Performance Analysis and Design Optimization of Shell and Tube Heat Exchangers

praveen math (praveen6.math18@gmail.com)
School of Mechanical Engineering REVA University, Bangalore, INDIA

Original Article

Keywords: Heat Exchangers, Turbulence, Shell and Tube, CFD Analysis, Validation, Optimization

Posted Date: April 20th, 2021

DOI: https://doi.org/10.21203/rs.3.rs-407371/v1

PERFORMANCE ANALYSIS AND DESIGN OPTIMIZATION OF SHELL AND TUBE HEAT EXCHANGERS

Praveen Math

School of Mechanical Engineering
REVA University, Bangalore, INDIA
Email- praveen6.math18@gmail.com

Abstract- Shell and Tube heat exchangers are having special importance in boilers, oil coolers, condensers, pre-heaters. They are also widely used in process applications as well as the refrigeration and air conditioning industry. The robustness and medium weighted shape of Shell and Tube heat exchangers make them well suited for high pressure operations. The aim of this study is to experiment, validate and to provide design suggestion to optimize the shell and tube heat exchanger (STHE). The heat exchanger is made of acrylic material with 2 baffles and 7 tubes made of stainless steel. Hot fluid flows inside the tube and cold fluid flows over the tube in the shell. 4 K-type thermocouples were used to read the hot and cold fluids inlet and outlet temperatures. Experiments were carried out for various combinations of hot and cold water flow rates with different hot water inlet temperatures. The flow conditions are limited to the lab size model of the experimental setup. A commercial CFD code was used to study the thermal and hydraulic flow field inside the shell and tubes. CFD methodology is developed to appropriately represent the flow physics and the procedure is validated with the experimental results. Turbulent flow in tube side is observed for all flow conditions, while the shell side has laminar flow except for extreme hot water temperatures. Hence transition k-kl-omega model was used to predict the flow better for transition cases. Realizable k-epsilon model with non-equilibrium wall function was used for turbulent cases. Temperature and velocity profiles are examined in detail and observed that the flow remains almost uniform to the tubes thus limiting heat transfer. Approximately 2/3rd of the shell side flow does not surround the tubes due to biased flow contributing to reduced overall heat transfer and increased pressure loss. On the basis of these findings an attempt has been made to enhance the heat transfer by inducing turbulence in the shell side flow. The two baffles were rotated in opposite direction to each other to achieve more circulation in the shell side flow and provide more contact with tube surface. Various positions of the baffles were simulated and studied using CFD analysis and the results are summarized with respect to heat transfer and pressure loss.

Keywords – Heat Exchangers; Turbulence; Shell and Tube; CFD Analysis; Validation; Optimization

I. INTRODUCTION

A heat exchanger is a piece of equipment built for efficient heat transfer from one medium to another. The media may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power plants, chemical plants, petrochemical plants, petroleum refineries, natural gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air. There are three primary classifications of heat exchangers according to their flow arrangement. In parallel- flow heat exchangers, the two fluids enter the exchanger at the same end, and travel in parallel to one another to the other side. In counter-flow heat exchangers the fluids enter the exchanger from opposite ends. The counter current design is the most efficient, in that it can transfer the most heat from the heat (transfer) medium due to the fact that the average temperature difference along any unit length is greater. In a cross-flow heat exchanger, the fluids travel roughly perpendicular to one another through the exchanger.

For efficiency, heat exchangers are designed to maximize the surface area of the wall between the two fluids, while minimizing resistance to fluid flow through the exchanger. The exchanger's performance can also be affected by the addition of fins or corrugations in one or both directions, which increase surface area and may channel fluid flow or induce turbulence.
The driving temperature across the heat transfer surface varies with position, but an appropriate mean temperature can be defined. In most simple systems this is the "log mean temperature difference" (LMTD). Sometimes direct knowledge of the LMTD is not available and the NTU method is used.

Most common type of heat exchangers is
- Double pipe heat exchanger
- Shell and Tube heat exchanger
- Plate heat exchanger

This paper deals with working principle, experimentation and validation of shell and tube heat exchanger.

1.2 Shell and Tube Heat Exchangers

Shell and tube heat exchangers consist of a series of tubes. One set of these tubes contains the fluid that must be either heated or cooled. The second fluid runs over the tubes that are being heated or cooled so that it can either provide the heat or absorb the heat required. A set of tubes is called the tube bundle and can be made up of several types of tubes: plain, longitudinally finned, etc. Shell and tube heat exchangers are typically used for high-pressure applications (with pressures greater than 30 bar and temperatures greater than 260 °C). This is because the shell and tube heat exchangers are robust due to their shape. Several thermal design features must be considered when designing the tubes in the shell and tube heat exchangers like tube diameter, tube thickness, tube length, tube pitch, tube corrugation, baffle design, etc.

2.2. Watermark Extraction algorithm –

The extraction algorithm process is the inverse of the embedding process. It is assumed that the watermark as well as the see value is available at the receiver end to the authorized users.
The operation of channel separation is applied on the watermarked color image to generate its sub images, and then 2-level discrete wavelet transform is applied on the sub images to generate the approximate coefficients and detail coefficients.

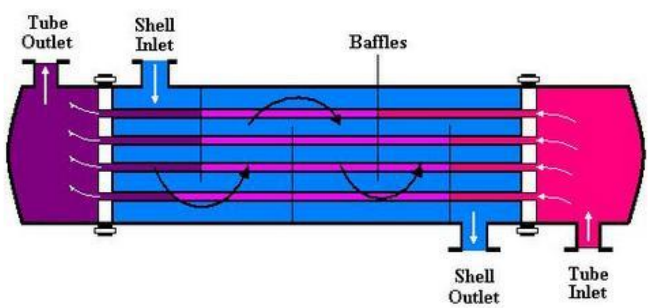


Fig.1: Shell and tube Heat Exchanger Cross Section

1.3 Applications and uses

The simple design of a shell and tube heat exchanger makes it an ideal cooling solution for a wide variety of applications. One of the most common applications is the cooling of hydraulic fluid and oil in engines, transmissions and hydraulic power packs. With the right choice of materials they can also be used to cool or heat other mediums, such as swimming pool water or charge air. One of the big advantages of using a shell and tube heat exchanger is that they are often easy to service, particularly with models where a floating tube bundle (where the tube plates are not welded to the outer shell) is available.

II EXPERIMENTAL SETUP

2.1 General Description

Computer controlled heat exchanger module- HT30XC supplied by Armfield was used for experiment, who is a leading supplier of engineering teaching equipment from United Kingdom.

The HT30XC is a service unit, which provides controlled cold water flow, bi-directional hot water flow and the instrumentation required to do a series of in-depth investigations into heat exchanger performance. The individual heat exchangers can be quickly changed over, to enable comparisons between different types of heat exchanger to be made. The HT30XC requires a user supplied personal computer for the operator interface. The computer connects the HT30XC using a USB interface, providing a simple and straightforward installation and setup procedure. Once the appropriate heat exchanger has been installed and set up, all other functions can be performed under computer control. Appropriate measures have been implemented so that in the case of computer failure or communications breakdown, the system shuts itself down in a safe manner.

Fig. 2: Heat Transfer Service Unit-HT30XC

2.2 Hardware Description

The service unit provides two fluid streams to the heat exchanger, a hot water stream and a cold water stream. The hot water stream is heated in a vessel fitted with an electric heater. The heater is switched on and off by a solid state relay (SSR), which is under software control. A thermostat limits the maximum water temperature to 75ºC for operator safety. A gear pump circulates water from the vessel, through the heat exchanger and back into the heater vessel. Both the pump speed and direction are under software control, enabling for co-current and counter-current investigations over a wide range of flow rates. The cold water stream is generated from a mains water supply. The flow through the heat exchanger is adjusted by a variable flow valve, again under software control. A manually adjustable pressure regulator is used to minimise the effect of mains pressure fluctuations.

Conditioning circuits for up to 10 K-type thermo- couples are included, (the thermocouples themselves are supplied with the heat exchangers). The instrumentation also includes flow meters to measure the flow rates of the two fluid streams. Switching on the unit puts it into ‘Standby’ mode. From this mode it is necessary for a regular series of pulses to be received from the software (via the built in USB interface) to fully power up the unit. This ensures that unless the control software is running, the heaters, the pump and the cold water control valve cannot be switched on. The unit also includes an emergency stop switch. All electrical circuits are located in a bench mounted ABS supporting base, and protected by a Residual Current Device (RSD) for operator safety. The ABS base includes a drip tray and drain tap in case of water spillage or leakage.

2.4 Technical Details

The shell and tube heat exchanger supplied is designed to demonstrate liquid to liquid heat transfer in a 1-7 shell and tube heat exchanger (one shell and seven tubes with two transverse baffles in the shell).

The miniature shell and tube heat exchanger consists of the following features,

- Hot fluid in the inner tubes and cold fluid in outer shell to minimise heat loss from the exchanger without the need for additional insulation

- Seven stainless steel tubes, 6.35mm OD

- The outer annulus, end caps and baffles constructed from clear acrylic to allow visualisation of the heat exchanger construction and minimise thermal losses

- Cold fluid (cold water) enters one end of the shell at the bottom and exits at the opposite end at the top having flowed over and under two transverse baffles inside the shell

Thermocouples are installed at the following four locations: Hot fluid inlet, Hot fluid outlet, Cold fluid inlet, Cold fluid outlet

Table 1:Overall Dimensions	
Height	0.19m
Width	0.43m
Depth	0.39m
Heat transfer area	20,000mm^2
Volume	0.06m^3
Weight	5kg

2.5 Experiment

Experiments were carried out for various combinations of hot and cold water flow rates with different hot water inlet temperatures. The flow conditions are limited to the lab size model of the experimental setup. Experiment was carried out until the system reaches a steady state condition. Measure data were recorded for every 10 seconds. Temperatures, flow rates, water density etc., are recorded and tabulated directly in excel format. The converged values are used as an input for CFD analysis.

Totally 252 different cases were considered per below matrix for experiment.

Table 2:Experiment Matrix Hot Flow Vs. Cold Flow Vs. Hot Temperature							
Cold Flow (LPM)	0.5	1	1.5	2	3	4	5
Hot Flow (LPM)	2	2.5	3	3.5	4	4.5	-
Temperature (T1)	40	45	50	55	60	65	-

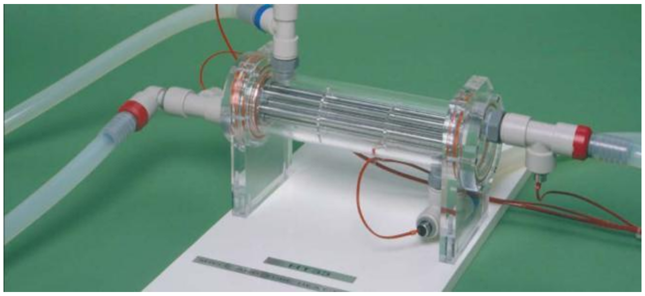


Fig 3: Shell and Tube Heat Exchanger

III ANALYSIS

A commercial CFD code was used to study the thermal and hydraulic flow field inside the shell and tubes. CFD methodology is developed to appropriately represent the flow physics and the procedure is validated with the experimental results.

3.1 Modelling

Heat exchanger modelling was done by using 3D modelling software CATIA. Actual dimensions of the physical part were used to create the CAD model. The final designed model was used for grid generation.

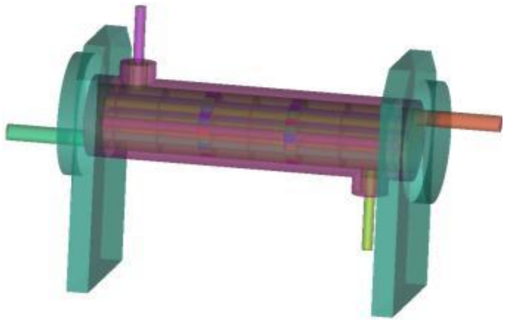


Fig.4: STHE-Physical Model and Analysis Model

3.2 Computational Geometry

The model was imported to Ansys ICEMCFD software and the pre-processing works like geometry clean- up, Parts segregation, naming were done. The parts were segregated to different names as shell, tubes, baffles, stands etc., in order to apply different boundary conditions. Monitoring planes were created at different locations in the shell and tube to monitor the flow.

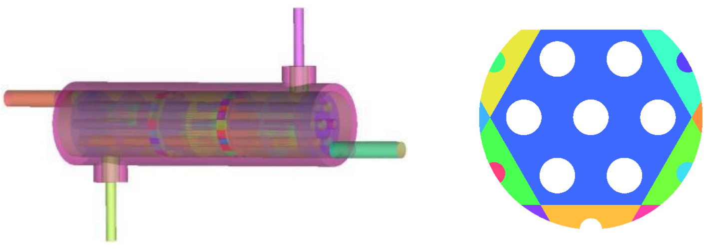


Fig.5: Computational Geometry - Heat Exchanger, Baffle

3.3 Computational Grid

Grid generation for the model was done using Ansys ICEMCFD; grid independence study was carried out by comparing 3 different mesh sizes, from that 6mm global mesh size was selected. Tetrahedral mesh having 6 mm global size along with 3 inflation layer (Prism mesh) was generated. Around 2.1 million elements were created by discretizing the domain. Thin cuts were defined between surfaces having less clearance to ensure good quality and error free mesh.

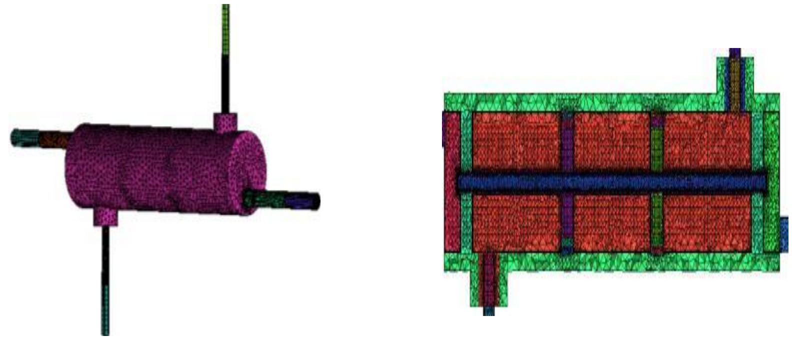


Fig. 6: Computational Mesh - Heat Exchanger, Cross Section

3.4 CFD Analysis

A commercial CFD code Ansys Fluent was used to study the thermal and hydraulic flow field inside the shell and tubes. CFD methodology is developed to appropriately represent the flow physics and the procedure is validated with the experimental results.

Reynolds number was calculated for flow in shell and tubes at different location to study the type of flow. Turbulent flow in tube side is observed for all flow conditions, while the shell side has laminar flow except for extreme hot water temperatures. Suitable turbulent models was selected for different flow condition, Hence transition k-kl-omega model was used to predict the flow better for transition cases and Realizable k-epsilon model with non-equilibrium wall function was used for turbulent cases. From 252 cases of experiment 27 cases were selected for analyses which are average and extreme conditions.

Table 3:Analysis Matrix			
Cold Flow (LPM)	1	3	5
Hot Flow (LPM)	2	3	4.5
Temperature (T1)	40	50	60

Material properties like acrylic (Poly methyl methacrylate) and stainless steel were assigned for solids and water for hot and cold fluids. Simulation was carried out till the solution gets converged. Results of the experiment and analysis were compared and an appropriate simulation procedure was developed.

3.5 Post processing

Temperature and velocity profiles are examined in detail and observed that the flow remains almost uniform to the tubes thus limiting heat transfer.

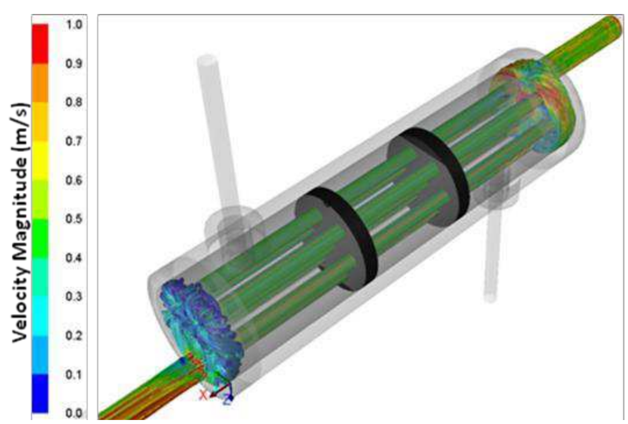


Fig.7: Pathlines Colored by Velocity Magnitude - Shell Side

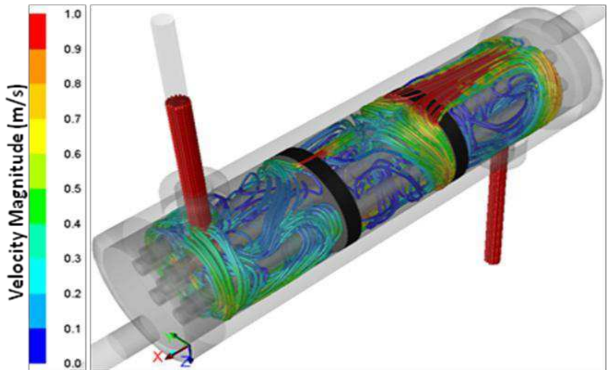


Fig.8: Pathlines Coloured by Velocity Magnitude -Tube Side

Approximately 2/3rd of the shell side flow does not surround the tubes due to biased flow contributing to reduced overall heat transfer. More pressure loss was observed in the shell side due to the flow experiencing 90 degree bend in the baffle locations.

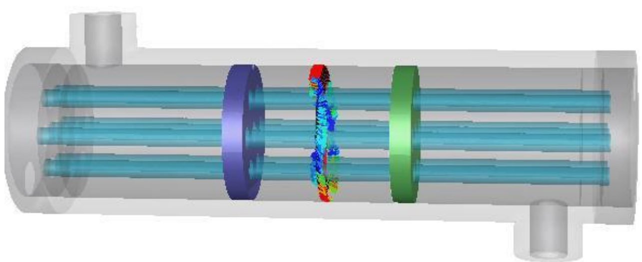
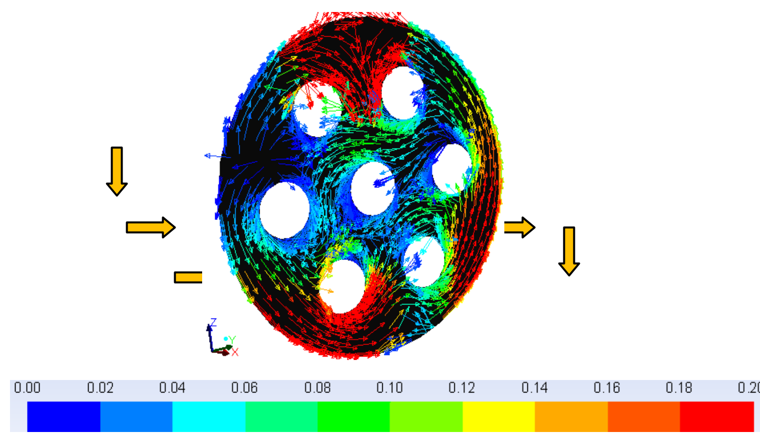


Fig.9: Velocity Vector Location



Velocity Magnitude (m/s)

Fig. 10: Velocity Vector for Baseline model in Cold Flow Path

IV CONCLUSION

Generally for heat exchangers there are three goals that are normally considered in the optimal design of heat exchangers:

(1) Minimizing the pressure drop (pumping power),

(2) Maximizing the thermal performance and

(3)Minimizing the entropy generation (thermodynamic irreversibility).

On the basis of the findings by interpreting the results of analysis an attempt has been made to enhance the heat transfer by inducing turbulence in the shell side flow. The two baffles were rotated in opposite direction to each other to achieve more circulation in the shell side flow and provide more contact with tube surface. Various positions of the baffles were simulated and studied using CFD analysis and the results are summarized with respect to heat transfer and pressure loss. The two baffles of STHE were rotated in opposite directions in 4 steps like 60^{o}, 120^{o}, 240^{o}, and 300^{o} rotations respectively.

Comparison was made with the results of baseline model. It has been observed that the pressure drop value has reduced up to 10% from the baseline model with same heat transfer rate. There is no appreciable change in heat transfer, this could be due to the space between two baffles were less and the flow could not surround the tubes.

Table 4 : Design Optimization Total Pressure Drop in Shell side Flow		
Model	Total Pressure Drop (Pa)	Reduction in Pressure drop compared to Baseline (%)
Baseline	13211	-
60^o Rotation	12194	8
120^o Rotation	11876	**10**
240^o Rotation	12127	8
300^o Rotation	12164	8

The velocity vectors at a plane normal to the flow of cold water (in shell side) are shown below, which depicts how the flow circulates inside the shell between two baffles. Among all the models 120^o rotation has least pressure drop with same hear transfer rate.

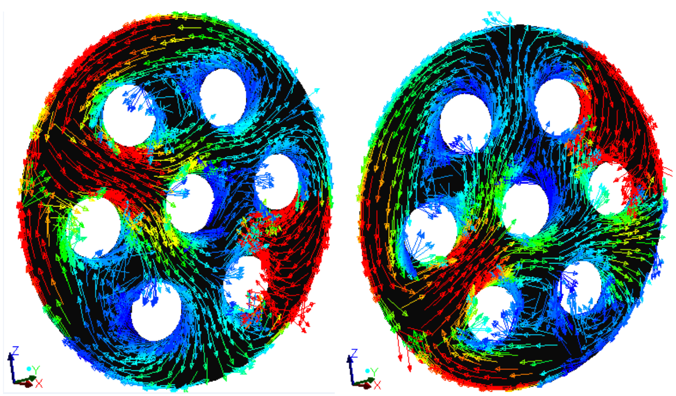


Fig.13: Velocity Vector for 60^o and 120^o Baffle Rotation

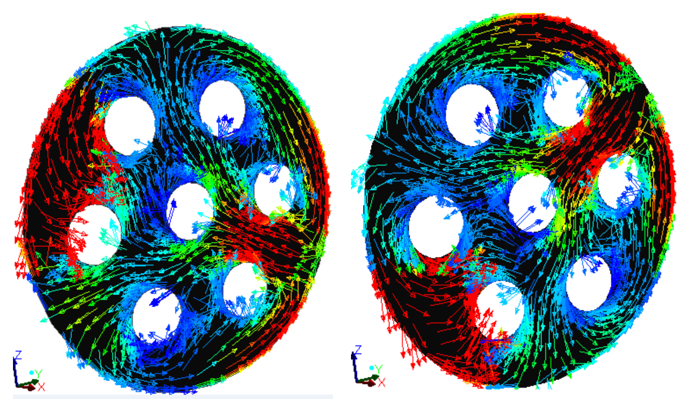


Fig.14: Velocity Vector for 240^o and 300^o Baffle Rotation

DECLERATIONS :

I PRAVEEN MATH, author of the above titled paper hereby declare that the work included in the above paper is original and is an outcome of the research carried out by the authors indicated in it. **Various positions of the baffles were simulated and studied using CFD analysis and the results are summarized with respect to heat transfer and pressure loss.** The authors declare that they have no competing interests and no funding agency. Computer controlled heat exchanger module- HT30XC supplied by Armfield was used for experiment, who is a leading supplier of engineering teaching equipment. The HT30XC is a service unit, which provides controlled cold water flow, bi-directional hot water flow and the instrumentation required to do a series of in-depth investigations into heat exchanger performance.

V. REFERENCES

[1] M. Mirzaei, H. Hajabdollahi, H. Fadakar, Multi-objective optimization of shell-and-tube heat exchanger by constructal theory, Appl. Therm. Eng. 125 (2017)9–19.

[2] L. Liu, N. Ding, J. Shi, N. Xu, W. Guo, C. Wu, Failure analysis of tube-to-tubesheet welded joints in a shell-tube heat exchanger, Case Stud. Eng. Fail. Anal. (2016)32–40.

[3] A.A. Abd, S.Z. Naji, Analysis study of shell and tube heat exchanger for clough company with reselect different parameters to improve the design, Case Stud.Therm. Eng. 10 (2017) 455–467.

[4] S. Shinde, U. Chavan, Numerical and experimental analysis on shell side thermo-hydraulic performance of shell and tube heat exchanger with continuous helicalFRP baffles, Therm. Sci. Eng. Prog. (2017),https://doi.org/10.1016/j.tsep.2017.11.006.

[5] D. Eryener, Thermoeconomic optimization of baffle spacing for shell and tube heat exchangers, Energy Convers. Manag. 47 (11–12) (2006) 1478–1489.

[6] C. Yu, Z. Ren, M. Zeng, Numerical investigation of shell-side performance for shell and tube heat exchangers with two different clamping type anti-vibrationbaffles, Appl. Therm. Eng. (2018).

[7] H.S. Dizaji, S. Jafarmadar, S. Asaadi, Experimental exergy analysis for shell and tube heat exchanger made of corrugated shell and corrugated tube, Exp. Therm.Fluid Sci. (2017) 475–481.

[8] A. Alimoradi, F. Veysi, Optimal and critical values of geometrical parameters of shell and helically coiled tube heat exchangers, Case Stud. Therm. Eng. (2017)73–78.

[9] B. Gao, Q. Bi, Z. Nie, J. Wu, Experimental study of effects of baffle helix angle on shell-side performance of shell-and-tube heat exchangers with discontinuoushelical baffles, Exp. Therm. Fluid Sci. (2015) 48–57.

[10]R.K. Sinnott, Chemical Engineering Design, 2 ed, 6 Pergamon, 1993.

[11]H. Li, V. Kottke, Effect of baffle spacing on pressure drop and local heat transfer in shell-and-tube heat exchangers for staggered tube arrangement, Int. J. Heat.Fig. 6.Tube length against heat transfer coefficient and pressure drop for shell side.Fig. 7.Fouling rate changing on shell side against heat transfer.Fig. 8.Fouling rate changing on shell side against heat transfer.A.A. Abd et al.Case Studies in Thermal Engineering 12 (2018) 563–568567

Figures

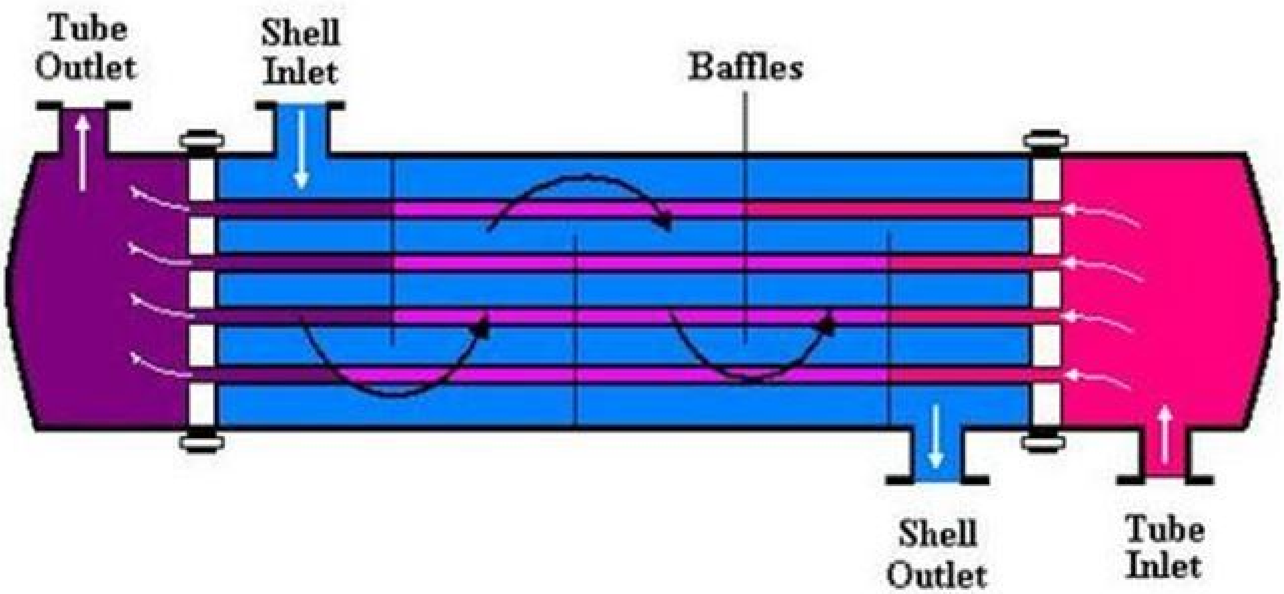


Figure 1

Shell and tube Heat Exchanger Cross Section

Figure 2

Heat Transfer Service Unit-HT30XC

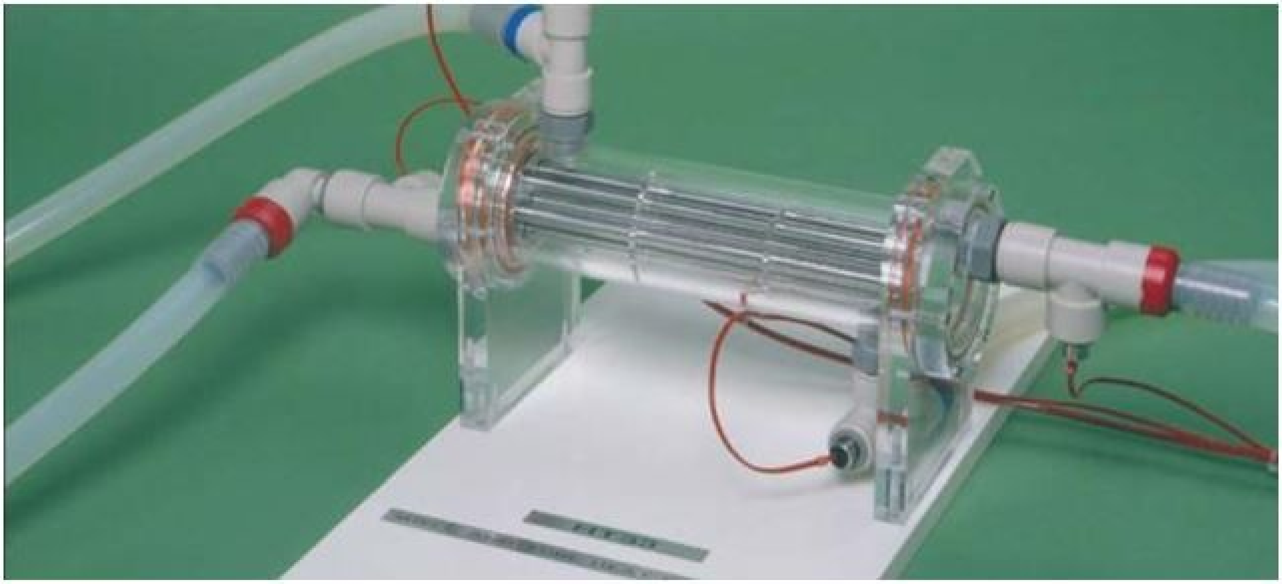


Figure 3

Shell and Tube Heat Exchanger

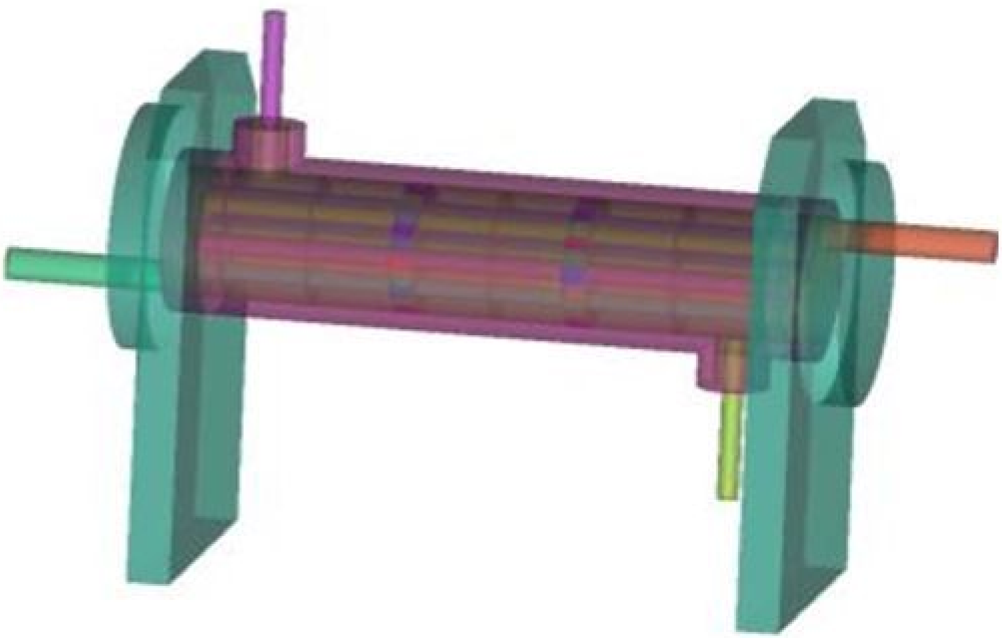


Figure 4

STHE-Physical Model and Analysis Model

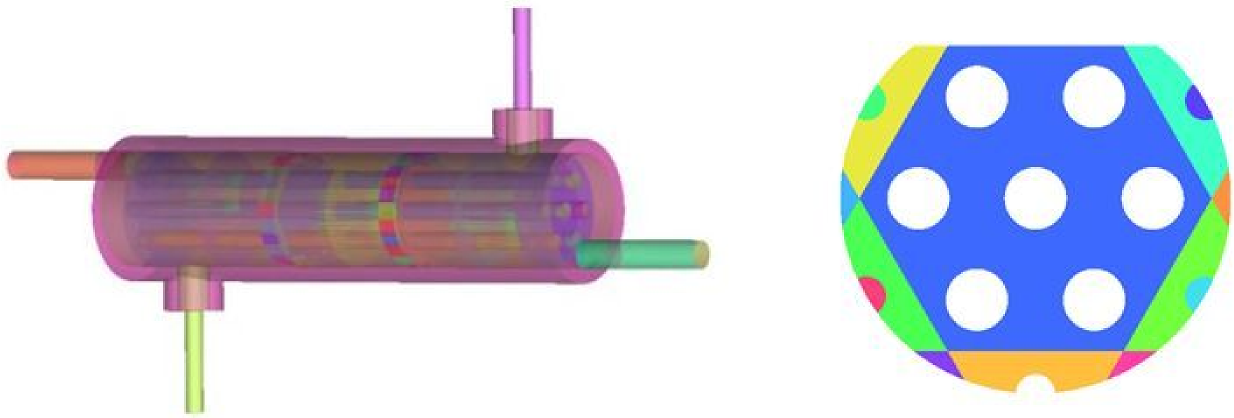


Figure 5

Computational Geometry - Heat Exchanger, Baffle

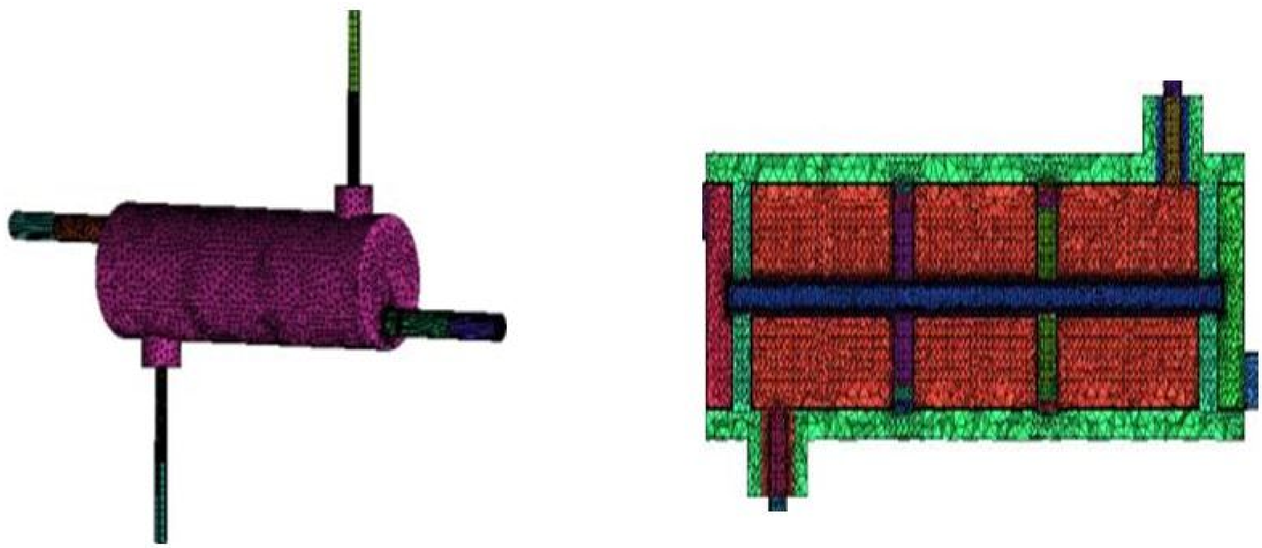


Figure 6

Computational Mesh - Heat Exchanger, Cross Section

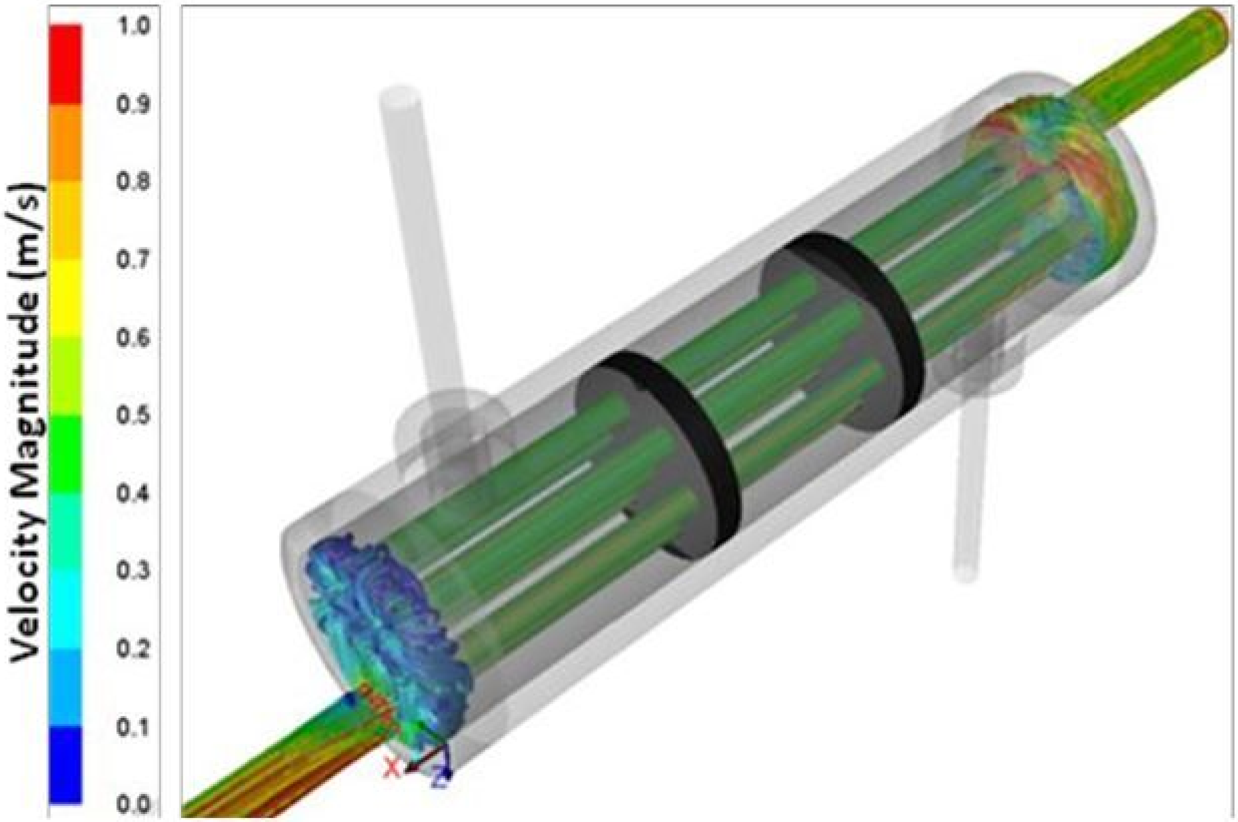


Figure 7

Pathlines Colored by Velocity Magnitude - Shell Side

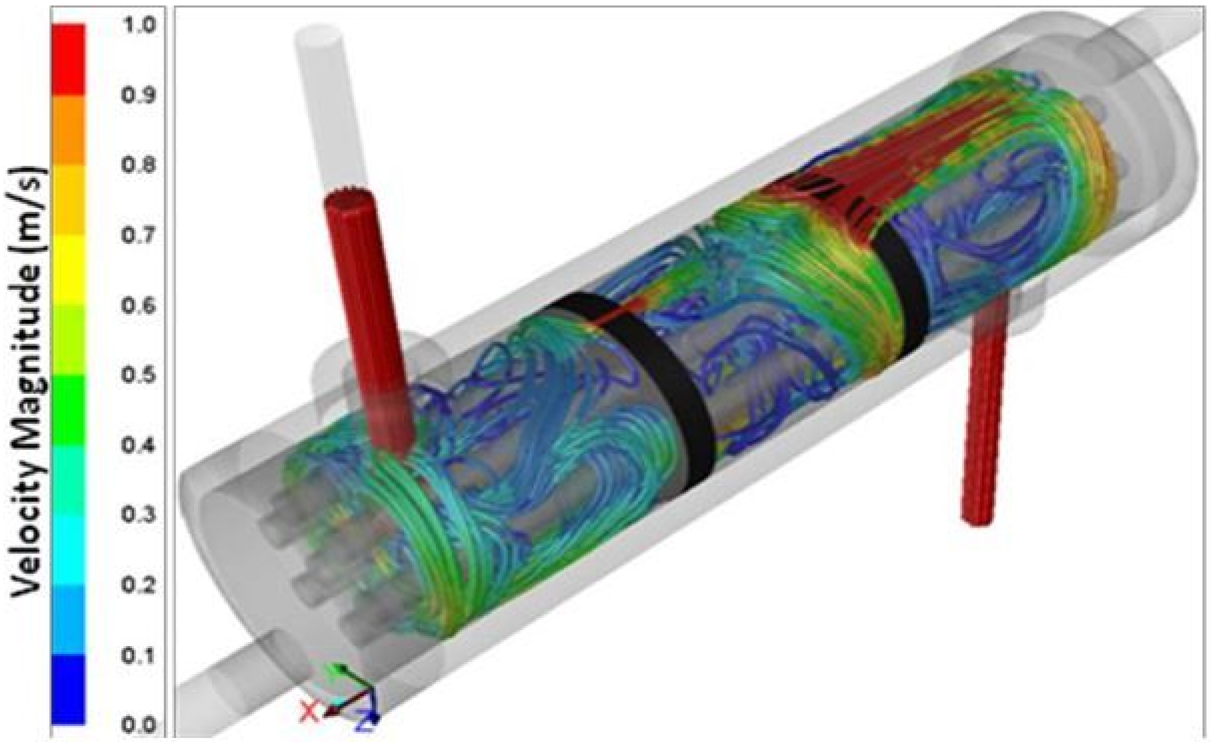


Figure 8

Pathlines Coloured by Velocity Magnitude -Tube Side

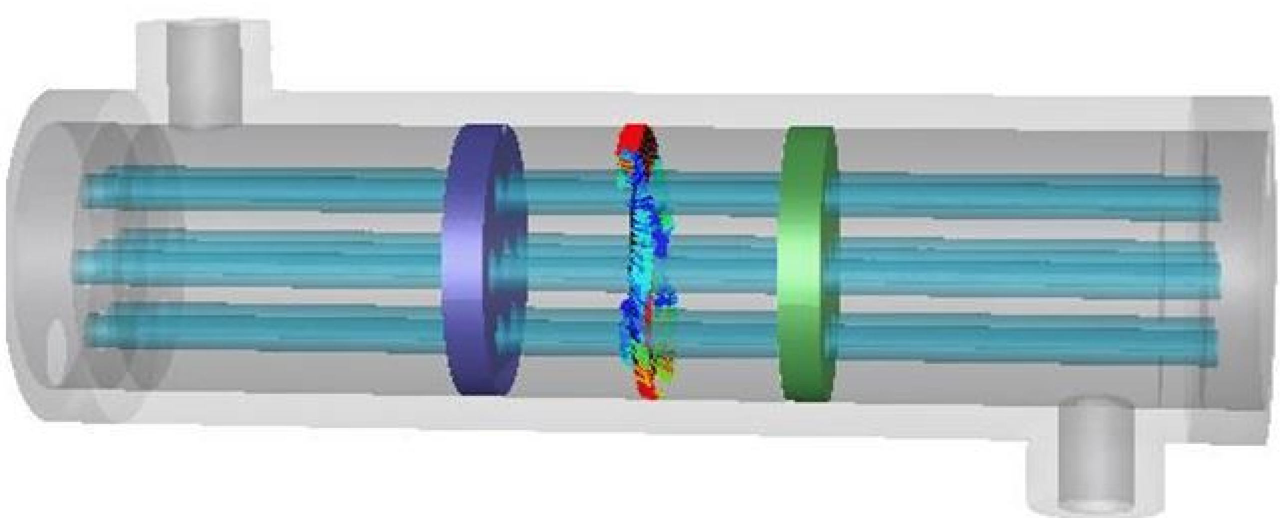


Figure 9

Velocity Vector Location

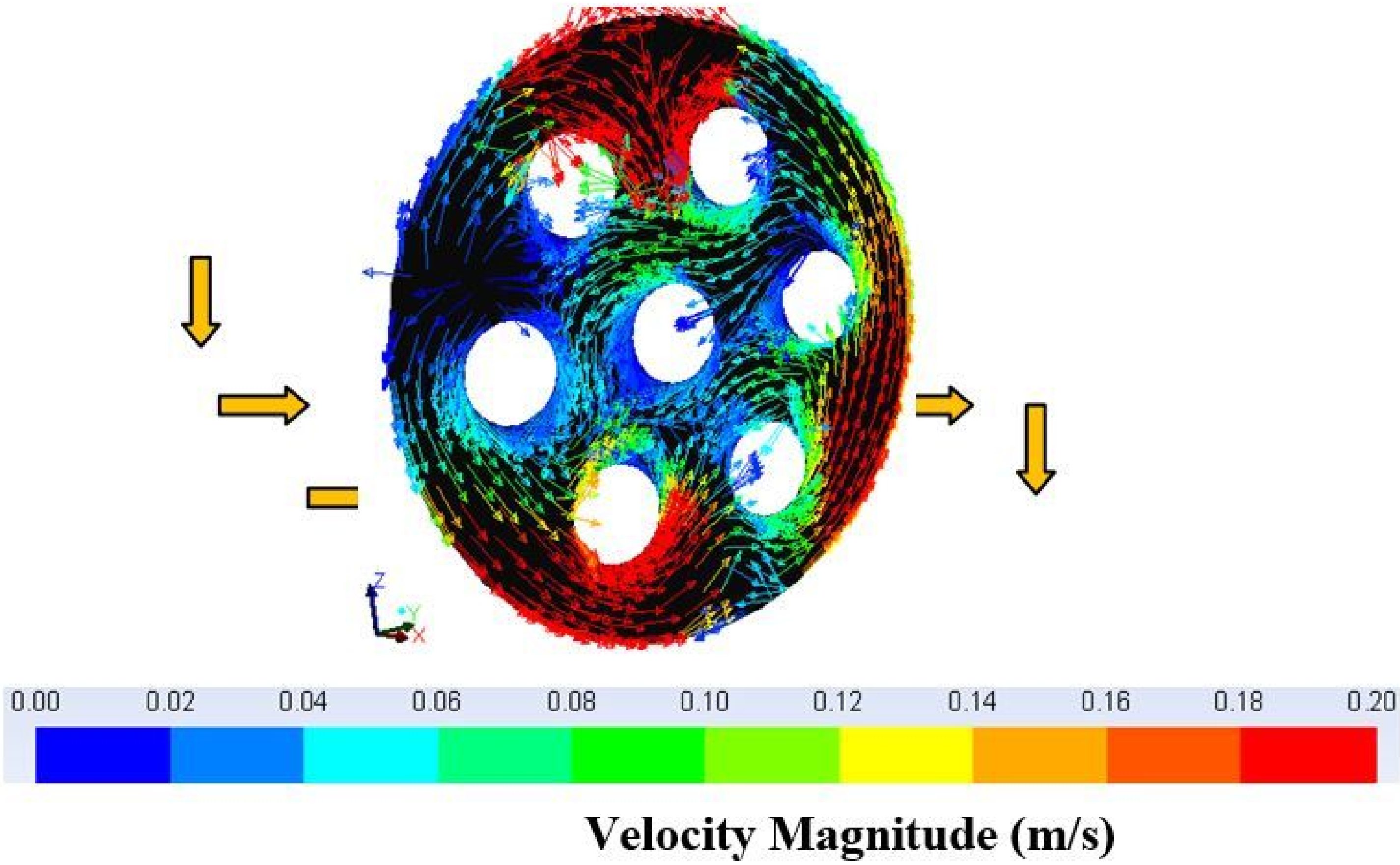


Figure 10

Velocity Vector for Baseline model in Cold Flow Path

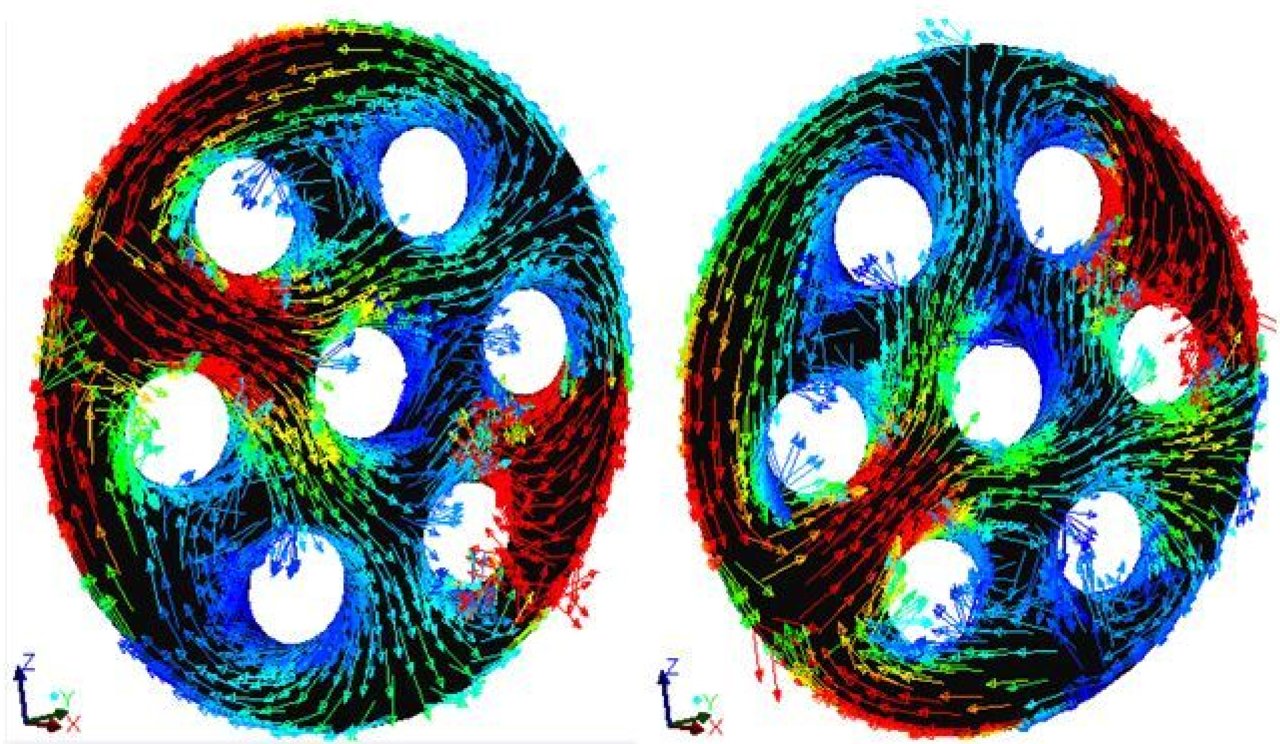


Figure 11

Velocity Vector for 60o and 120 o Baffle Rotation

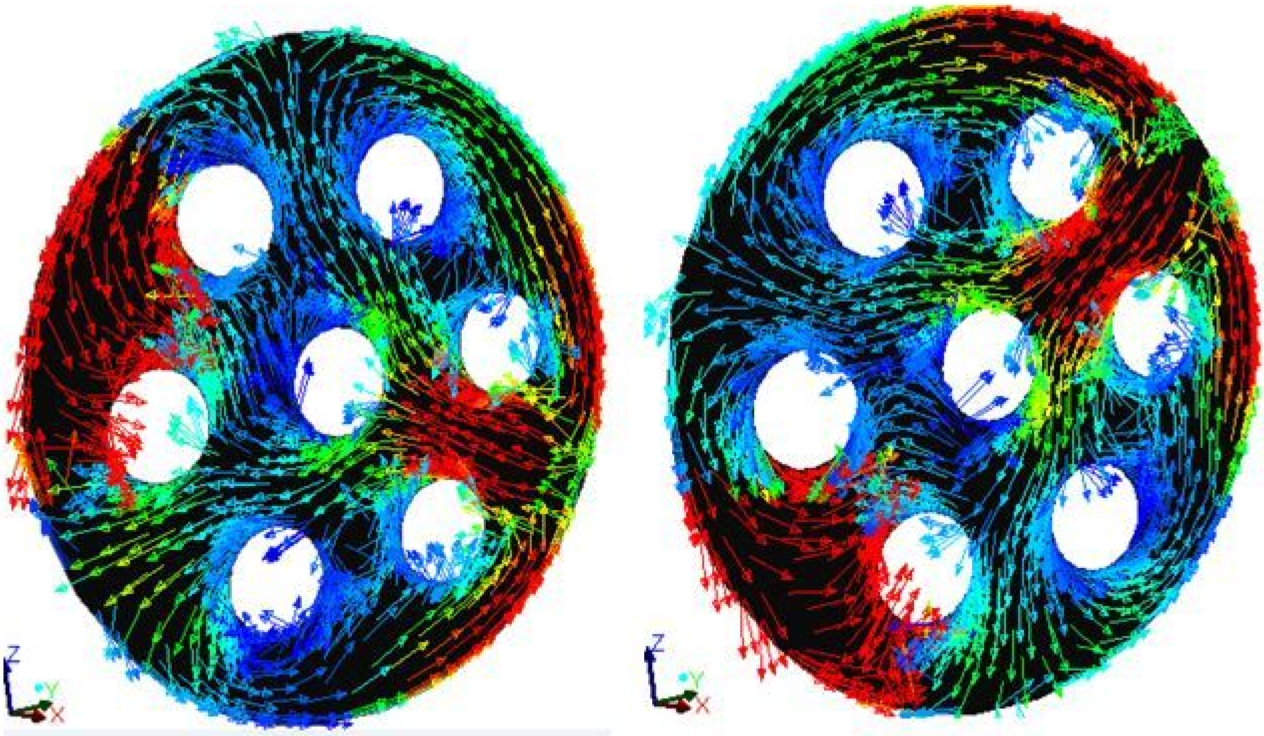


Figure 12

Velocity Vector for 240o and 300 o Baffle Rotation

Supplementary Files

This is a list of supplementary files associated with this preprint. Click to download.

- sm.doc