University of Baghdad
College of Engineering
JE
JOURNAL OF ENGINEERING

Journal of Engineering

journal homepage: www.joe.uobaghdad.edu.iq
Number 2 Volume 25 February 2019

Chemical, Petroleum and Environmental Engineering

CFD Application on Shell and Double Concentric Tube Heat Exchanger

Basma Abbas Abdulmajeed*
Professor
College of Engineering - University of Baghdad
email:basma1957@yahoo.com

Hawraa Riyadh Jawad
M Sc student
College of Engineering - University of Baghdad
email:eng.hawraa90@gmail.com

ABSTRACT

This work is concerned with the design and performance evaluation of a shell and double concentric tubes heat exchanger using Solid Works and ANSY (Computational Fluid Dynamics). Computational fluid dynamics technique which is a computer-based analysis is used to simulate the heat exchanger involving fluid flow, heat transfer. CFD resolve the entire heat exchanger in discrete elements to find: (1) the temperature gradients, (2) pressure distribution, and (3) velocity vectors. The RNG k-ε model of turbulence is used to determining the accurate results from CFD.

The heat exchanger design for this work consisted of a shell and eight double concentric tubes. The number of inlets are three and that of outlets are also three for all the fluids that pass through the heat exchanger.

A comparison was made for the numerical and experimental results and it was found that the percentage error for the hot oil outlet temperature was (6.8%) and the percentage error was (-21%) for cold water outlet temperature.

Key Words : CFD ; shell-and-double concentric tube; heat exchanger; performance study; numerical simulation.

تطبيق ديناميكا الموائع الحسابية على المبادل الحراري ذو القشرة والانابيب المتداخلة المتمركزة

بسمة عباس عبد المجيد
استاذ
كلية الهندسة – جامعة بغداد

حوراء رياض جواد
طالبة ماجستير
كلية الهندسة – جامعة بغداد

الخلاصة

ان هذه الدراسة تعتمد على تصميم وتقييم اداء المبادل الحراري ذو القشرة والانابيب المتداخلة المتمركزة باستخدام Solid Works و ANSYS (ديناميكا الموائع الحسابية). الهدف الرئيسي من هذا المشروع البحثي هو التحقق من المبادل الحراري المصمم باستخدام طريقة (Kern) من خلال استخدام برامج ديناميكا الموائع الحسابية التجارية.

تقنية ديناميكا الموائع الحسابية التي هي عبارة عن تحليل يستند الى الكمبيوتر يستخدم لمحاكاة المبادل الحراري التي تنطوي على تدفق الموائع، نقل الحرارة. ديناميكا الموائع الحسابية يحل كل المبادل الحراري في عناصر منفصلة للحصول على تدرجات درجة الحرارة ،توزيع الضغط، و ناقلات السرعة. يتم استخدام نموذج الاضطراب (RNG k-ε) للحصول على نتائج دقيقة من ديناميكا الموائع الحسابية. يتكون هيكل المبادل الحراري لهذا المشروع من قشرة وتضم داخلها ثمانية انابيب مزدوجة متحدة المركز. هناك ثلاث مداخل وثلاث منافذ لجميع السوائل.

*Corresponding author
Peer review under the responsibility of University of Baghdad.
https://doi.org/10.31026/j.eng.2019.02.09
2520-3339 © 2018 University of Baghdad. Production and hosting by Journal of Engineering.
This is an open access article under the CC BY-NC license http://creativecommons.org/licenses/by-cc-nc/4.0/).
Article received: 28/10/2017
Article accepted: 9/1/2018

تم اجراء مقارنة للنتائج العددية والعملية وتبين ان نسبة الخطأ لدرجة حرارة الزيت الساخن الخارجة من المبادل كانت (8.6%) وكانت نسبة الخطأ (21%) بالنسبة لدرجة حرارة الماء البارد الخارج من المبادل.

الكلمات الرئيسية : ديناميكا الموائع الحسابية ، القشرة والانابيب المتداخلة متحدة المركز ، مبادل حراري، دراسة الاداء، محاكاة رقمية.

1. INTRODUCTION

The shell-and-double concentric-tube heat exchanger is one of the devices to transfer heat. They are used to transfer the thermal energy between three fluids at different temperatures. A small simulated prototype for any process in the industry can be done by manufactured and subjected to the same boundary conditions. It is an expensive route in cost and may take a long time because of the repetitive manufacturing process. The development of CFD or the computer programing and CFD made the numerical analysis to take this action instead of the prototype. CFD is an advantageous technique to analyze the problems, such as heat transfer and fluid flow. Three stages represent the main fundamentals of the numerical process to be simulated.

Mahmoud, et al., **2014**, studied numerical simulations of the turbulent, three-dimensional fluid flow, heat transfer, and friction for shell and tube heat exchanger using ANSYS FLUENT CFD technique. The parameters studied are: the friction factor and the Nusselt number and at different values of mass flow rates (from 0.5 kg/s to 2.0 kg/s of water fluid flow) with the corresponding effect on thermal enhancement factor, and a correlation relating heat transfer and friction for different fan diameters and blade angles have been developed.

Mohammad, et al., **2014,** studied a shell and tube type of heat exchanger which had simple baffles using the software of CFD. The turbulence model used is k-ω SST model, which gave better results. Three types of meshes were tried; coarse, medium and fine. They showed that aspect ratio has no significant effect. The velocity and temperature profiles both were compared with experimental results. The comparison showed that it was accepted to a good degree.

The performance of a triple concentric pipe heat exchanger numerically was studied by **G.A. Quadir, et al., 2014**. It was done by using the FEM or the finite element method. It was conducted using different flow arrangements and for steady-state conditions. Conditions of insulated/non-insulated conditions heat exchanger were studied. They represented the results in the dimensionless form of temperature variations for the three fluids with different flow rates. This was done along the length of the heat exchanger.

Sinziana, et al., 2012, studied a calculation algorithm and experimental mode of triple concentric-tube heat exchangers. In this study, they determined the partial coefficients of heat transfer afferent to three fluids that exchange heat between them based on experimental results. It was assumed that there was no phase transformation during the experimental work,.

Gurbir and Hemant, 2014, investigated shell and tube heat exchanger for both experimental and numerical data. In experimental work, the temperature variations are calculated for parallel and counter flow by varying the mass flow rate of fluid of 2L/min and 3L/min, while in CFD simulation (by using SOLID WORKS software, GAMBIT and ANSYS 13.0) they found the temperature gradients, pressure distribution, and velocity vectors. The analysis showed that there is a difference between temperatures values computed from the experiment and the simulation by ANSYS 13.0.

In **2015, Basma, and Fadhil,** published a study concerning the design of shell-and-double concentric tube heat exchangers. They studied both design and performance calculations of the heat exchanger.

The design was conducted according to Kern method with volumetric flow rates of 3.6 m^3/h and 7.63 m^3/h for the hot oil and water respectively. They studied: temperature, the flow rate of hot oil and that of cold water and the pressure drop. They compared the theoretical and experimental showing the percentage error for the hot oil outlet temperature to be (- 1.6%). The errors for the pressure drop in both the shell and the concentric tubes were (17.2%) and (- 39%) respectively. Also, an error (- 3.3%) for cold water outlet temperature was calculated. (18%) the error was for the pressure drop in the annulus formed. The total power consumed percentage error was (-10.8%). Also, they compared the new design and the conventional heat exchanger considering, length, mass, pressure drop and total power consumed.

To increase the efficiency of the heat exchanger, **Basma, and Noor, 2017,** used γ- Al_2O_3/water nanofluid as a cold stream in the shell and double concentric tube heat exchanger. A hot stream of basis oil was used counter-currently. The results showed that as nanofluid concentrations increased, each of overall heat transfer coefficient and the Nusselt number increased. They showed that because of the low concentration used, the pressure drop of nanofluid increased slightly than that of the base fluid.

The main objective of this research project is to study the designed heat exchanger, with the use of the Kern's technique, using the software of Computational Fluid Dynamics (CFD) by simulation the 3D geometry for counterflow smooth tube heat exchanger using hot oil that passes through the shell side and the inner tube side, while the cooling water passes counter currently in the annulus between the concentric tubes by using computational fluid dynamic (ANSYS-FLUENT 15) software.

2. EXPERIMENTAL WORK

2.1 Manufacturing and Description

The experimental rig is a shell and double concentric tube heat exchanger with dimensions of 1.3 m in length and 1.18 m effective tube length. **Table 1** shows the general dimensions of the heat exchanger.The heat exchanger was designed for counterflow configuration, in which one of the fluids passes through the shell side and the inner tube side, while the other fluid passes counter currently in the annulus between the concentric tubes and investigates an increase in the quantity of heat transferred as will be seen with a decrease in the length and mass of the heat exchanger compared with the conventional one.

2.2 Measuring Instrumentation

Temperature, pressure and flow rate were measured in the piping system, using thermocouples, pressure gauges, and flow meters have been used as the main measuring instruments.

The parameters to be measured during the test are:

1. The inlet and outlet temperatures of the tube side (inner and annulus) and the shell side.
2. The inlet and outlet pressures of the tube (inner and annulus) side and the shell side.
3. The flow rates of the tube side and the shell side.

2.3 Test Procedure

Before starting the experimental tests, several checks must be done to ensure that all testing components are well installed and ready for operation process. These checks include the following:

1- The main power supply.
2- The power supply for the electrical component (pump, heater).
3- Making sure filling the water tank and empty the water system from bubbles.
4- Making sure the level of oil in the tank above the heaters.
5- Checking up the fittings, joints, valves, and connections (no water leakage).
6- Checking up the valves before the pressure gauges opened and the valves after the reheater tank closed until the temperature of oil rise to the desired temperature.

After all these checking steps, the experimental procedure is:

1- Switching on the main power supply.
2- Switching on the oil heater and regulate it on the required inlet oil temperature (75, 80, 85, 90, 95, and 100 ºC). After reaching the required temperature, the oil pump is switched on the oil flow rate is regulated to the desired value (20, 40, and 60 l/min.) to enter to the inner tube side and the shell side in the heat exchanger.
3- Switching on the water pump at the same time of the hot oil pumping and regulating the water flow rate on the desired value (40 l/min) by using the gate valve to enter to the annulus side in the heat exchanger.

   After reaching the steady state condition. The following data are read and saved:
   - The cold water temperature and the hot oil temperature at the outlet by using the fitted thermocouples.
   - The cold water pressure and hot oil pressure at the inlet and outlet by using the connected pressure gages.

The above procedure is repeated with changing inlet temperature and flow rate for the hot oil.

3. ANALYSIS OF CFD

To simplify numerical simulation, certain assumptions are introduced such as steady-state conditions, no heat loss, no heat generation, no phase change, constant properties of fluids, no radiation effects, no process of mixing and the axial directions conduction is also neglected.

Heat exchanger geometry has been made in geometry module of Work Solid 2014. Flow in the heat exchanger is counterflow and consists of hot oil in inner tubes and shell and cold water in annuals between tubes. The heat exchanger has 16 tubes. In **Fig.2** and **3** simplified geometry is shown.

Then, by making a mesh, the modeling is continued, firstly partial coarse mesh is chosen with 9 million cells, number of nodes is 1 million, and number of elements is 2.5million **Fig.4a.** This mesh that includes a mixture of elements (quadrilateral and hexagon elements), has two faces of triangle and quadrilateral in boundaries. Geometry has been divided into several parts to use automatic methods available for mesh processor of software. Then, the mesh is installed with a good case in a mesh structure. That means a decrease in numerical distribution, especially near the wall area. Fine mesh is produced with 14 million cells, a number of nodes is 2.5 million, and the number of elements is 6 million, **Fig.4b.** For this mesh, edges and areas have meshed with velocity and high temperature as excellent (delicate and fine).

Contours have been analyzed according to coarse mesh and fine mesh and it should be noted that fine mesh in an area with pressure and gradient of high temperature is better solved in comparison with a coarse mesh. Therefore, to take care of these special areas, the coarse mesh has been adaptable to solve this gradient. Adaptability criterion is gradients of temperature and pressure. The velocity of a mixture of hot and cold fluids that are seen in the outlet, leads to more modification of mesh. Adaptability has been established based on gradients of temperature and pressure to achieve a completely independent network model for the mesh.

Mesh includes different types of cells, but 50% of them are hexagonal elements. Details of the ratio of components making mesh can be seen in Pi chart, **Fig. 5**.

Afterwards, it is seen that turbulence boundary conditions have a little effect on results and solutions. Walls have been specified in view of boundary conditions in separate. 'No slip' condition is considered for each wall. Except for walls of the tubes, any tube has been set with heat flux. Walls of tubes have been set a couple to heat transfer among fluids of crust direction and tube direction. Details about all boundary conditions can be seen in **Table 2.**

3.1 The Governing Equation

The governing equations of the problem to be studied are considered the starting point of any numerical simulation. For this case, incompressible turbulent flow is considered and the governing equations of fluid dynamics and their simplification are to be used. The conservation of mass, momentum, and energy can be used to derive the equations governing the motion of a fluid in addition to the transport equation for turbulent viscosity and its scale. The steady state is considered and the equations used are **Ansys Fluent User's Guide**:

- **The equation of continuity:**

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

- **The equation of momentum:**

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)\text{ in the x}-\text{dir} \tag{2}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)\text{ in the ydir.} \tag{3}$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)\text{ in the z-dir} \tag{4}$$

Energy equation:

$$\rho c_V \left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=K\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)+\emptyset \tag{5}$$

$$\emptyset=\mu\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+2\left(\frac{\partial w}{\partial z}\right)^2+\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2+\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)^2+\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2\right] \tag{6}$$

3.2 Mesh Generation

Fig 6 represents mesh generation. If a comparison is made between **Fig.6** and **Fig. 4a**, it can be observed that the number of cells in the shell are less than the number of cells inside the heat exchanger because there are no fluid flow and heat transfer through the shell of the heat exchanger, while inside the heat exchanger there are fluid flow and heat exchange so, the number of cells is more. Also, the number of cells inside the heat exchanger varies from place to place because of the difference of the flow velocity where in places of entry and exit of fluids there is a pool of fluid more than the rest of the places so the number of cells in these places is more than others. It is known that the increase in the number of cells as the flow becomes more clear and the results are more accurate.

4. ANALYSIS OF DATA

4.1 Comparison of CFD with Experimental Results

The RNG k-ε model was used with a different flow to be compared with the results that are obtained from experimental work.

The outlet temperature in direction of the shell and inner tubes have been shown in **Fig. 7**. The outlet temperature in the shell and inner tubes is almost 15-20 % more than the predicted value by the RNG k-ε model.

The outlet temperature in direction of the annulus between tubes has been shown in **Fig. 8.** The outlet temperature in the annulus is almost 5-9 % more than the predicted value by the RNG k-ε model.

This can be because of different reasons such as complex geometry in the shell and inner tubes direction and numerical distribution. This can be due to small baffles in used tubes in the laboratory one. **Table 3** shows these comparison.

4.2 Contour Plots

Drawing contour of distribution of temperature and velocity contours along heat exchanger can be seen in **Fig. 9** and **10**, respectively.

The heat exchanger is almost 1.3 m long. The velocity and temperature contours will give us an idea of details of flow all over the cross section in different situations along the length of the heat exchanger. To facilitate the work, drawings have been considered in 15 different situations.

The number of iteration is set to 106250 and the solution is calculated and various contours, vectors, and plots are obtained, **Fig.11**. The results are stabilized at iteration 101500, but attempts were continued to 106500 to ascertain the correctness of the solution and stability of the results. **Fig. 12** represents the component of the test rig.

5. CONCLUSIONS

CFD is used in analyzing and modeling real-world problem which is the design of shell-and-double concentric tube heat exchanger. The RNG $k - \varepsilon$ model was used and it can give a view of the distribution of flow and temperature. The profiles of temperature are determined all over the section of the surface. Also, it can be seen along the length of the heat exchanger and in different situations. Also, the distribution of velocity was determined.

It is concluded that the difference between the temperature of the experimental values and the simulated values are fairly agreed i.e. it is accepted. The maximum deviation between the experimental and numerical results is (+12.2%).

REFERENCE

- "*Ansys Fluent User's Guide*", 2015, Ansys Inc., South Pointe 275 Technology Drive Canonsburg.
- Basma, A. and Fadhil, A., 2015, *Shell and Double Concentric Tube Heat Exchanger Calculations and Analysis*, Journal of Engineering, Number 1, Volume 21 January, pp.62-82.
- Basma, A., and Noor, S., 2017, *Study and Analysis of Concentric Shell and Double Tube Heat Exchanger Using γ- Al2O3 Nanofluid*, Journal of Engineering, Number 9, Volume 23 September, pp.50-62.
- Quadir G. A, Irfan A. B., and Salman A. N.J., 2014, *Numerical investigation of the performance of a triple concentric pipe heat exchanger* International Journal of Heat and Mass Transfer, August, pp.165-172.
- Gurbir S., and Hemant K. 2014, *Computational Fluid Dynamics Analysis of Shell and Tube Heat Exchanger*, Journal of Civil Engineering and Environmental Technology, Volume 1, Number 3. pp. 66-70.
- Mahmoud G.Y., Ahmed A. A., Osama E. A., Essam E. K., 2014, *On The Computations of Thermal Behaviour of Shell and Tube Heat Exchanger*, AIAA Propulsion and Energy Forum and Expositions, July, Cleveland, Ohio, Cleveland Medical Mart & Convention Center.
- Mohammad R. S., Tooraj Y., and Mostafa K. M., 2014, *Numerical Analysis of Shell and Tube Heat Exchanger with Simple Baffle By CFD*, Indian Journal Scientific Research 7 (1): 1334-1345, ISSN: 0976-2876 (Print), ISSN: 2250-0138(Online).
- Mohan.K, Prakash.K, Sathya S.C 2016, *Efficiency Improvement in Shell and Tube Heat Exchanger Using CFD Tool,* International Journal of Innovative Research in Science, Engineering and Technology, Vol. 5, Issue 3, March 2016, ISSN(Online) : 2319-8753 ,ISSN (Print) : 2347-6710.

- Muhammad M. A., Nasir H., Muhammad H. B., et al., 2012, *CFD applications in various heat exchangers design: A review*, Applied Thermal Engineering, 32, pp. 1-2.
- Sinziana R., Ion O., and Irena L., 2012, *Heat Transfer Coefficient Solver for a Triple Concentric- Tube Heat Exchanger in Transition Regime* REV. CHIM. (Bucharest), 63, No. 8, August, pp.820-824.

NOMENCLATURE

T = temperature, °C
μ = the viscosity
cv = specific heat at constant volume , kJ/kg.°C
G_k = the generation of turbulence kinetic energy
K = thermal conductivity constant
P = pressure, kpa
S = the modulus of the mean rate-of-strain tensor
T = temperature
u = the velocity of fluid in the x-direction
w = the velocity of fluid in the z- direction
ν = the velocity of fluid in the y- direction
ρ = the density
∅ = the dissipation function

ABBREVIATIONS

LES = Large Eddy Simulation
RANS = Reynolds Average Navier- Stokes
CFD = computational fluid dynamics
RSM = Reynolds Stress Model

Table 1. Heat exchanger dimensions.

No	Description	Unit	Value
1	The Overall Dimensions	mm	1180*220*8
2	The diameter of shell	mm	203
3	Outer diameter of Tube1	mm	10
4	Inner diameter of Tube1	mm	6
5	Outer diameter of Tube2	mm	25
6	Inner diameter of Tube2	mm	20
7	Number of tubes	…..	16
8	Number of baffles	…..	10

Table 2. Boundary conditions.

	BC Type	Shell/ inner tube		Annulus tube
Inlet	Inlet velocity	0.155m/s	1.67m/s	0.83m/s
Outlet	Outlet pressure	0	0	0
Wall	No slip condition	No heat flux		Coupled
Temperature	Inlet temperature	373 k		293 k
Mass flow(kg/sec)		0.274	0.274	0.667

Table 3. Comparison of CFD with experimental results.

CFD result		Experimental result	
Temperature		Temperature	
Shell/tube	annulus	Shell/tube	Annulus
317	307	314	301

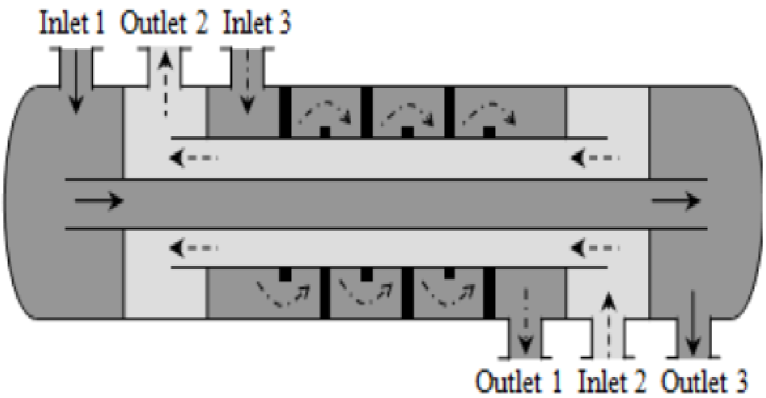


Figure 1. Diagram of shell-and-double concentric-tube heat exchanger.

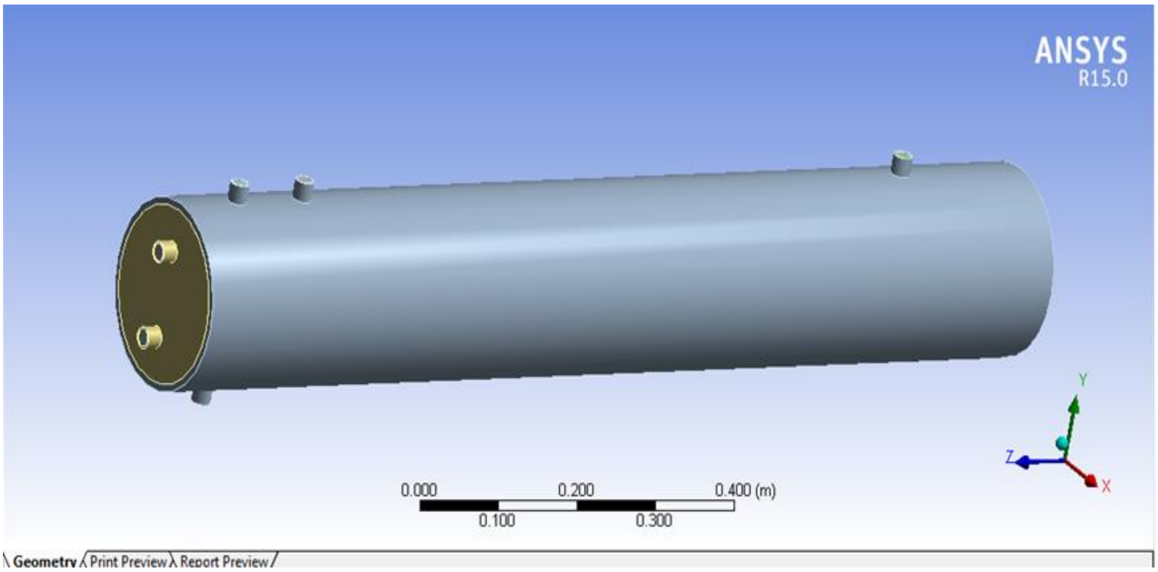
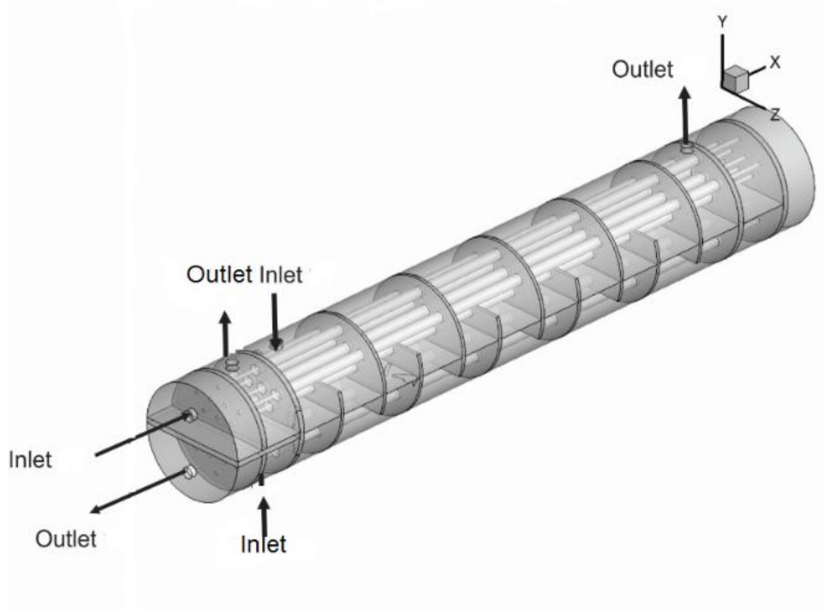
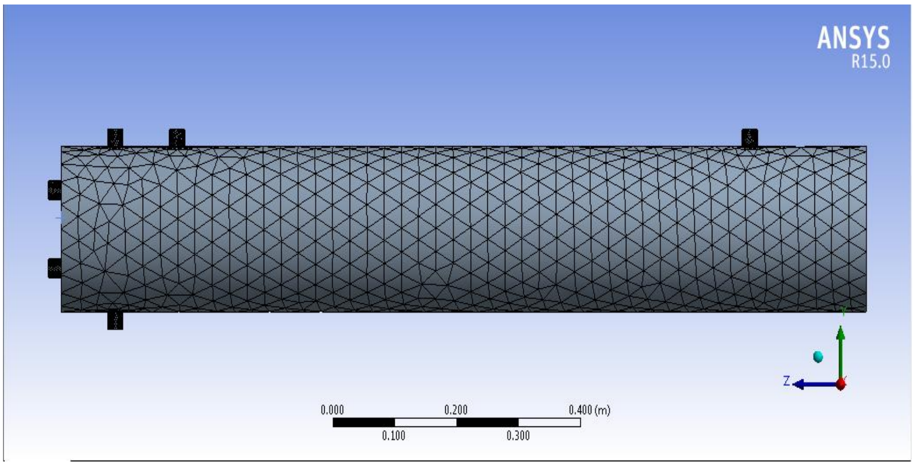


Figure 2. The outer part of geometry of the test section.

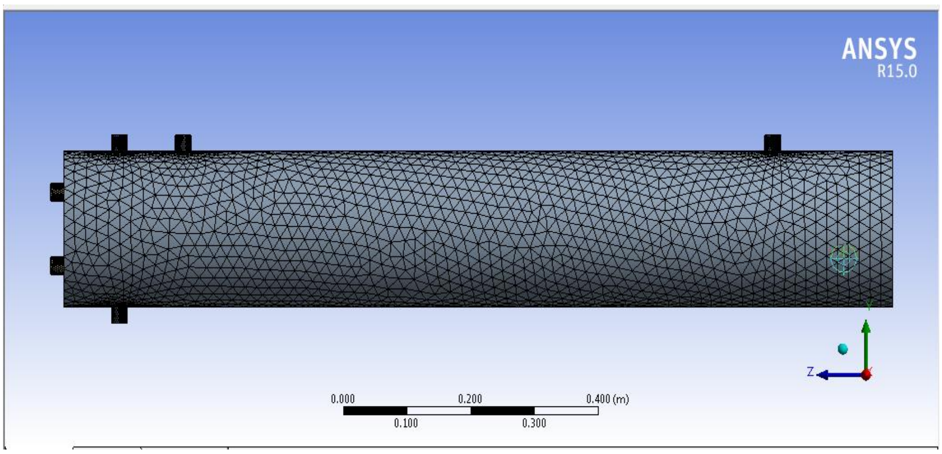


(b)

Figure 3. The inner part geometry of the test section.



(a) Coarse mesh



(b) Fine mesh

Figure 4 a, b. Mesh generation of the present work geometry.

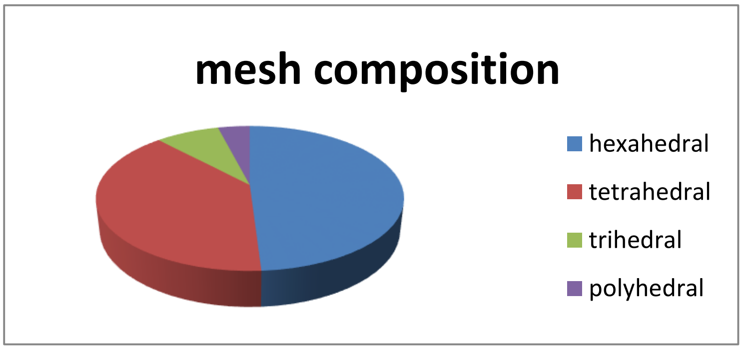


Figure 5. Mesh composition.

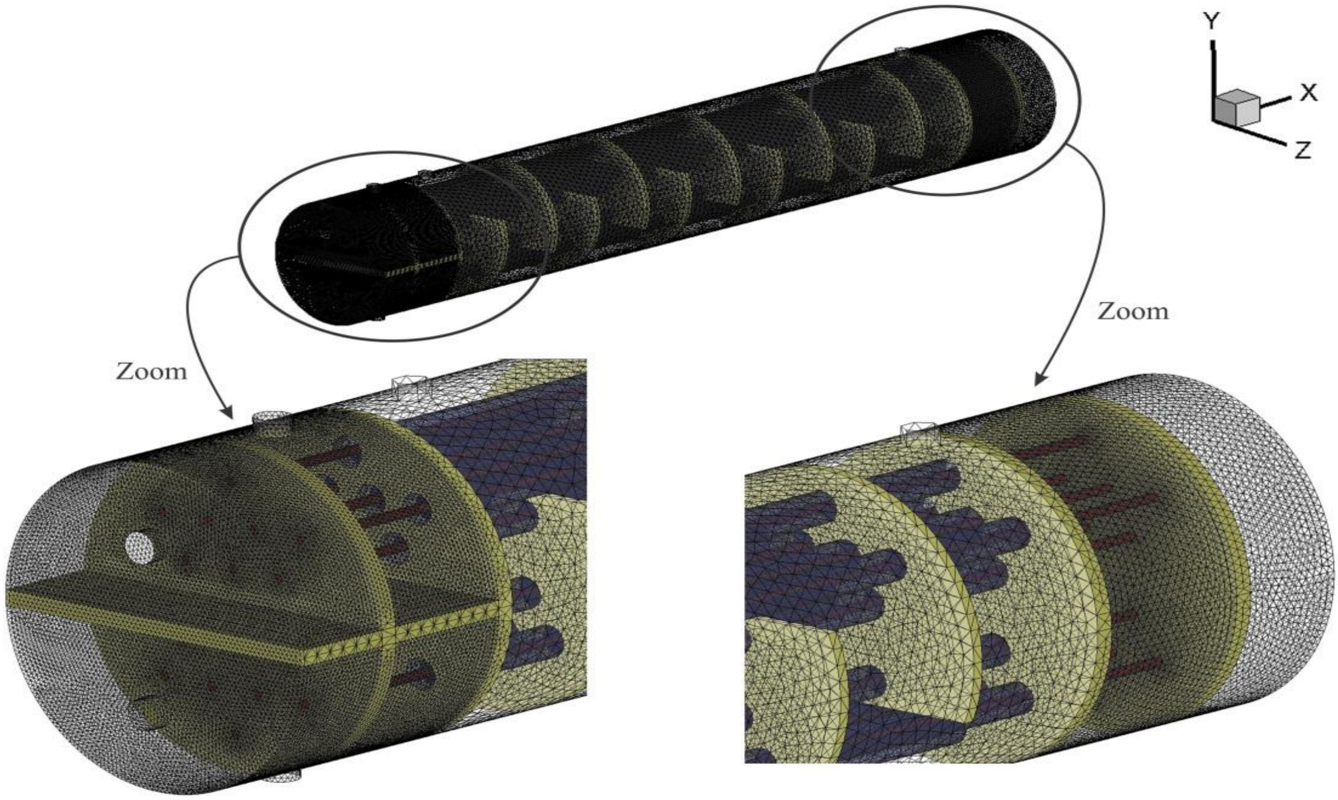


Figure 6. Mesh generation.

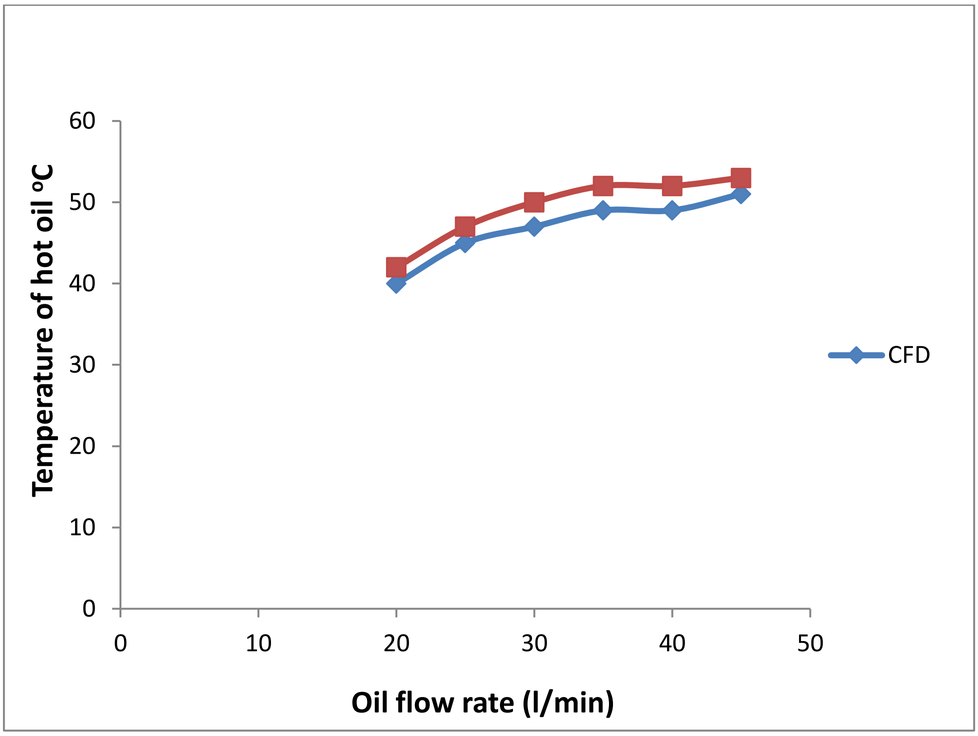


Figure 7. Comparison between experimental and CFD outlet temperature in shell and inner tube.

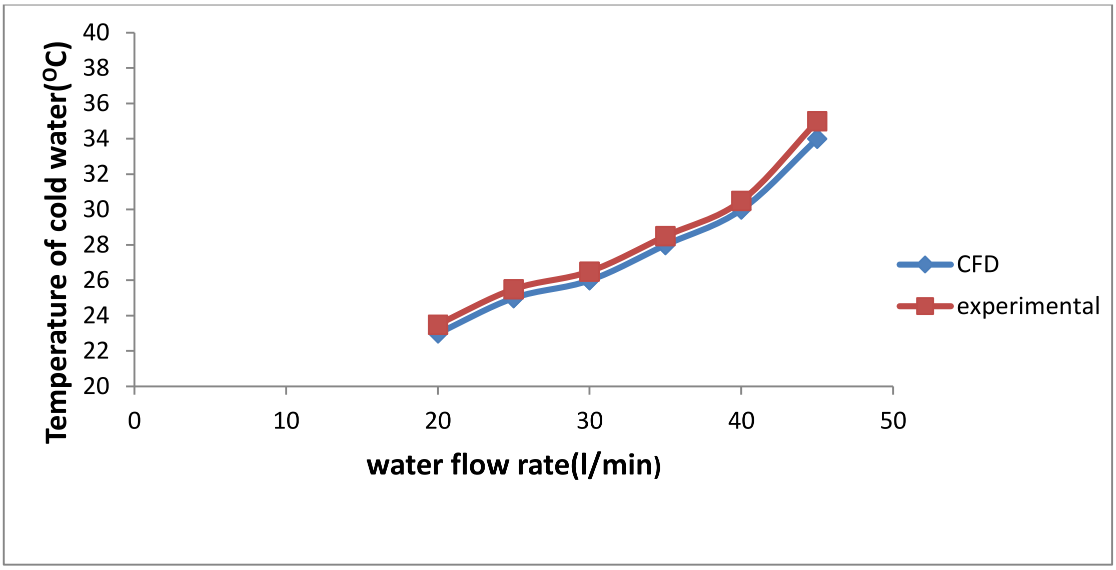


Figure 8. Comparison between experimental and CFD outlet temperature in annulus.

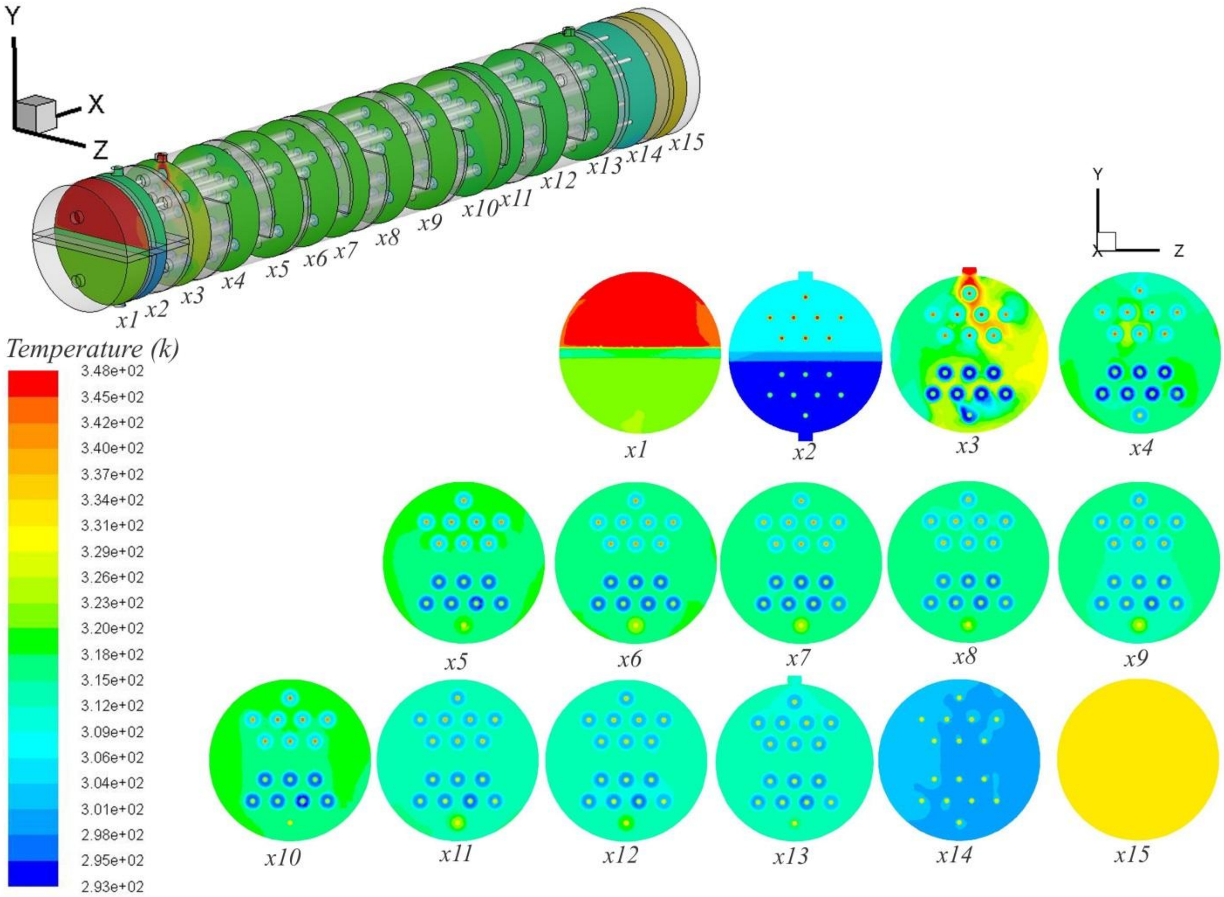


Figure 9. Drawing contour of temperature in symmetrical plane.

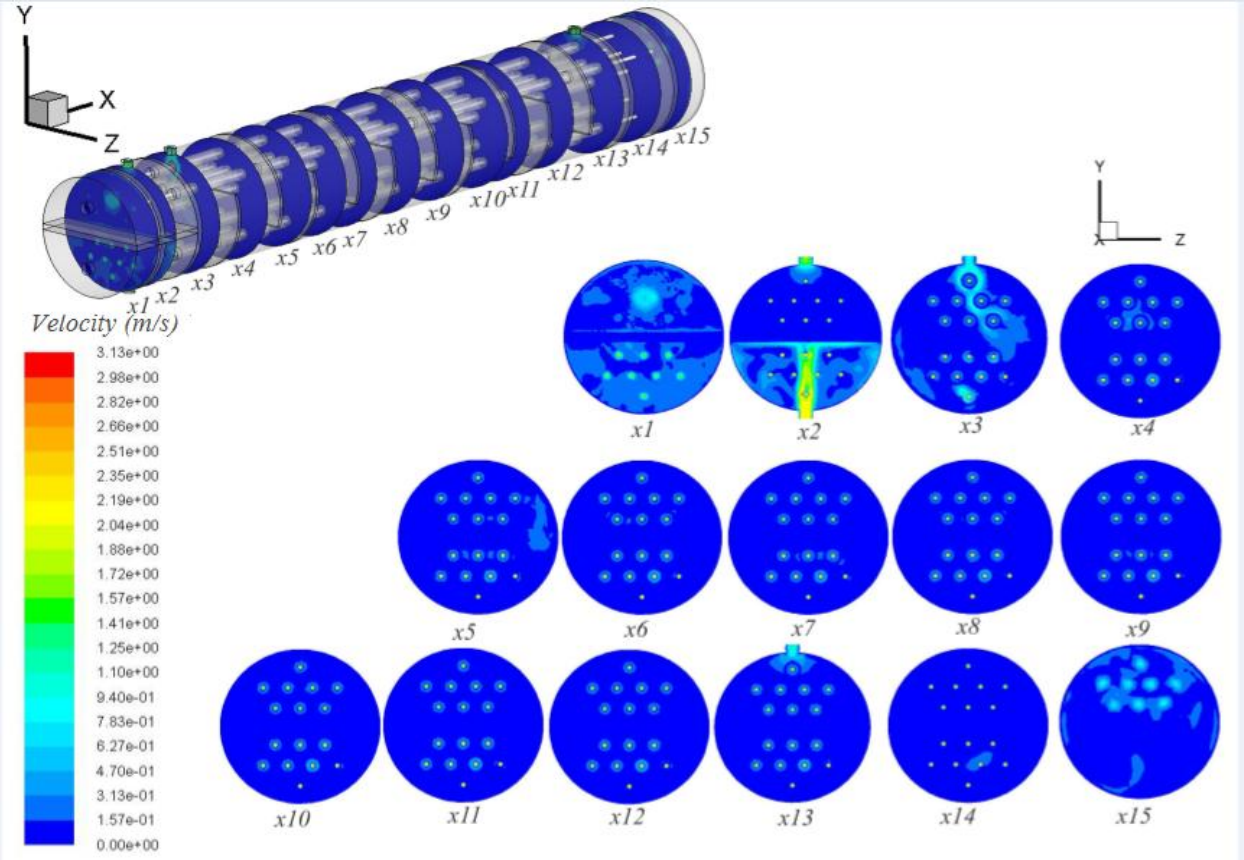


Figure 10. Drawing contour of velocity in symmetrical plane.

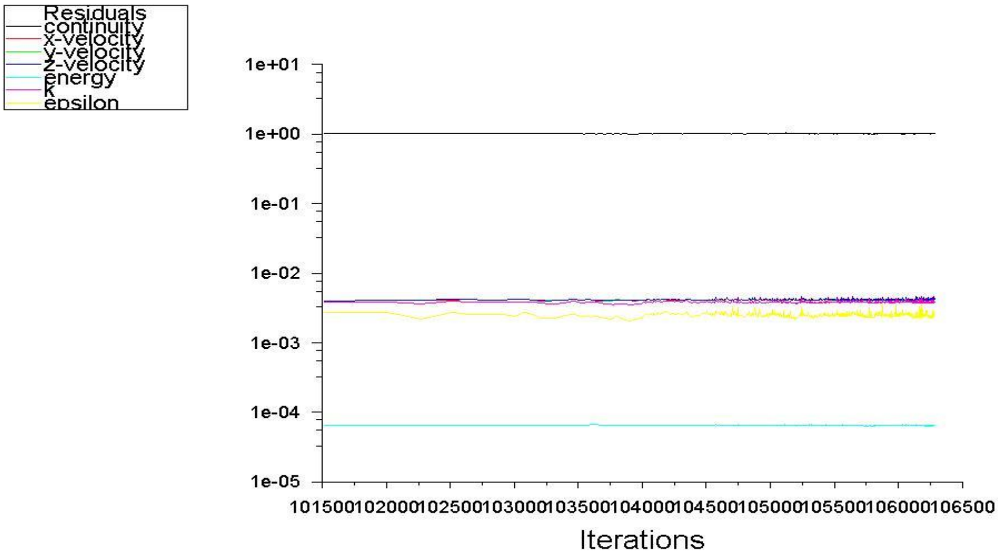


Figure 11. Residuals of numerical simulation.

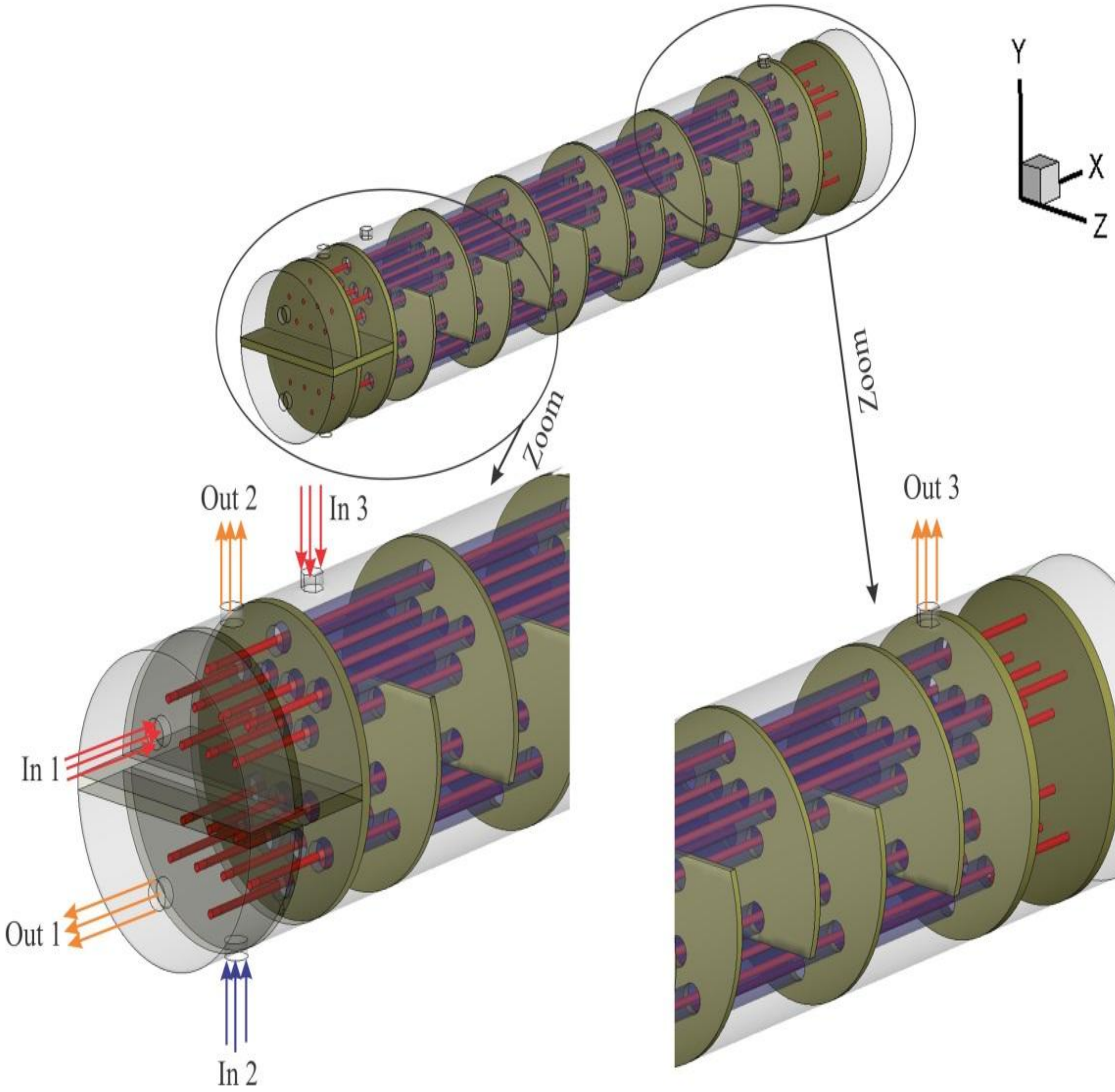


Figure 12. Component of test rig