

## Computational Fluid Dynamic analysis of Screw tube relevant for fusion applications

P. Domalapally<sup>1</sup>

<sup>1</sup> Research Centre Rez, Rez, Czech Republic

*E-mail contact of main author: P\_Kumar.Domalapally@cvrez.cz*

**Abstract.** Determination of the likely heat loads which may be expected on the First Wall (FW) of the European DEMO is still underway. This uncertainty notwithstanding the engineering design of the heat sink components must proceed, hence the scientific community is using the so called bottom-up approach to determine the maximum heat flux that the component could sustain given currently existing material limitations and forecast operating conditions. The current work attempts to study the heat absorption capability of the heat sink using a turbulence/critical heat flux enhancer inside the cooling channel known as a screw tube, by using Computational Fluid Dynamics (CFD). A screw tube is a cooling tube with a helical triangular fin on its inner surface. The nut-like inner surface works as a combination of enhanced heat transfer area and turbulence promoter. In the literature, several experiments have been performed to determine the heat evacuation capability improvement from screw tubes but none has studied the fluid dynamics and heat transfer in detail. In this work, a commercial CFD code ANSYS-FLUENT is used to study the flow physics. In this aspect, several turbulence models were tested and the effect on flow dynamics is evaluated. In the next step, the most appropriate turbulence model will be selected. Thus the current work lays the foundations for further thermal hydraulic optimization of the screw tube to be performed taking into account the European DEMO conditions.

### 1. Introduction

To account for the future energy needs, fusion energy is one of the most attractive alternatives among the several technologies that were proposed in the recent past. Out of numerous ways of harnessing fusion energy, the TOKAMAK concept of magnetic confinement is the most economical. International Thermonuclear Experimental Reactor (ITER) is an experimental reactor which is being built in Cadarache, France [1], to demonstrate that fusion yields more energy than the invested one. The next step to ITER, before the fully commercialization as a nuclear power plant, is called DEMOnstration reactor (DEMO) [2]. Main concern in designing a working and efficient nuclear fusion machine is protecting and preserving all the components, which have to withstand the severe operating conditions.

The core temperature of the plasma during the operation of the machine would be around 150 MK [1]. Though the plasma is confined by magnetic field, the first component that will face plasma has to withstand extreme particle, neutron and radiation loads. In ITER the heat loads are already defined and depending on the location inside the reactor these Plasma Facing Components (PFCs) will receive heat fluxes in the range of 1 up to several MW/m<sup>2</sup> [3, 4]. The highest heat load will act on the vertical target of the divertor of the fusion reactor which is estimated to be around 20 MW/m<sup>2</sup> [5]. In order to withstand these loads in ITER the heat sinks are designed with CuCrZr as the heat sink material, using water as the coolant at 70 °C and 4 MPa as operating conditions [6]. On the contrary, the European DEMO most likely will use Reduced Activation Ferritic Martensitic steel (RAFM) called Eurofer97 as both heat sink and structural material for the FW, where one concept uses water as a coolant at 15.5 MPa and 285 °C. While for divertor CuCrZr is used as the heat sink material, with water flowing at

16 m/s at 150 °C and 5 MPa. At higher heat fluxes it is possible that the coolant will undergo phase change, which actually increases the Heat Transfer Coefficient (HTC) to tremendous values. At the same time it is dangerous as there is an upper limit of operation whenever we have boiling, which is termed as Critical Heat Flux (CHF). These limits makes it essential to optimize the thermo-hydraulic design, for the safe operation of the reactor minimizing the PFCs temperature and pumping power requirement while extending the life. If simple channels/tubes are used then very high velocities and pressures will be required at high heat fluxes, which may not be practical so turbulence enhancers must be used. In the case of water the usage of turbulence enhancers also increases the CHF limit which allows the component to operate in safe region with extended safety margin.

In previous studies, different CHF and turbulence enhancers for water as a coolant were proposed such as porous media, jet cooling, swirl tube, annular flow tubes hypervapotron and screw tube [7]. When operated in subcooled boiling regime water provides very high HTC; details about the subcooled boiling and boiling physics can be found in [8]. Among the above mentioned concepts the screw tube is one of the most attractive options. This concept could be optimal thermo-hydraulically as well as thermo-mechanically, which is very important for the design of the heat sink [9]. Some experiments using this geometry were conducted at JAERI [10-12]. None of the experiments were performed in detail to study the flow physics and the CHF enhancement as it was done for swirl tube and hypervapotron [13]. It is planned to perform in the coming future detailed experiments to study the flow and boiling physics with this concept using different materials and operating conditions. Before reaching that stage it is sought out to initially study the flow physics and then to perform optimization of its geometry. As a first step, the flow physics within the channel without any heat flux acting on it with different turbulence models will be studied. As a second step, a proper turbulence model, which is suitable for this application and also computationally economical, will be selected based on single phase heat transfer calculations. In the next step using the available experimental data heat fluxes, which can induce boiling inside the channel, will be modelled using transition boiling model as in STAR-CCM+ [14, 15]. Final part of the optimization will be performed for different heat fluxes and flow conditions using the selected turbulence model and fine-tuned parameters of the boiling model.

## 2. Literature review

In the past, several authors have worked on screw tube: especially they did some experimental work to check CHF limits. This section overviews the work that has been carried out.

One of the first articles to mention screw tube is by Smid et al. [9]. In the article the authors tested and compared different types of mockups (smooth, swirl tapes and hypervapotrons) with the same width using FE 200 test facility of CEA for performing the experiments. The authors concluded their article by stating that hypervapotron had better thermal hydraulic performance than swirl tube but swirl tube had better thermomechanical performance. In order to gain the advantages of both, the definition of the best concept could be a combination of the two: a circular channel with helical fins which is nothing but a screw tube.

Boscary et al. [16] carried out CHF experiments on swirl, screw and hypervapotron tubes, which are the most efficient geometries to remove high incident heat fluxes. From the tests conducted both at CEA and JAERI, it was found that an IncidentCHF (ICHF) of 41.6 MW/m<sup>2</sup> was obtained for M7 screw tube, at  $P_{local} = 1$  MPa,  $V_{axial} = 20$  m/s,  $\Delta T_{sub,local} = 136^{\circ}\text{C}$ .

The article by Raffray et al. [17] describes screw tube as a tube whose inner surface is machined like that of a nut. The nut-like inner surface can work as a combination of fin effect provider and turbulence promoter at the surface to enhance heat transfer. It was mentioned

that application of this tube for fusion devices was proposed by Araki et al. [18]. It was mentioned in the article that at the velocity of 10 m/s, the screw tube gave an ICHF of 46 MW/m<sup>2</sup>, which is more than twice that of the smooth tube, and 1.5 times higher than that of the swirl tube. Based on these results, it was concluded that the screw tube was an attractive CHF enhancement concept certainly warranting further studies. However, in view of its limited CHF experimental data base and concerns about the screw geometry potentially acting as a crack initiator, it is not presently considered as a reference concept for the ITER divertor.

The article by Masaki et al. [10] presents the results of the high heat load tests on screw tube to evaluate the HTC. It was mentioned that the results of the CHF experiments showed that the HTC of the screw tube at the non-boiling region was roughly three times higher than that of the smooth tube, i.e. 1.5 times that of the swirl tube.

The article by Ezato et al. [19], presents the results of thermal fatigue experiments on a screw tube. From the experiments, it was found that heat removal performance of such a screw tube was twice higher than that of a smooth tube. The author also did numerical simulations along with the experiments using finite element method code ABACUS and it was found that the numerical model correctly predicted thermal response of the test sample. The paper concluded by stating that the thermomechanical analyses and fractographic observations revealed that fatigue cracks started from the heated outer side and propagated toward the inner surface. This indicates that the screw geometry does not act as a crack initiator under the one-sided heated condition with high heat flux appearing in fusion machines. In another article [20] by the same authors they tried to examine heat removal capability of the screw tube made of F82H (Reduced Activation Ferritic Martensitic steel) and compared it with that of OFHC-Cu. From the experiments it was found that using the F82H screw tube, ICHF of 13 MW/m<sup>2</sup> was obtained at the axial flow velocity of 4 m/s, which was about half value of the OFHC-Cu tube at the same flow velocity. In their next article [21] the authors investigated the heat removal limits at higher temperature and pressure range than those at the previous campaign, on screw tube. The experimental results indicated that the effect of the screw fine on enhancement of heat removal performance such as mixing or continuous separation of the coolant near wall region could be effective at the higher coolant temperature conditions up to 100 °C.

Suzuki et al. [12] reported CHF experiments with screw tube made of F82H. The authors reported similar results as Ezato et al. in [20], but the authors were again concerned that the screw threads may cause fatigue cracking due to stress concentration at the tip of the threads so they also performed thermal fatigue tests. After post processing using an optical microscope the results indicated that the stress concentration at the tip of the screw thread had little effect on the fatigue crack initiation from the cooling tube. Strain amplitude at the outer surface of the cooling tube mostly affects the fatigue crack initiation.

The article by Ezato et al. [22] presents the CHF tests carried out to examine its heat removal capability at higher cooling water temperatures as compared with the previous experiments, with DEMO relevant cooling conditions. Although the ICHF of the screw tube decreases by about 50% with an increase in  $T_{\text{sub}}$  by 140 K, it is remarkable that this CHF value remains almost twice as high as that of the smooth tube under the same cooling conditions.

From the literature survey it can be observed that all the authors mention that screw tube is having very high potential as a heat sink device and it can have ~1.5 times higher CHF than that of swirl tube. Though none of the authors investigated the physical mechanism causing the increase in CHF and HTC. Also none of the authors tried to do flow analysis in order to understand the flow physics inside the screw tube channel. It is what being attempted here to understand the flow physics and then do further analysis to understand the heat transfer mechanism.

### 3. Geometry and flow conditions

Figure 1 shows the basic geometry of the screw tube, as explained before. It is a tube whose inner surface is machined like that of a nut. The main advantages are: 1) It can be fabricated at low cost by simple mechanical tapping. 2) The nut-like inner surface can work as a combination of fin effect provider and turbulence promoter at the surface to enhance heat transfer.



FIG. 1. Geometry of screw tube showing internal thread.

The geometry chosen for the current analysis comes from the experimental work performed by Masaki et al. [10]. This data has inlet conditions with 3 different velocities at 3 different pressures but with constant inlet temperature which will give an opportunity to concentrate on fluid dynamics of the problem. Unfortunately none of the data that is available in the literature have detailed experiments conducted on fluid dynamics of screw tube as per the author's knowledge. The details of the geometry of the thread for the current analysis are presented in figure 2. The minimal inside diameter and the maximal inside diameter of the screw fin are 9.02 and 10.1 mm, respectively. The fin pitch is 1.5 mm. These are defined by ISO 261.

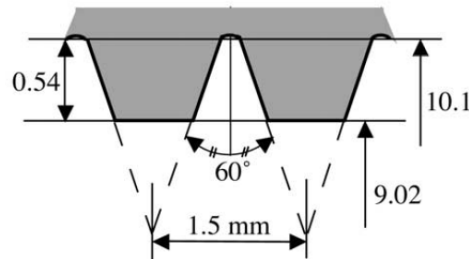


FIG. 2. Geometry of the inner surface of the screw tube in [10].

Axial velocities of the cooling water were of 4.0, 5.6 and 8.0 m/s, at local pressures of 0.93, 0.88 and 0.74 MPa respectively. The temperature of the inlet water was  $\sim 25^{\circ}\text{C}$ . It was mentioned in the article [10] that the axial velocity of the cooling water was defined from a flow rate measured at outlet and the maximal inside diameter of the screw tube. The original geometry was made as a flat tile mockup with carbon fiber composite as armour material which was brazed on to a Cu alloy (CuCrZr), with 1 mm thick Cu interlayer. The screw tube was directly machined into CuCrZr tube in one stroke. In the present analysis none of the materials is considered, only a fluid volume created by the tube is considered. The total length of the test section in the original setup was 327 mm, which is kept in the present analysis.

### 4. Modelling

As mentioned in the introduction chapter, the main objective of the present analysis is to perform CFD analysis of a screw tube to study the flow field using different turbulence models. In order to perform the numerical analysis, ANSYS-FLUENT 16.1 [23] is chosen. Using the turbulence models the pressure drop and flow behavior are reported in the current analysis. The results obtained from the analysis are then compared against each other. The turbulence models that are used and compared are: 1. Standard k-epsilon, 2. RNG k-epsilon,

3. Realizable k-epsilon, 4. Standard k-omega, 5. BSL k-omega, 6. SST k-omega, 7. Spallart-Allmaras, 8.SAS, 9. RSM, 10. DES- Realizable k-epsilon, 11. DES - Spallart-Allmaras, 12. DES- SST. Details about these models can be found in [24, 25]. All the models that are used here are run until steady state is reached; all the models are based on the Navier–Stokes equations. Enhanced wall function to handle  $y^+ \sim 1$  are used in the models where  $y^+$  is defined in [25]. Pressure based segregated solver is used for performing the simulations; gravity effect is included. In order to account for fully developed turbulence at the entrance of the screw tube,  $\sim 20$  diameters of the normal sectioned tube is considered with the same inner diameter as that of screw tube. Before performing the simulations it is important and advisable to carry out grid sensitivity studies to establish proper grid size for the analysis. For this, all the k-epsilon models are chosen. For the analysis polyhedral mesh is used as shown in figure 3. The flow conditions used for the convergence study are inlet velocity of 5.6 m/s and pressure of 0.88 MPa.



FIG. 3. Grid used for the analysis of the screw tube.

While performing the grid independence analysis the pressure drop, surface friction coefficient in the channel is closely monitored. The number of elements in the screw region is kept maximum compared to the number of elements in the non-screw region. After performing the grid independent analysis,  $\sim 8$  million elements proved to be sufficient.

## 5. Results and discussion

After applying proper solver settings and boundary conditions, simulations are run using the optimized grid for different turbulence models. The simulations were run using three different velocities as explained in section 3.

Figure 4 shows the variation of pressure drop for different turbulence models, where the number on the x-axis denotes the turbulence model in the order described in section 3. Figure 4 contains data for all the three velocities and each figure has two plots where one of them shows pressure drop data for smooth tube region before the screw tube and the other pressure drop in the screw region. In each figure, for screw tube region the mean value of pressure from all the models is indicated whereas for smooth tubes the value of pressure drop estimated from the Darcy-Weisbach equation is given. For smooth tube region the error from each model with respect to the analytical value estimated differs within  $\pm 15\%$ . The lowest errors are reported in all the cases for the BSL and SST k-omega models. As the flow in the smooth pipe region is still not fully developed the exact value of the pressured drop is difficult to estimate which has to be found by only using experimental data. It is difficult to say anything at this moment about the screw tube region as unfortunately the experimental data is not available. Nevertheless the author is performing further heat transfer simulations and the simulated data will be compared against the available experimental data and further conclusions will be drawn. Once this is performed the best model to predict the heat transfer behavior will be chosen for further analysis of the screw tube. In principle, all the turbulence models give different values of the pressure drop and this difference is higher for higher velocities.

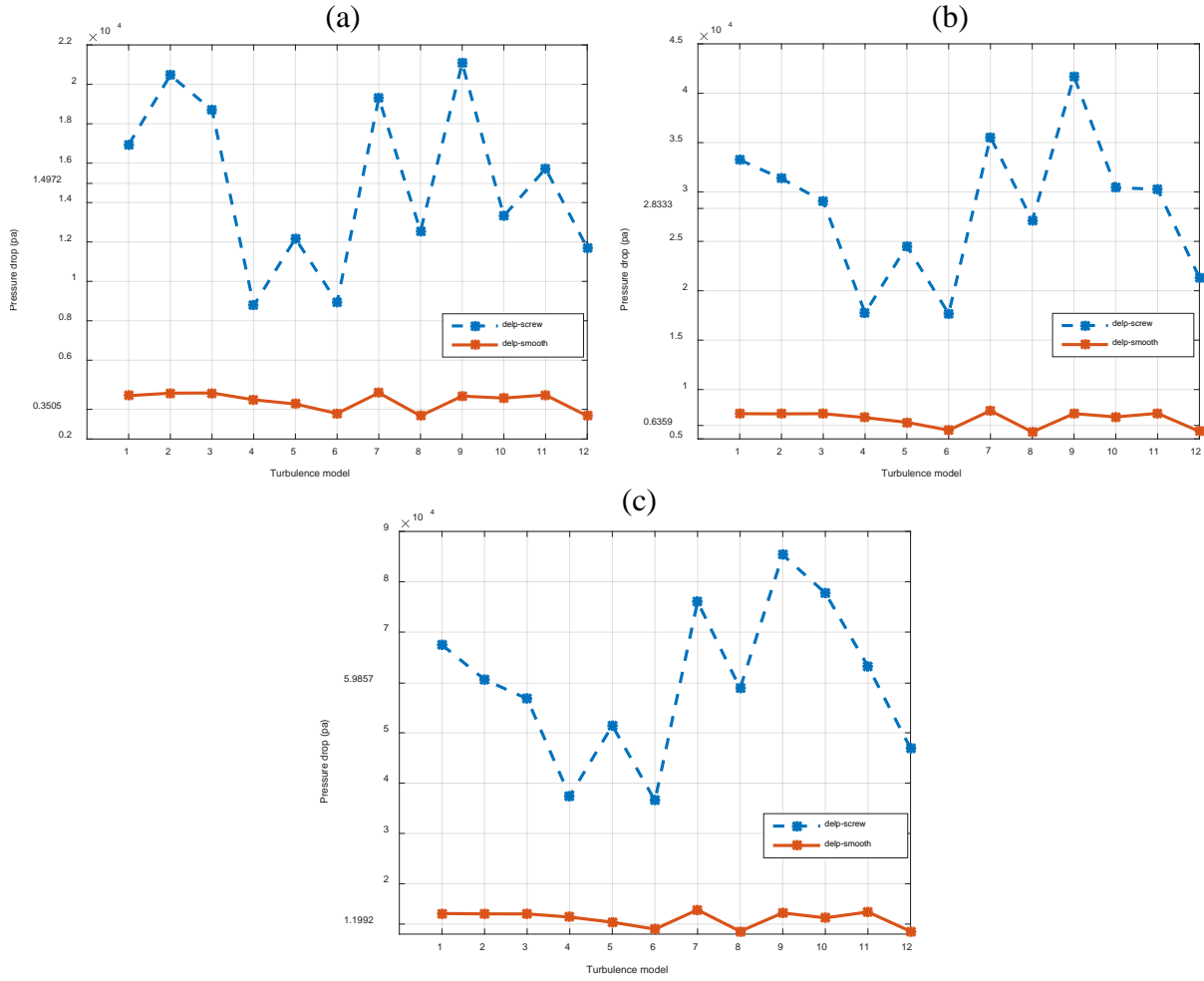


FIG. 4. Variation of the pressure drop using different turbulence models with (a) 4 m/s, 0.93 MPa, (b) 5.6 m/s, 0.88 MPa, and (c) 8 m/s, 0.74 MPa, Inlet velocity and pressures respectively.

Figure 5 shows the variation of helicity and normalized helicity at the beginning and at the end of screw tube using different turbulence models. Helicity is a scalar quantity, defined as the dot product of velocity and vorticity vectors. It is expected that the screw shape of the pipe will cause the flow to be swirled/helical inside the pipe. So helicity is one of the important factors that can be checked to estimate the magnitude of rotation of the fluid as well as the normalized helicity can be used to estimate the angle between velocity vector and the vorticity vector. Higher the value of the helicity higher the vorticity/higher rotation. The sign of the normalized helicity determines the direction of the swirl of the vortex relative to the stream wise velocity component.

Figure 5 shows data from two velocities and each figure has two subplots and each plot contains the variation of helicity and normalized helicity before and after the screw tube (top and bottom plots respectively). It is clear that higher the velocity higher is the helicity and so the angle between the vortexes relative to the stream wise velocity component. Normalized helicity turned from negative to positive as it passed through the swirl indicating that the screw is generating a lot of turbulence inside the tube. Figure 6 shows the velocity vectors in the screw tube at the entrance, in the middle, 0.01 m before the exit of screw tube for different velocities using SST k-omega model, it can be observed from the figure that along the flow there is swirling of the fluid inside the tube.

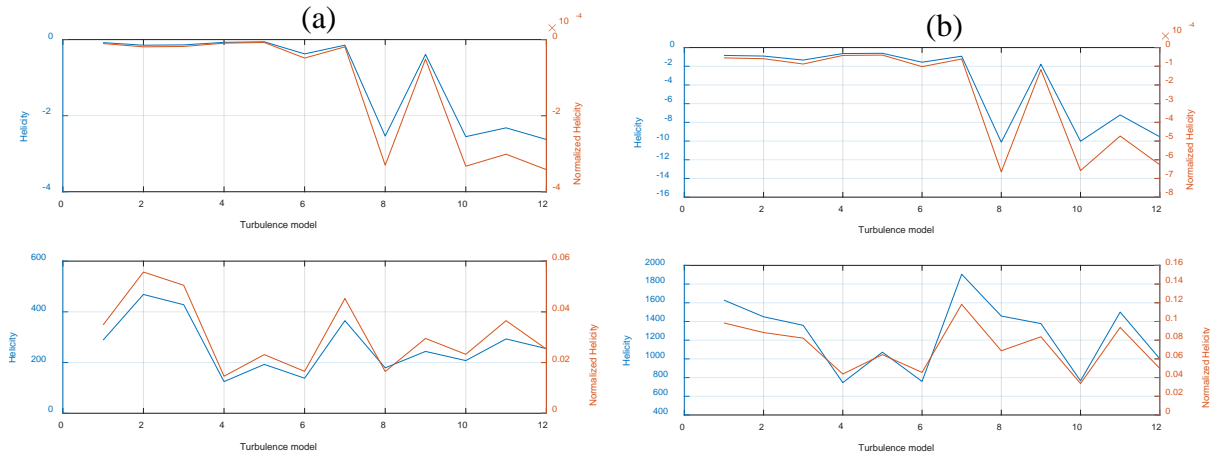


FIG. 5. Variation of the helicity and normalized helicity at the beginning (top) and end (bottom) of screw tube using different turbulence models with (a) 4 m/s, 0.93 MPa, (b) 8 m/s, 0.74 MPa.

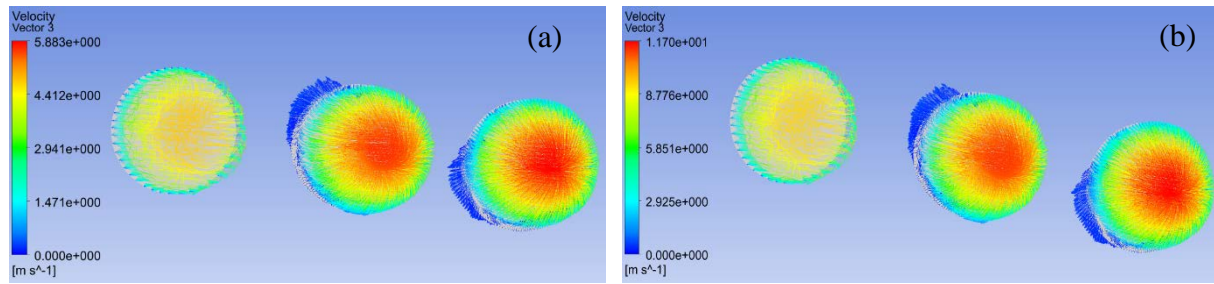


FIG. 6. Variation of the velocity vectors at different locations and turbulence models with (a) 4 m/s, 0.93 MPa, (b) 8 m/s, 0.74 MPa, Inlet velocity and pressures respectively.

## 6. Conclusions and future work

The article focuses on fluid dynamic analysis of screw tube and its usage as a heat sink component in the future fusion reactors. In the article, a brief introduction about the screw tube is given which is followed by the literature survey. The work done by Masaki et al. is used to select the geometry of the screw tube in order to see the effect of different turbulence models on the behavior of screw tube. The pressure drop in the screw tube is expected using the different turbulence models and it is found that none of the models actually agrees with each other. Due to lack of proper experimental data related to the fluid dynamic aspects the author was unable to compare the exact pressure drop with respect to the experimental data. However the data related to heat transfer is available, so as a next step author would like to do the heat transfer analysis using the available results from the turbulence modelling to determine the best turbulence model that can be used for analyzing the heat transfer behavior in the screw tube. As a future work the author will extend and complete his study on the heat transfer analysis of the screw tube both in single and two phase flow including boiling. Also the author would like to conduct experiments related to screw tube to get detailed flow physics and boiling heat transfer.

## 7. Acknowledgements

This activity was partly supported by the R4S project (MSMT grant no. LQ1603), the EUROfusion project (Euratom funding, grant agreement number 633053) and co-funded by the Ministry of Education, Youth and Sports of the Czech Republic (grant no. MSMT-41274/2014-2), for which the author is thankful. The author is also thankful to Dr. Ladislav Vala for reading the first draft of the paper and providing valuable suggestions.



## 8. References

- [1] <http://www.iter.org/>.
- [2] <http://www.efda.org/>.
- [3] Mitteau, R., et al., A shaped First Wall for ITER, *Journal of Nuclear Materials*, 415 (2011), S969–S972.
- [4] Autissier, E., et al., Design optimization of plasma facing component with functional gradient material Cu/W interlayer, *Fus. Eng. Des.* 88 (2013) 1714–1717.
- [5] Suzuki, S., et al., Development of the plasma facing components in Japan for ITER, *Fus. Eng. Des.* 87 (2012) 845–852.
- [6] Merola, M., et al., Engineering challenging and development of the ITER Blanket system and Divertor, *Fus. Eng. Des.* 96-97 (2015) 34-41.
- [7] Domalapally, P., Entler, S., Comparison of cooling schemes for high heat flux components cooling in fusion reactors, *Acta Polytechnica* 55(2) (2015) 86–95.
- [8] Kolev, N.I., *Multiphase Flow Dynamics 2*, Springer, Berlin (2011).
- [9] Smid, I., et al., Comparison between various thermal hydraulic tube concepts for the ITER divertor, *Fusion Technologies* (1996) 263-266.
- [10] Masaki, K., et al., High heat load test of CFC divertor target plate with screw tube for JT-60 superconducting modification, *Fus. Eng. Des.* 61/62 (2002) 171/176.
- [11] Boscary, M., et al., Critical heat flux in subcooled water of one-sided-heated screw tubes, *Fusion Technol.* 5 (1999) 289.
- [12] Suzuki, S., et al., First wall and divertor engineering research for power plant in JAERI, *Fus. Eng. Des.* 81 (2006) 93–103.
- [13] Cattadori, G., et al., Hypervapotron Technique in Subcooled Flow Boiling CHF. *Experimental Thermal and Fluid Science* 7 (1993) 230–240.
- [14] STAR-CCM+ version 9.06.011 user's guide, CD-apdapco.inc. (2012).
- [15] Domalapally, P., Subba, F., Computational thermal fluid dynamic analysis of Hypervapotron heatsink for high heat flux devices application, *Fus. Eng. Des.* 98–99 (2015) pp. 1267–1270.
- [16] Boscary, J., et al., Dimensional analysis of CHF in subcooled water flow under one-side heating conditions for fusion application, *Fus. Eng. Des.* 43 (1998) 147–171.
- [17] Raffray, A.R., et al., Critical heat flux analysis and R&D for the design of the ITER divertor, *Fus. Eng. Des.* 45 (1999) 377–407.
- [18] Araki, M., et.al, Critical heat flux experiment on the screw tube under one side heating conditions, *Fusion Technol.* 29 (1996) 519–528.
- [19] Ezato, K., et al., Thermal fatigue experiment of screw cooling tube under one-sided heating condition, *Journal of Nuclear Materials* 329–333 (2004) 820–824
- [20] Ezato, K., et al., Critical heat flux testing on screw cooling tube made of RAFM-steel F82H for divertor application, *Fus. Eng. Des.* 75–79 (2005) 313–318.
- [21] Ezato, K., et al., Experimental examination of heat removal limitation of screw cooling tube at high pressure and temperature conditions, *Fus. Eng. Des.* 81 (2006) 347–354
- [22] Ezato, K., et al., Critical heat flux experiments using a screw tube under DEMO divertor-relevant cooling conditions, *Fus. Eng. Des.* 83 (2008) 1097–1101.
- [23] <http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+Fluent>.
- [24] ANSYS-FLUENT-16.0, User's guide, ANSYS.inc. (2015).
- [25] Wilcox, D.C., *Turbulence modeling for CFD*, DCW Industries.inc., California (1993).