

Numerical and experimental study of flow distribution in tubular space of fin-and-tube heat exchanger with modified inlet manifold

Tomasz Stelmach¹

¹ Institute of Thermal Power Engineering, Cracow University of Technology, Faculty of Mechanical Engineering

Abstract. This paper presents the experimental and numerical investigation of flow distribution in the tubular space of cross-flow fin-and-tube heat exchanger. The tube bundle with two rows arranged in staggered formation is considered. A modified heat exchanged manifold, with inlet nozzle pipe located asymmetrically is considered. The outlet nozzle pipe is located in the middle of the outlet manifold, with a standard shape. An experimental stand allows one to investigate the volumetric flow rate in heat exchanger tubular space using the ultrasonic flowmeters. Various inlet mass flow rate i.e. 3 m³/h, 4 m³/h and 5 m³/h are considered. The experimental results are compared with CFD simulation performed in ANSYS CFX program using the SSG Reynolds Stress turbulence model. A relatively good agreement is found for tube Re numbers varied from 1800 to 3100. .

1 Introduction

Fin-and-tube heat exchangers are widely used in chemical, petrochemical, refrigeration automotive and energy industry. This kind of heat exchangers operate in cross-flow arrangement of working fluids. In tubular space of heat exchanger flows the liquid with high heat transfer coefficient (water or oil), and in intertubular space flows gas with the low heat transfer coefficient. The extended heat transfer surfaces due to the application of fins (mostly rectangular or circular) allows to transfer a large amount of heat between working fluids. To achieve high thermal and hydraulic efficiency the elliptical tubes are used. The main advantage of elliptical tube is the favorable hydraulic performance compared to circular tube. Therefore, less pumping power is needed for gas flow through the tube bundle [1-4]. The fin-and-tube heat exchangers consists of inlet and outlet manifolds and the tube bundle (forming the tubular space of heat exchanger). The compact shape is achieved thanks to the application of heat exchanger manifolds with small volume [4,5,12]. Therefore, it is needed to ensure the uniform flow distribution from the inlet manifold to heat exchanger tubes, to allow the correct operation of the device [13,14]. If the flow maldistribution occurs, then in case of large heat fluxes transferred, the flow boiling [9,10] may occur. The flow boiling in heat exchanger tubes may intensify the fouling (scaling) processes [8,11], which leads to excessive thermal stresses in tube bundle. The part of tubes can work under the compressive stresses, which may result in tube buckling [12]. In this paper a flow distribution in heat exchanger with a modified shape of the inlet manifold is studied. The experimental stand described in [6,7] is used for

experimental investigation of water flow distribution in heat exchanger. Furthermore, the numerical simulations are performed using the SSG Reynolds stress model, to check the agreement between the experimental results and numerical simulation

2 Tested heat exchanger

The studied heat exchanger consists of tube bundle (6) connected with the inlet (5) and the outlet (8) manifolds. The tubes are arranged in staggered formation. The tube length is 2000 mm. The ultrasonic flow meters Sontex Superstatic 749 (7) are installed in each tube of the heat exchanger to capture the volumetric flow rate. The inlet and outlet nozzle pipes are connected with feed water tank (1) through the connecting tubes (4). The pump (2) is used to supply the water to heat exchanger under the specified volumetric flow rate. To control the flow rate the total flow rate meter (3) is used. The flow rate readings from total volumetric flow rate meter are then compared with the sum of individual tube flow rates obtained from the measurements, to check the correctness of tube flow meters indications. The measurements allow one to check if the flow maldistribution exists in heat exchanger, as well as to validate the results of CFD simulations. To ensure the correct inflow of the water from inlet nozzle pipe to heat exchanger inlet manifold, the extended inlet pipe with the length of 100 mm is used.

* Corresponding author: tomasz.stelmach@gkpgc.pl

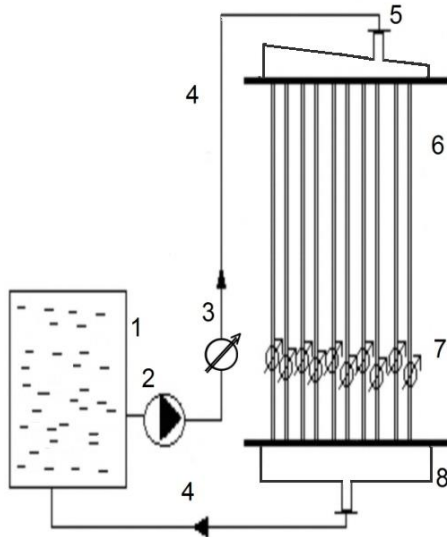


Fig. 1. Scheme of the tested heat exchanger tubular space of fin-and-tube heat exchanger: 1 - water feed tank, 2 - pump, 3 – total volumetric flow rate meter, 4 - connection pipes, 5 - modified inlet manifold with a nozzle pipe, 6 – tube bundle, 7 – single tube flow meter, 8 - outlet manifold and an outlet nozzle pipe.

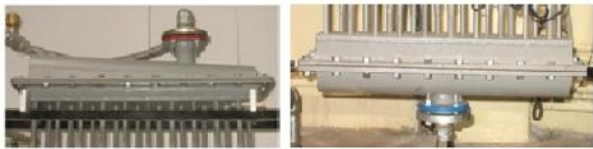


Fig. 2. Inlet and outlet manifold of the tested heat exchanger.

Figure 2 shows the inlet and outlet manifolds. The inlet manifold is equipped with the nozzle located asymmetrically, the volume of the manifold is extended by using the straight wedges. The outlet manifold is connected with the outlet nozzle pipe located in the middle of the manifold.

3 Volumetric flow rate distribution in heat exchanger tubes

The geometry of the heat exchanger inlet and outlet manifolds is presented in Fig. 3. The nozzle pipe of tested inlet manifold is located in a distance of 195 mm far from the right side of the inlet manifold, and 440 mm from the left side.

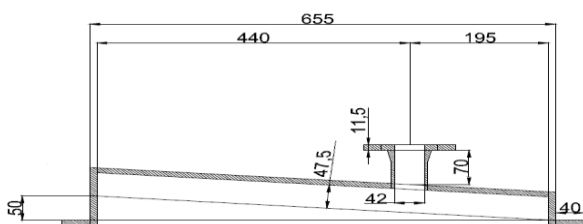


Fig. 3. Scheme of the inlet manifold with main dimensions.

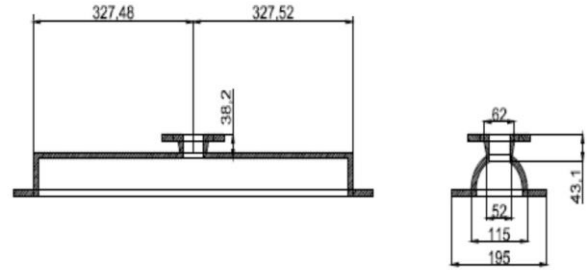


Fig. 4. Outlet manifold geometry.

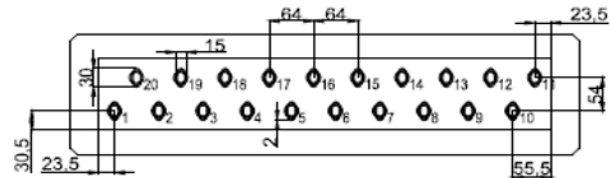


Fig. 5. Tube numbering and sieve plate geometry.

In this study, the measurements are carried out for the specified total volumetric flow rates equal to 3 m³/h, 4 m³/h, and 5 m³/h. That studied volumetric flow range leads to the transitional and turbulent flow regime in heat exchanger tubes. The results are presented for the tubes in the first row of tube bundle 1-10 and the second row of tube bundle 11-20 (Fig. 5). The water temperature is constant and set equal to 20 °C. The readings from flow meters are recorded after the steady-state flow distribution is reached.

Measurement no 1 – total volumetric flow rate of $\dot{V} \approx 3$ [m³/h]:

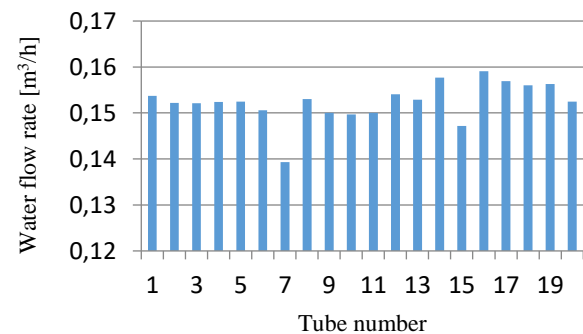


Fig. 6. Volumetric flow rate in heat exchanger tubes for a total flow rate of $\dot{V} = 3,0482$ [m³/h].

Table 1. The volumetric flow rate and Reynolds (Re) numbers in heat exchanger tubes for a total flow rate of $\dot{V} = 3,0482$ [m³/h].

Tube no.	1	2	3	4	5
Flow rate [m ³ /h]	0,1537	0,1522	0,1521	0,1524	0,1525
Re	1830	1812	1811	1814	1815
Tube no.	6	7	8	9	10
Flow rate [m ³ /h]	0,1506	0,1393	0,1530	0,1500	0,1497
Re	1793	1658	1821	1786	1782

Tube no.	11	12	13	14	15
Flow rate [m ³ /h]	0,1500	0,1541	0,1529	0,1577	0,1472
Re	1786	1835	1820	1877	1752
Tube no.	16	17	18	19	20
Flow rate [m ³ /h]	0,1591	0,1569	0,1560	0,1563	0,1525
Re	1894	1868	1857	1861	1815

Measurement no 2 – total volumetric flow rate of $\dot{V} \approx 4$ [m³/h]:

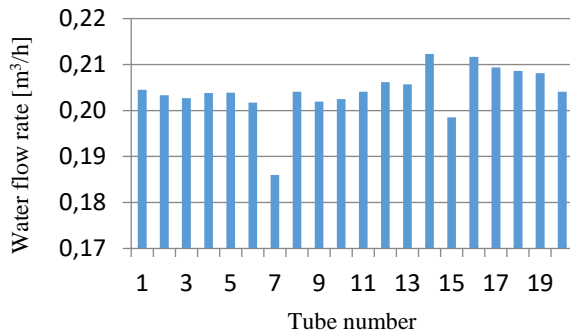


Fig. 7. Volumetric flow rate in heat exchanger tubes for a total flow rate of $\dot{V} = 4,0831$ [m³/h].

Table 2. The volumetric flow rate and Reynolds (Re) numbers in heat exchanger tubes for a total flow rate of $\dot{V} = 4,0831$ [m³/h].

Tube no.	1	2	3	4	5
Flow rate [m ³ /h]	0,2045	0,2033	0,2027	0,2038	0,2039
Re	2435	2420	2413	2426	2427
Tube no.	6	7	8	9	10
Flow rate [m ³ /h]	0,2017	0,1860	0,2041	0,2019	0,2025
Re	2401	2214	2430	2404	2411
Tube no.	11	12	13	14	15
Flow rate [m ³ /h]	0,2041	0,2062	0,2057	0,2123	0,1985
Re	2430	2455	2449	2527	2363
Tube no.	16	17	18	19	20
Flow rate [m ³ /h]	0,2117	0,2094	0,2086	0,2081	0,2041
Re	2520	2493	2483	2477	2430

Measurement no 3 – total volumetric flow rate of $\dot{V} \approx 5$ [m³/h]:

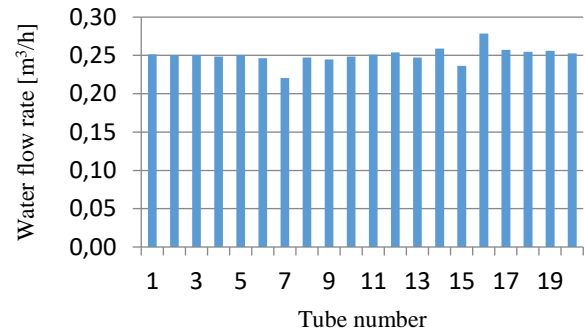


Fig. 8. Volumetric flow rate in heat exchanger tubes for a total flow rate of $\dot{V} = 5,0048$ [m³/h].

Table 3. The volumetric flow rate and Reynolds (Re) numbers in heat exchanger tubes for a total flow rate of $\dot{V} = 5,0048$ [m³/h].

Tube no.	1	2	3	4	5
Flow rate [m ³ /h]	0,2514	0,2501	0,2506	0,2483	0,2507
Re	2993	2977	2983	2956	2985
Tube no.	6	7	8	9	10
Flow rate [m ³ /h]	0,2465	0,2203	0,2474	0,2446	0,2484
Re	2935	2623	2945	2912	2957
Tube no.	11	12	13	14	15
Flow rate [m ³ /h]	0,2508	0,2540	0,2474	0,2589	0,2362
Re	2986	3024	2945	3082	2812
Tube no.	16	17	18	19	20
Flow rate [m ³ /h]	0,2786	0,2571	0,2548	0,2560	0,2527
Re	3317	3061	3033	3048	3008

4 Fluid flow simulation

CFD simulation are carried out to study the flow distribution in tubular space of the studied heat exchanger. The simulation is performed using the commercial code ANSYS CFX [15]. A geometrical model and a sample mesh is presented in Figure 9. The mesh consists of 1.3 million nodes and 4.47 million of element. The grid independency study is made and further refinement do not influence the obtained values of volumetric flow rate in heat exchanger tubes.

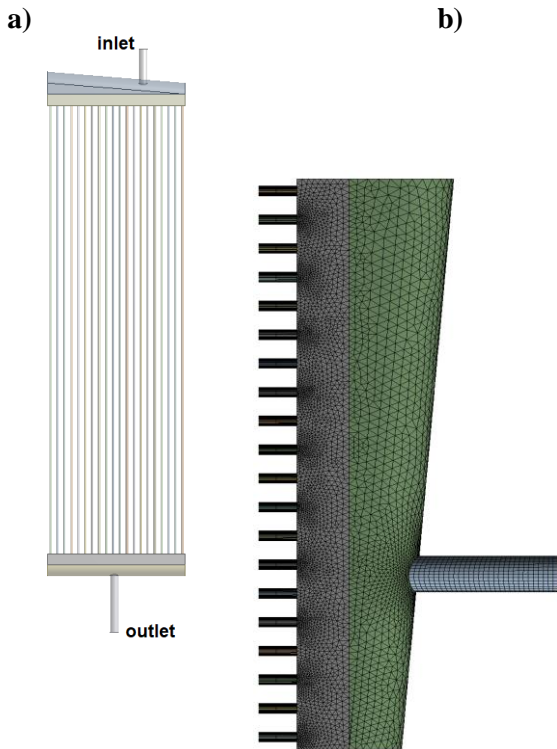


Fig. 9. Geometrical model of the heat exchanger (a) and CFD mesh (b).

As mentioned before the total volumetric flow rate is given based on the total volumetric flow meter indication at the location of inlet nozzle pipe. The inlet boundary condition in CFD simulation is applied at this region. The outflow boundary condition with a given relative pressure valued as 0 is specified at the location of outlet nozzle pipe. The SSG Reynolds Stress Transport turbulence model is employed for simulating the water flow in tubular space and manifolds of heat exchanger. This turbulence model predicts flow separation and secondary flow, therefore it is applied in this simulation. Figure 10 shows the streamline distribution at the location of inlet manifold and within the entire fluid flow domain.

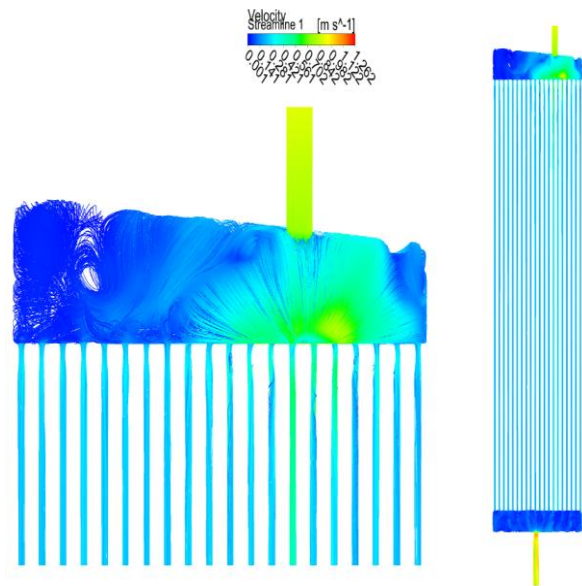


Fig. 10. Velocity streamlines obtained for total volumetric flow rate of $\dot{V} = 4,08 \text{ [m}^3/\text{h]}$.

From Figure 10 one can observe the highest flow velocity in tubes located below the inlet nozzle pipe. In other tubes the flow is undisturbed with rather uniform flow rate. The slightly higher value of flow velocity in tubes located downstream the inlet nozzle pipe. The comparison of volumetric flow rates in individual tubes is shown in Figures 11-13.

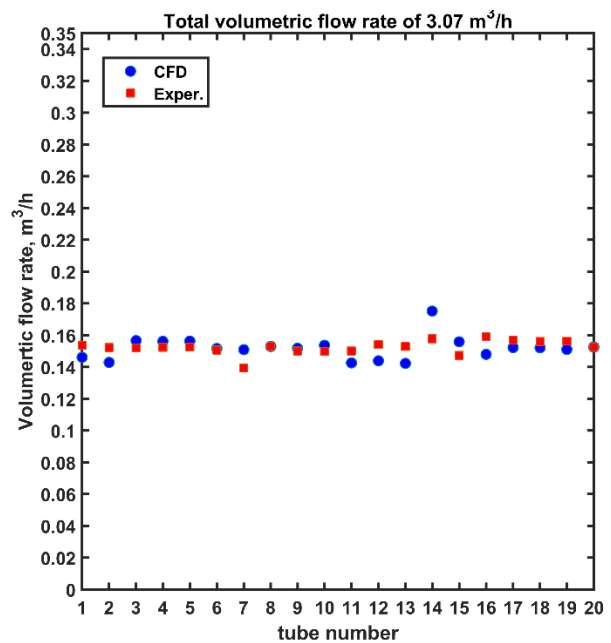


Fig. 11. Volumetric flow rate in heat exchanger tubes calculated using CFD and obtained from measurements for a total volumetric flow rate of $\dot{V} = 3,07 \text{ [m}^3/\text{h]}$.

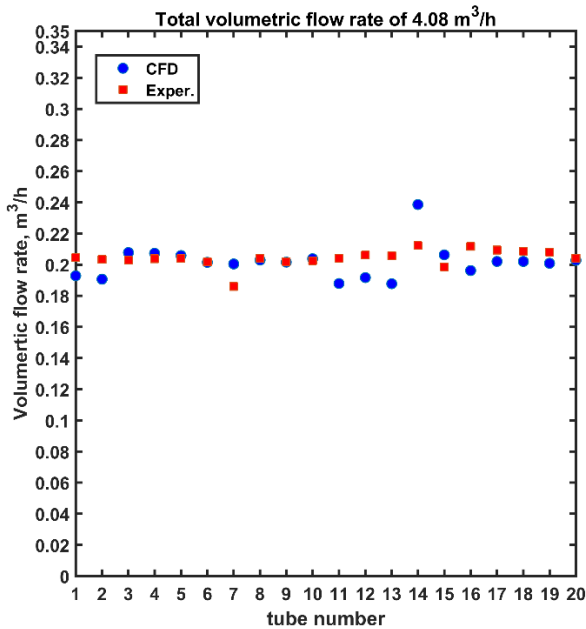


Fig. 12. Volumetric flow rate in heat exchanger tubes calculated using CFD and obtained from measurements for a total volumetric flow rate of $\dot{V} = 4,08$ [m³/h].

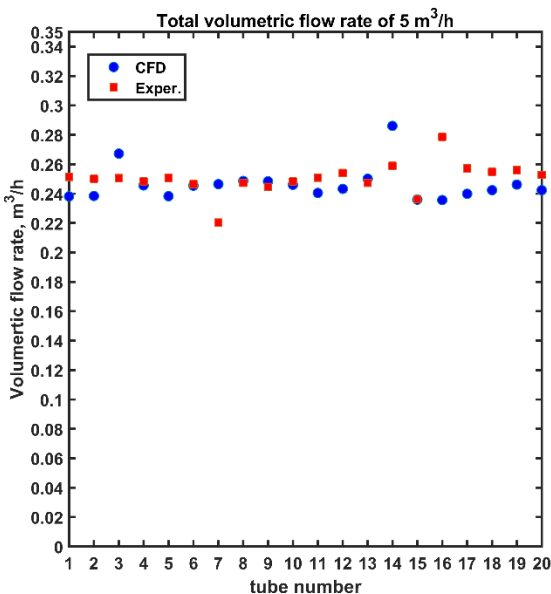


Fig. 13. Volumetric flow rate in heat exchanger tubes calculated using CFD and obtained from measurements for a total volumetric flow rate of $\dot{V} = 5$ [m³/h].

Figures 11-13 show both the measurements results and CFD simulation results obtained for the case when water flow is laminar and Re number is varied in a range of 1650 to 1900 in heat exchanger tubes (measurement point no. 1, total volumetric flow rate of 3 m³/h). The CFD simulation results and experimental results are rather close with slight differences observed in a region downstream to inlet nozzle pipe. In this region the applied turbulence model gives the higher value of flow rate in tube no. 14 (located directly below the inlet nozzle pipe), while the lower flow rates are calculated in tubes 12 and 13 nearby the inlet nozzle pipe. The results obtained for

measurement point no. 2 are similar. Higher discrepancies between the measurements and CFD simulation are observed for total volumetric flow rate of 5 m³/h where the measurements indicated the highest flow in tube no 16, while CFD simulation in tube no 14. The maximum percentage difference between measurement results and CFD simulation exceeds 15% in tube no 16.

5 Summary

In this study the CFD simulation are performed to study the flow distribution in a fin-and-tube cross-flow heat exchanger with modified inlet manifold. The tube bundle with staggered arrangement of two rows of tubes is studied. The modification of heat exchanger inlet manifold is done by extending the flow volume through adding the specially designed wedges. The CFD simulations employed SSG Reynolds Stress model that is able to predict the flow separation and secondary flows. The tube Reynolds numbers in a range 1650 – 3300 occurred for the water flow inside heat exchanger tubes. The CFD simulation results (volumetric flow rate in heat exchanger tube) are compared with measurements results. The measurements are made by using the Sontex Superstatic 749 ultrasonic flowmeters installed in each tube of heat exchanger, For Re numbers close to 2000 a good agreement between the measurement results and CFD simulation is found. Higher discrepancies are obtained for tube Re number over 3000, where the CFD simulations are not able to correctly predict the tube location, where the flow is the highest. In this particular case, for total inlet volumetric flow rate of 5 m³/h the differences between CFD simulation results and measurement results exceeds 15%.

Acknowledgements

The author would like to acknowledge the Polish National Science Center for a financial support. The computational results presented in this paper are the part of the research grant OPUS 6. The project was financed by the Polish National Science Centre awarded based on the decision number DEC-2013/11/B/ST8/00340.

References

1. Łopata S., Ocloń P. Archives of thermodynamics **31**, 3, pp. 37-53 (2010)
2. Ocloń P., Łopata S. Technical Transactions. Mechanics **4-M/2011/B**, 7, year 108, pp. 391-400 (2011)
3. Ocloń P. (PhD Thesis, Kraków, (2013)
4. Łopata S., Ocloń P., (Fluid Dynamics, Computational Modeling and Applications, edited by L. Hector Juarez, part 3, chapter **14**, InTech, ISBN: 978-953-51-0052-2, pp. 327 – 356, Croatia 2012)
5. Łopata S., Ocloń P., Rynek Energii **102**, 5, pp. 112-124 (2012)
6. Łopata, S., Ocloń, P., Nowak-Ocloń, M. E3S Web of Conferences, **13**, art. no. 02001 (2017)

7. Ocloń, P., Łopata, S., Chłosta, K. *Procedia Engineering*, **157**, pp. 428-435 (2016)
8. Łopata, S., Ocloń, P. *Energy*, **92**, pp. 100-116. (2015)
9. Ocloń, P., Łopata, S., Nowak, M. *Heat and Mass Transfer/Waerme- und Stoffuebertragung*, **51** (4), pp. 553-566. (2015)
10. Ocloń, P., Nowak, M., Łopata, S. *Journal of Thermal Science*, **23** (2), art. no. 1003-2169(2014)02-0177-10, pp. 177-186. (2014)
11. Ocloń, P., Łopata, S., Nowak, M., Benim, A.C. *Progress in Computational Fluid Dynamics*, **15** (5), pp. 290-306. (2015)
12. Ocloń, P., Łopata, S. *Heat Transfer Engineering*, **39** (13-14), pp. 1139-1155. (2018)
13. Taler, D., Ocloń, P. *International Journal of Thermal Sciences*, **84**, pp. 309-322. (2014)
14. Taler, D., Ocloń, P. *Chemical Engineering and Processing: Process Intensification*, **83**, pp. 1-11. (2014)
15. ANSYS Inc, *Ansys CFX Theory Reference Guide*, (2016)