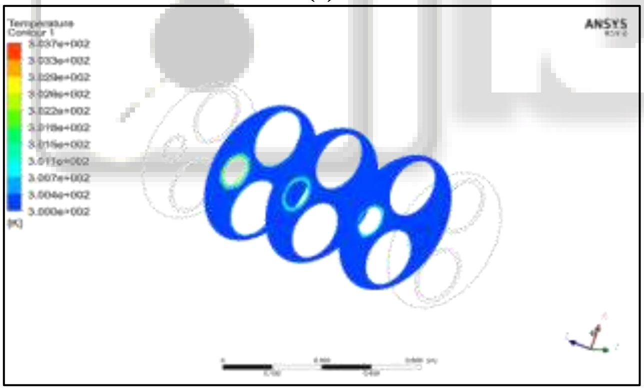


(c)

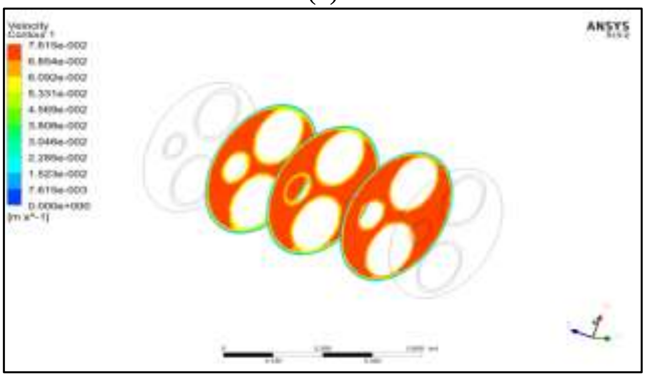
Fig. 4.9: Flow Contours with Heating Medium Velocity of 0.06m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.63



(a)

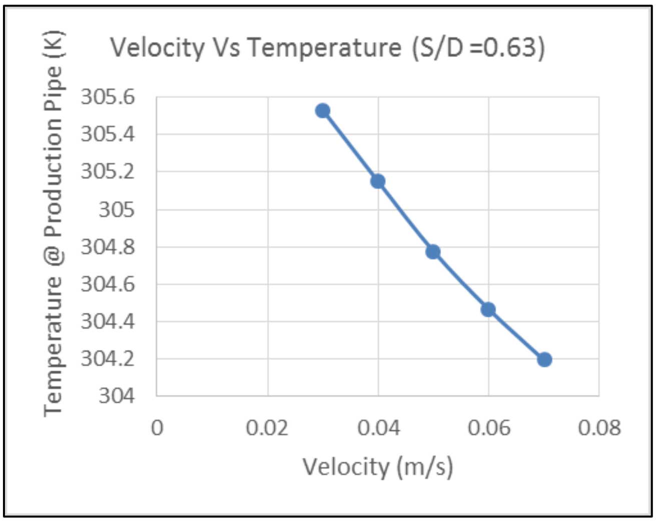


(b)

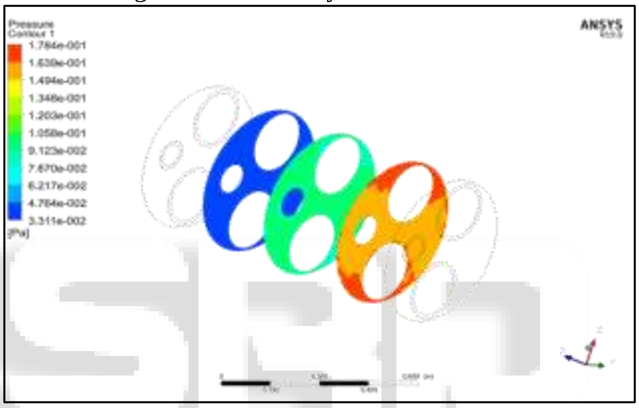


(c)

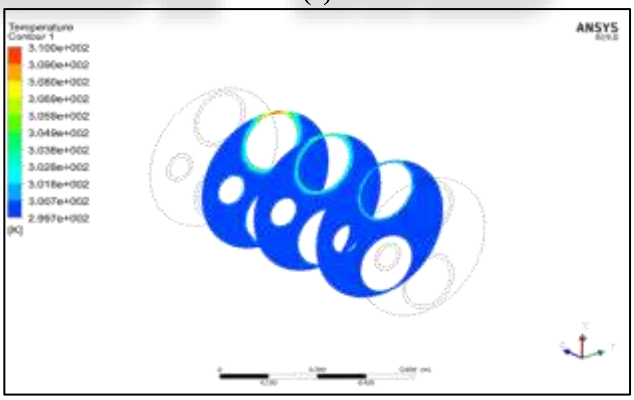
Fig. 4.10:Flow Contours with Heating Medium Velocity of 0.07m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.63



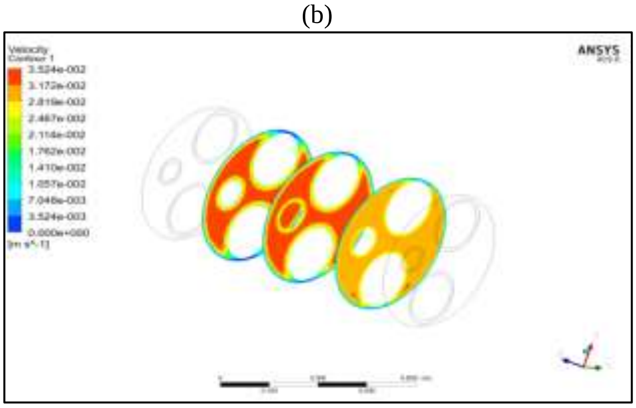
Plot 2: Velocity Vs Temperature With S/D=0.63 Case 3Thermal Analysis of 3 Bundle Pipe with S/D ratio of 0.68 with Heating medium Velocity from 0.03 m/s to 0.07 m/s.



(a)

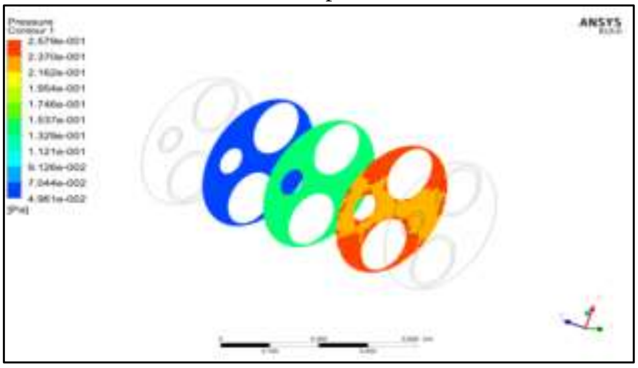


(b)

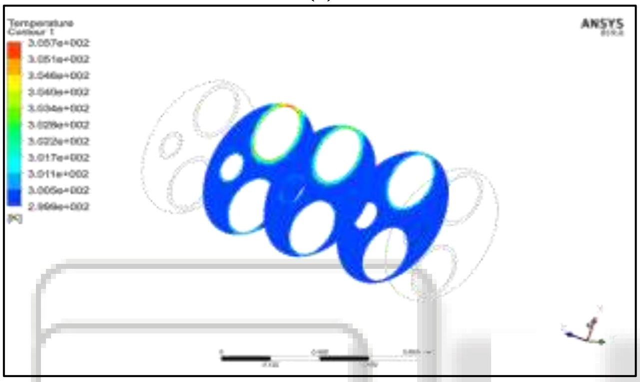


(c)

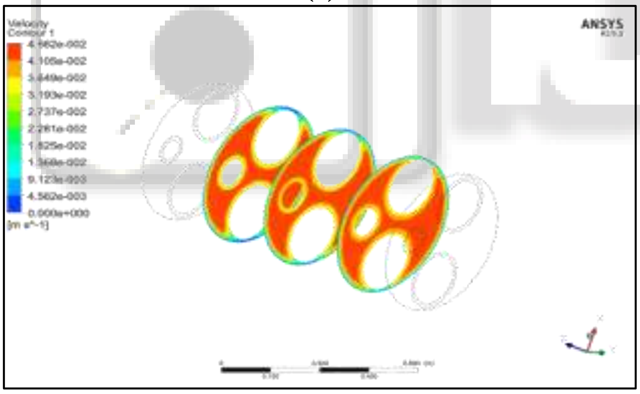
Fig. 4.11: Flow Contours with Heating Medium Velocity of 0.03m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.68



(a)

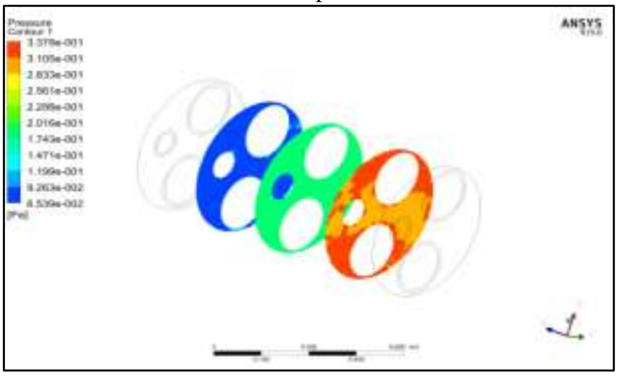


(b)

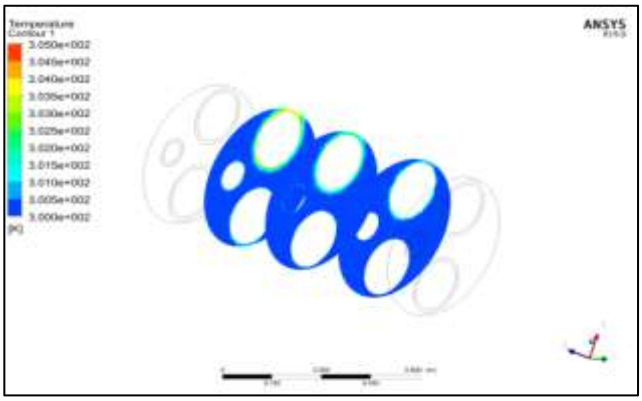


(c)

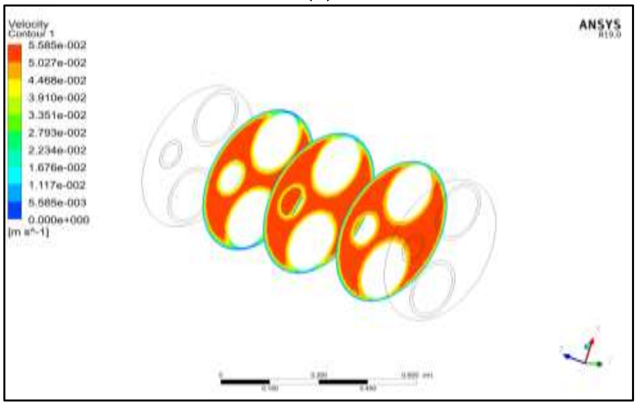
Fig. 4.12: Flow Contours with Heating Medium Velocity of 0.04m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.68



(a)

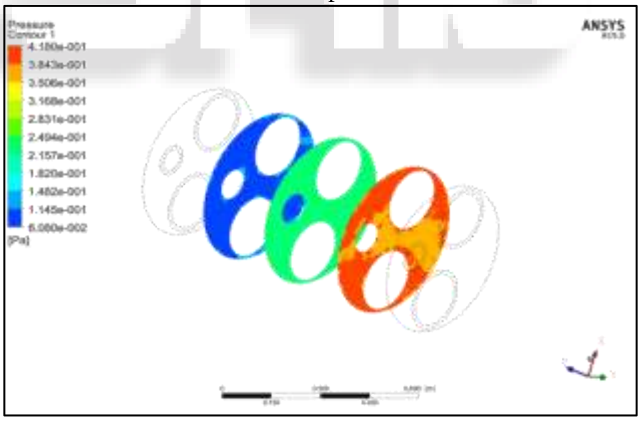


(b)

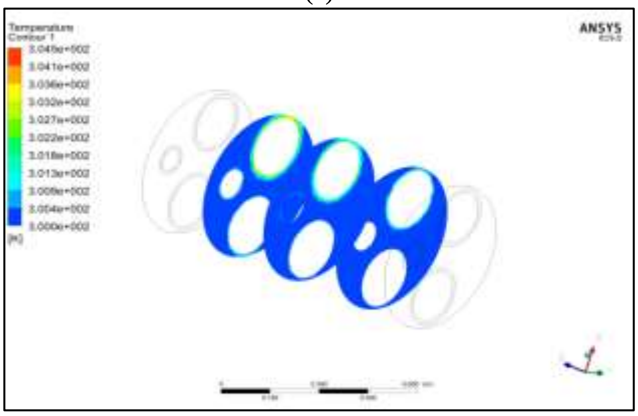


(c)

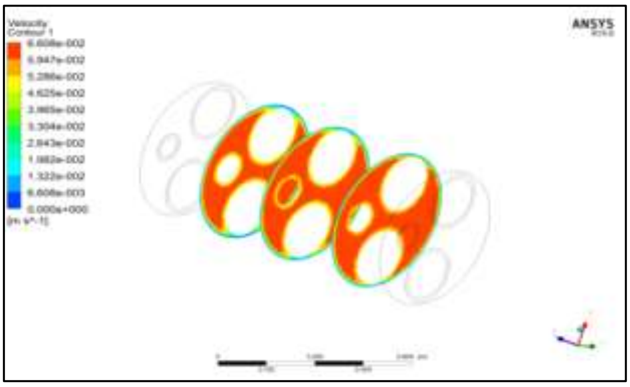
Fig. 4.13: Flow Contours with Heating Medium Velocity of 0.05m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.68



(a)

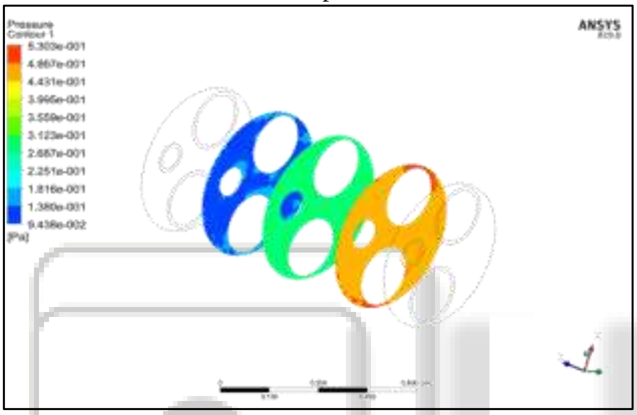


(b)

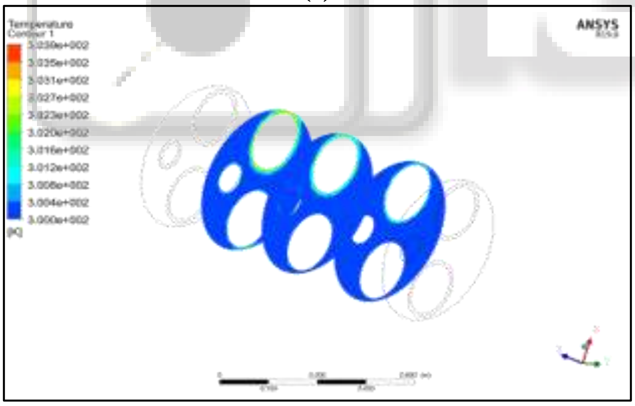


(c)

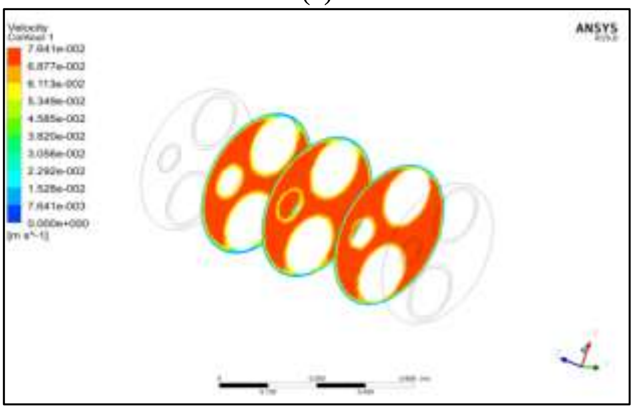
Fig. 4.14: Flow Contours with Heating Medium Velocity of 0.06m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.68.



(a)

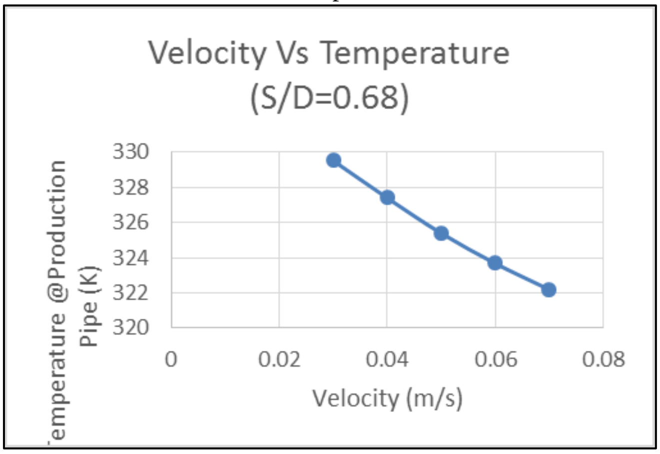


(b)

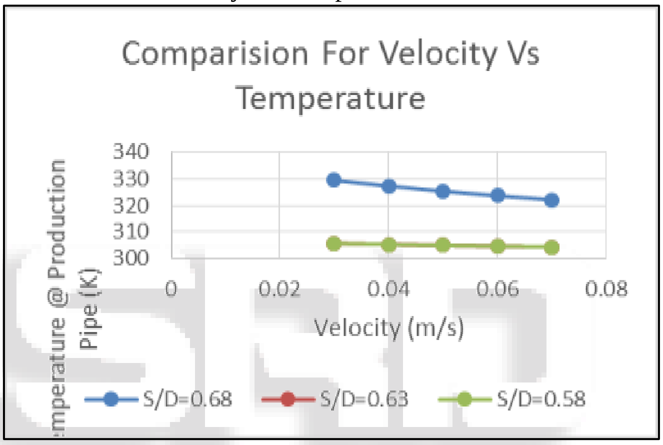


(c)

Fig. 4.15: Flow Contours with Heating Medium Velocity of 0.07m/s (a) Pressure Distribution over the Bundle Pipes (b) Temperature Distribution of 3 Bundle Pipes (c) Velocity Distribution of 3 Bundle Pipes with S/D ratio of 0.68



Plot 3: Velocity Vs Temperature for S/D=0.68.



Plot 4: Comparison of Velocity Vs Temperature

V. Conclusions

A three dimensional CFD model was developed to predict the heat transfer rates in a tube bundle with three internal pipes of Different S/D ratios. The outlet temperatures of all the streams to evaluate the heat transfer rates inside the tubes were obtained. Cases with different inlet temperatures and flow velocities of the heating pipe were run and it was observed that the higher the heating medium inlet temperature and flow velocity the more heat gained by production pipe and thus, higher arrival temperature of production fluid. Out of all the Cases Simulated S/D ratio of 0.58 have Optimum Heat gained by Production Pipe.

References

[1] Sadeghipour, M. S., and Asheghi, M. (1994) “Free Convection Heat Transfer from Arrays of Vertically Separated Horizontal Cylinders at Low Rayleigh Numbers”, International Journal of Heat and Mass Transfer, Volume 37, Pages 103–109 .

[2] Corcione, M. (2005) “Correlating Equations for Free Convection Heat Transfer from Horizontal Isothermal Cylinders Set in a Vertical Array”, International Journal of Heat and Mass Transfer, Volume 48, Pages 3660-3673

[3] Ashjaee, M., and Yousefi, T. (2007) “Experimental study of free convection heat transfer from horizontal isothermal cylinders arranged in vertical and inclined arrays”, Heat Transfer Engineering, Volume 28, Pages 460–471
[4] Reymond, O., Murray, D.B., and O’Donovan, S. (2008) “Natural convection heat transfer from two horizontal cylinders”, Experimental Thermal and Fluid Science,Volume 32, Pages 1702-1709
[5] Zhang, J.J., Chen, M.J., Wang, X, Chacko, J., and Brown, R.J. (2002) “Thermal analysis and design of hot water heated production flowline bundles”, OCT 14015, presented at Offshore Technology Conference held in Houston, May, 2002
[6] Chin, Y.D., and Bomba, J.G. (1999) “Structural and thermal optimisation of cased insulated flowlines”, OTC 11042, presented at Offshore Technology Conference held in Houston, May 1999
[7] Chin, Y.D., Perera, R., Prescot, C.N, Brown, R.J. and Cain, R.E. (2000a) “Thermal Performance of an Insulated Multiple Flowline Bundle Using Active Heating”, SPE 58971, Villahermosa, Mexico, 1-3 February 2000
[8] Chin, Y.D., Zhou, X., Perera, R., and Wang, X. (2000b) “A semi-empirical transient thermal model for multiple flowline bundle cool-down process”, OTC 12037, presented at the 2000 Offshore Technology Conference, Houston, Texas, 1-4 May 2000
[9] Duret, E., Lebreton, E., Heintze, E., Henriot, V., and Saint-Marcoux, J.F. (2000) “Pipeline bundles model implemented into a multiphase flow model”, SPE 62948, presented at the 2000 SPE Annual Technical Conference and Exhibition, Dallas, Texas, 1-4 October 2000
[10] Khlefa, A.E., Gee, F., Gary, H., and Ron, P. (2003) “Active Heating for Flow Assurance Control in Deepwater Flowlines”, OTC 15188, Houston, Texas, 05-08 May 2003.
[11] Fouad, F., Oris, H., Rusty, S. and Weihong, M., (2004) “Evaluation of Pipeline Configurations with Active Heating for Export of Waxy Crude Oil”, SPE 90054, Houston, Texas, 26-29 September 2004