31st International Conference Nuclear Energy for New Europe
NENE 2022
September 12-15
PORTOROŽ
SLOVENIA

Development Of A Thermohydraulic Model For DTT PFU

Patrik Tarfila, Boštjan Končar, Oriol Costa Garrido
Reactor Engineering Division, Jožef Stefan Institute
Jamova cesta 39
1000, Ljubljana, Slovenia
patrik.tarfila@gmail.com, bostjan.koncar@ijs.si, oriol.costa@ijs.si

Giacomo Dose[1], Francesco Giorgetti[2,3], Selanna Roccella[2]
[1]University of Rome "Tor Vergata", Industrial Engineering Department; [2]ENEA, Department of Fusion and Nuclear Safety Technology; [3]DTT S.c. a r.l.
[1]Via del Politecnico 1
[1]00133 Rome, Italy; [2]00044 Frascati, Rome, Italy; [3]Frascati, Rome, Italy
giacomo.dose@uniroma2.it, francesco.giorgetti@enea.it, selanna.roccella@enea.it

ABSTRACT

The development of a thermohydraulic model of the divertor plasma-facing units (PFUs) for the Divertor Tokamak Test (DTT) facility is presented. A computational fluids dynamics (CFD) simulation at a steady-state regime was performed, representing an operating condition relevant to the magnetic equilibrium in which DTT will operate (single null (SN) scenario). Input data for the simulation include inlet thermohydraulic parameters, heat loads (obtained from the physics simulation) and the geometric model of the DTT PFU. A coupled fluid-solid CFD model of the PFU was developed. The mesh sensitivity analysis was performed on a smaller model to select the final mesh for the full-scale CFD simulation. The simulation results include relevant thermohydraulic parameters such as pressure drop, temperature difference of the coolant between the inlet and outlet and temperature distribution in PFU solid structures. Numerical simulations were performed with the ANSYS CFD code.

1 INTRODUCTION

The Divertor Tokamak Test (DTT) facility is a tokamak fusion reactor, which will be built in Frascati (Italy) [1]. The main purpose is to develop a system for exhaust power and particles; the component used for this application is the divertor. With DTT, several divertor designs will be tested for different plasma configurations [2, 3].

Divertor is the exhaust for ashes and waste particles. These need to be removed, because they are contaminating the plasma and reducing its effectiveness. Since the plasma scrape-of-layer (SOL) is very thin, the heat fluxes on the surfaces of the divertor are very high, so they need to be made from a durable material and also water-cooling needs to be implemented [2, 3].

In preliminary calculations, hydraulic simulations without heat fluxes [4] and mechanical simulations with assumed heat transfer coefficient and fluid bulk temperature [5] were performed. A heat flux distribution was obtained from a physics simulation for a single null (SN) scenario (see Figure 1).

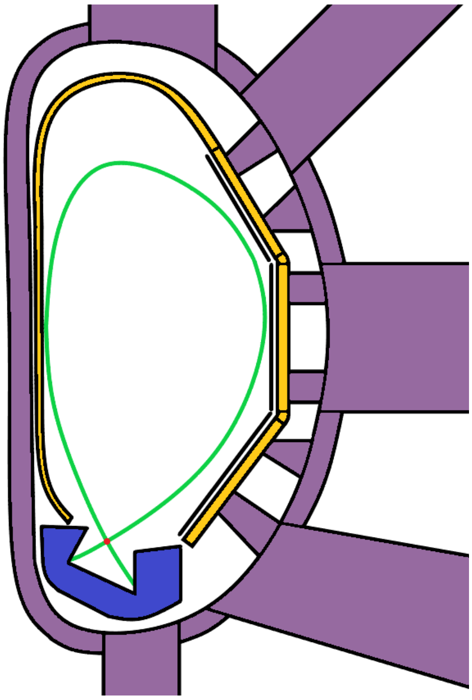

Figure 1: SN scenario in tokamak.

The main objective of the analysis presented in this paper is to calculate the temperature distributions in the solid domains of divertor for later thermomechanical simulations. The hydraulic boundary conditions were taken from the report [4], while the heat flux distribution was taken from the results of a plasma boundary simulation. One of the 9 plasma-facing units (PFUs) mounted on a divertor cassette is simulated, the geometry of which is shown in the Figure 2.

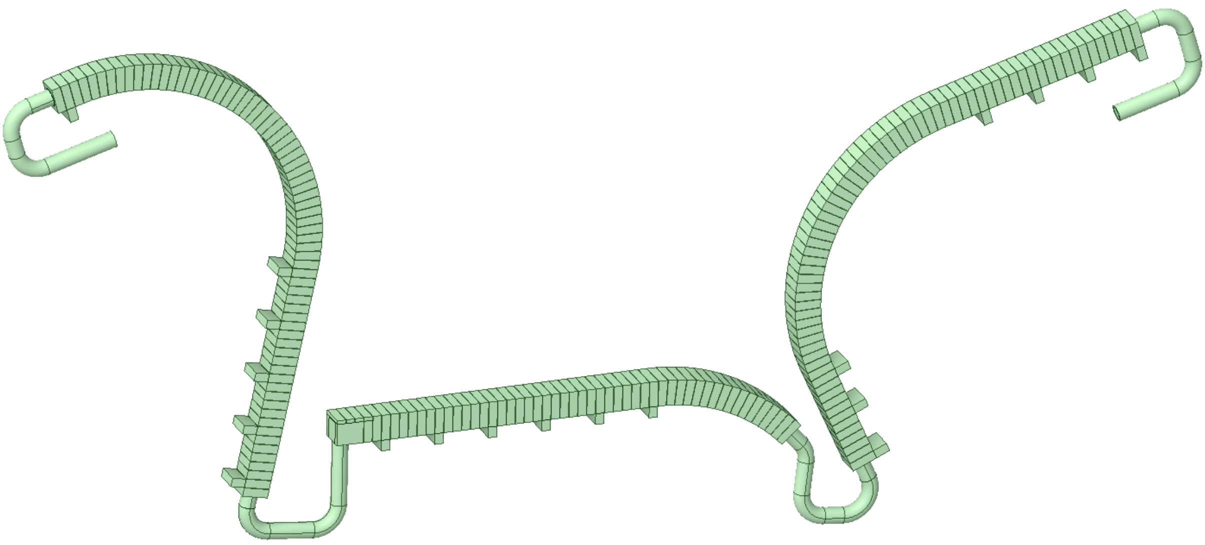

Figure 2: Geometry considered in simulation

2 PFU GEOMETRY MODEL

A computer aided design (CAD) model of PFU geometry was obtained from the DTT team and was accommodated for computational fluid dynamics (CFD) simulation, where additional edges were added for interfaces between individual parts and redundant edges, which would complicate the mesh, were removed. The geometry consists of 6 solid domains corresponding to different materials and one fluid domain (see Figure 3):

- Mono-block (Tungsten - W)
- Copper ring (Copper - Cu)
- Pipe (Copper-Chromium-Zirconium - CuCrZr)
- Twisted tape (Copper - Cu)
- Supports (Stainless steel - AISI316L(N))
- Connection pipe (Inconel - Alloy 625)
- Fluid (Water - H_2O)

All material properties (thermal conductivity, density and specific heat) are modeled as temperature dependent [6], since the heat fluxes are higher than 1 MW/m^2. As a result higher temperatures are expected, which can significantly change the material's behavior. Water properties were taken from the IAPWS tables in ANSYS CFX [7].

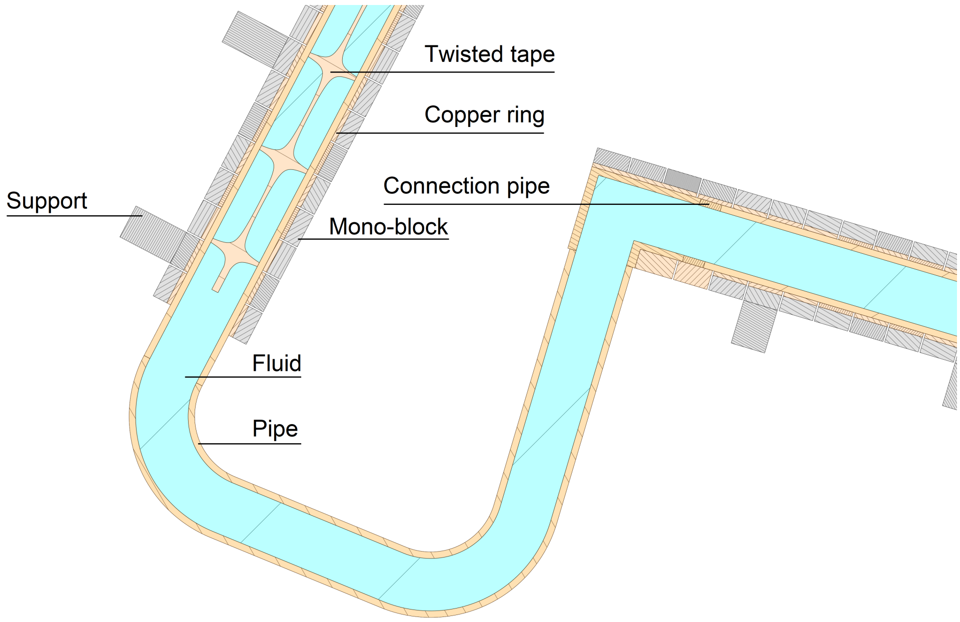

Figure 3: Geometry domains

3 COMPUTATIONAL MODEL

A coupled fluid-solid model was set up that solves the steady-state heat transfer in the solid and in the fluid. Simulation was performed using the finite volume code ANSYS CFX 21.2 [7].

Three-dimensional heat conduction equation is solved in the solid domains and transport equations for conservation of mass, momentum and energy are solved in the fluid domain. The Reynolds Averaged Navier-Stokes (RANS) equations are used for calculating turbulent flow. Turbulent model is $k - \omega$ Shear Stress Transport (SST).

3.1 Meshing

To set an appropriate mesh density, a mesh sensitivity analysis was first performed on a "reduced" model. The main purpose of the reduced model is to achieve mesh convergence and determine the mesh parameters for the full PFU model. The smaller geometry of the reduced model (see Figure 4) enabled faster calculations.

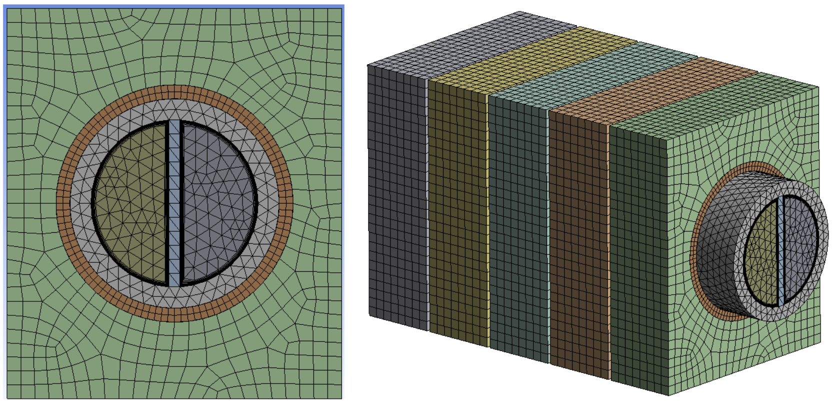

Figure 4: Mesh on the reduced model

The reduced model consists of 5 mono-blocks and it has one (360°) turn twisted tape. Twisted tape was included in the reduced model to define a correct mesh on the boundary layer in fluid near the pipe wall. The boundary conditions were the same as on the whole model (Section 3.2), except that the heat flux was set to a constant value of 5 MW/m^2.

Mesh convergence was checked on several different meshes and the results for three of them are presented in Table 1. Table 1 shows the maximum and average temperatures in solid domains, as well as the pressure drop and the maximum/average non-dimensional wall distance y^+ in the fluid (defined at the fluid-solid interface).

Table 1: Mesh parameter and results

TYPE:	Num. of	T_{max} [°C]			T_{avg} [°C]			p_{avg}	y^+_{max}	y^+_{avg}
MESH:	elem. [/]	MB	F	P	MB	F	P	[bar]	[/]	[/]
01	184839	400.80	77.24	144.91	149.04	60.57	83.66	2.78	46.67	30.30
02	875406	394.64	95.75	139.00	145.79	60.58	81.25	2.80	16.14	8.14
03	236994	395.58	97.35	139.63	147.15	60.59	82.04	2.79	14.92	7.84

Abbreviations: MB = mono-blocks, F = fluid, P = pipe

The mesh sensitivity with the reduced model shows that mesh 03 yield very similar results as mesh 02 with significantly less number of elements. Therefore the mesh parameters for the full PFU model are based on the mesh 03 of the reduced model. Twenty (20) inflation layers are applied on the fluid-solid interface with a height of 0.001 mm for the first near-wall layer. The number of elements is 9.5 million.

3.2 Boundary conditions

The boundary conditions (BC), presented in Figure 5, are based on 2021 reports [4, 5]. The inlet mass flow rate is 1.186 kg/s, outlet pressure 0 MPa, reference pressure 5 MPa, heat flux on the plasma-facing surfaces is based on SN plasma scenario and all other surfaces have adiabatic boundaries.

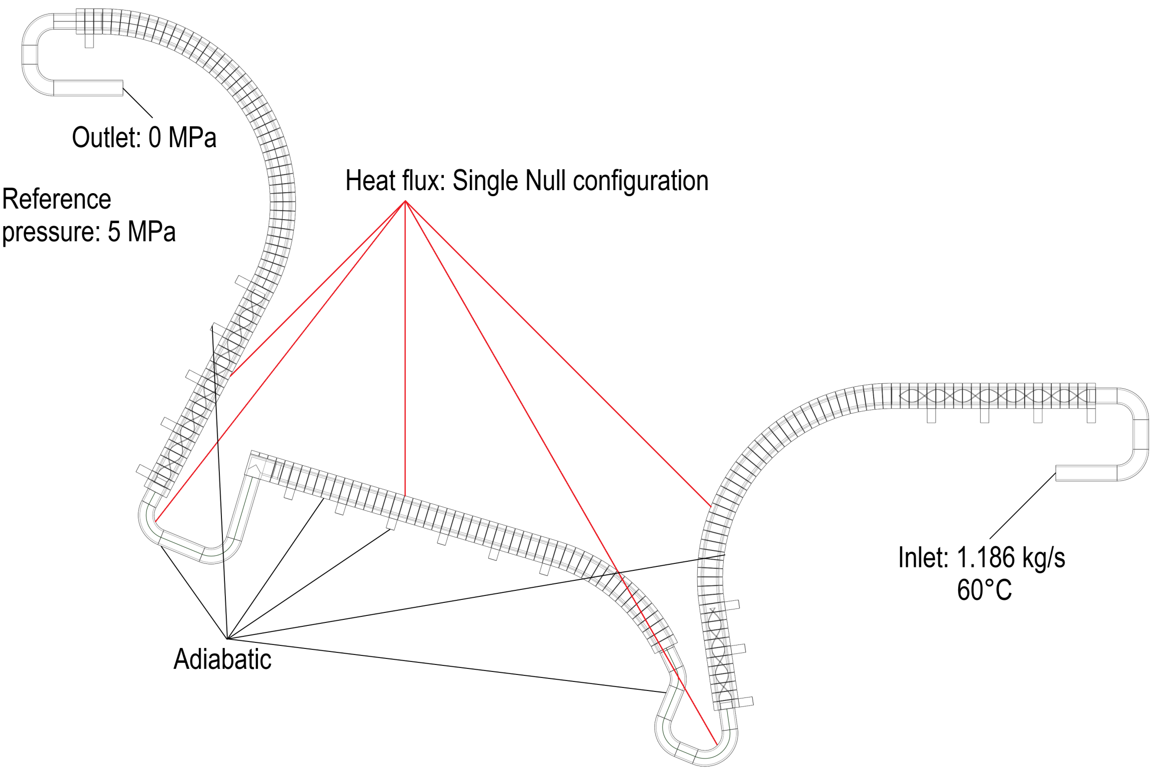

Figure 5: Boundary conditions on the whole model

The heat flux BC is defined from simulations for SN scenario and is provided in tabular form; 3 coordinates (X, Y, Z) and a value of heat flux. The values on mono-blocks are constant in toroidal direction, while on the bare pipes, the heat fluxes values are projected assuming a cosine function with the maximum value at the top of the pipe wall and zero at the sides.

Heat flux distribution along the PFU axis (length of the pipe) has 3 main peaks on the mono-block surfaces, occurring in the inboard, dome and outboard parts of the divertor PFU (see Figure 6). The maximum value of heat flux is 2.28 MW/m^2. The two smaller peaks in the valleys are heat flux values on the bare pipes.

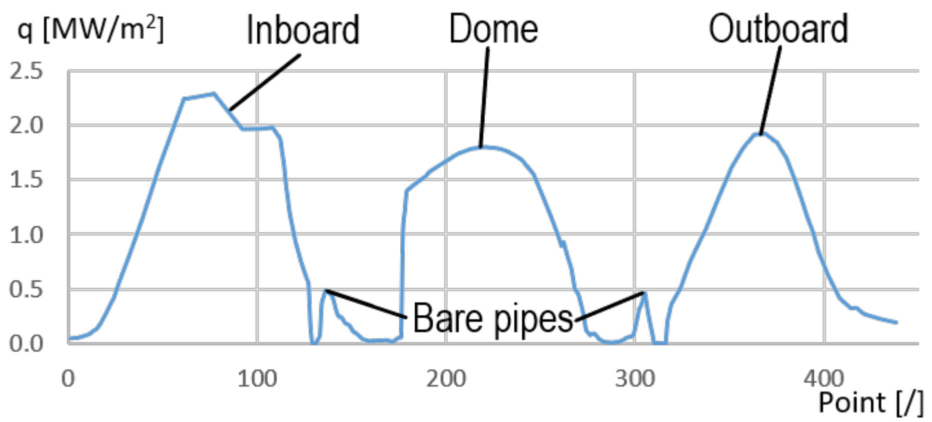

Figure 6: Heat flux distribution along the input data points (along the PFU axis)

The heat flux distribution applied as BC is presented in Figure 7, where the 3 main peaks from Figure 6 can be seen on the mono-block surfaces (ellipses) and two smaller peaks on the bare pipes (circles).

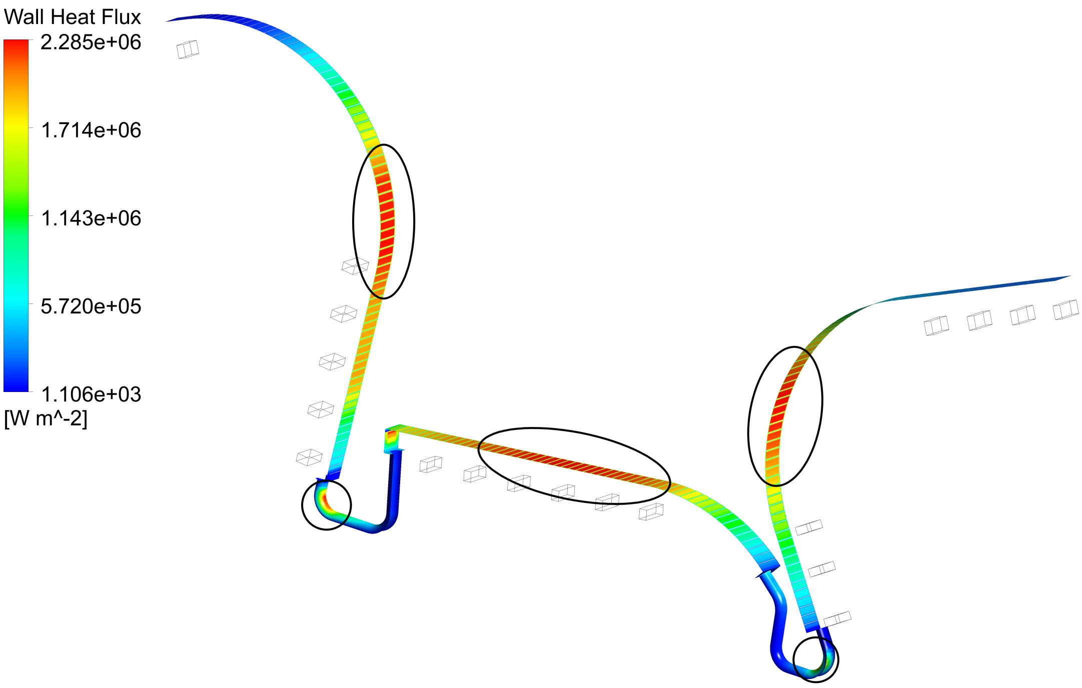

Figure 7: Heat flux distribution applied on PFU surfaces

4 RESULTS

The results of the most important parameters are presented in Table 2. It can be seen that the temperature is decreasing from the top of the mono-block surfaces towards the supports. Temperature difference in the fluid between inlet and outlet is roughly 10 °C and the pressure drop is around 4.8 bar, which is similar to the results of the preliminary simulations [4]. Parameter y^+ should be between 1 and 5, converging to 1. In our case, it is around 1.15, so the boundary layer is resolved directly without wall functions.

Table 2: Main results of the simulation

T_{max} Fluid [°C]	109.50
T_{max} Mono-blocks [°C]	219.70
T_{max} Pipe [°C]	126.43
T_{max} Copper rings [°C]	133.68
T_{max} Supports [°C]	83.94
ΔT Inlet-Outlet [°C]	10.36
Pressure drop [bar]	4.82
y^+_{avg} [/]	1.15

The temperature distribution on the mono-blocks is presented in Figure 8. Comparing this distribution with the applied heat fluxes (Figure 7), it can be observed that the temperatures follow the same pattern as the heat flux BC.

Velocity streamlines are presented on Figure 9. We can see that the twisted tapes increase the fluid velocity, thus increasing the heat transfer, at the cost of increased pressure drop. Average and maximum fluid velocity are about 7.4 m/s and 19 m/s respectively.

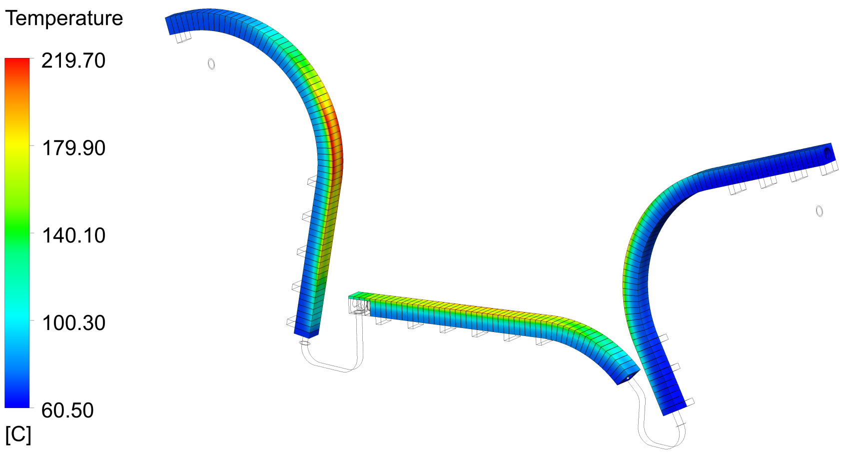

Figure 8: Temperature distribution on the mono-blocks

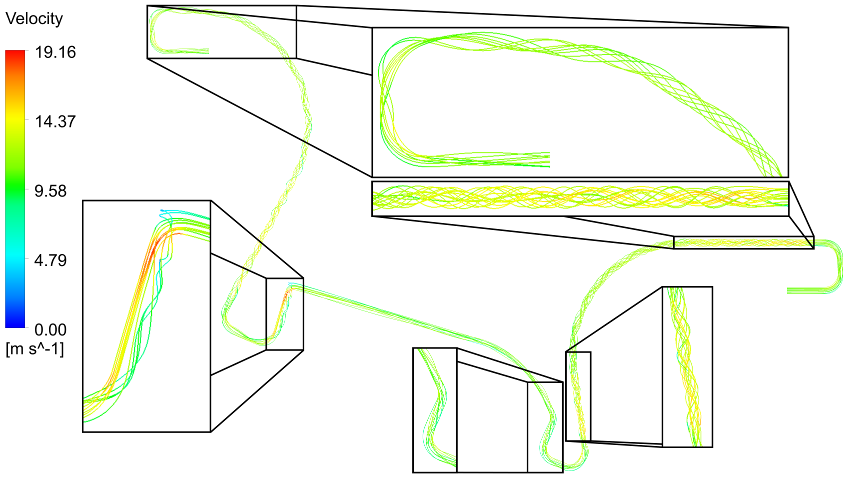

Figure 9: Velocity streamlines in fluid

The temperature distribution in the surrounding of the steel alloy (inconel) connection was also observed. The inconel material enables the welding of two pipe parts made of CuCrZr. Inconel has much lower thermal conductivity and in this case acts as a thermal resistor as can be seen in the fluid in Figure 10.

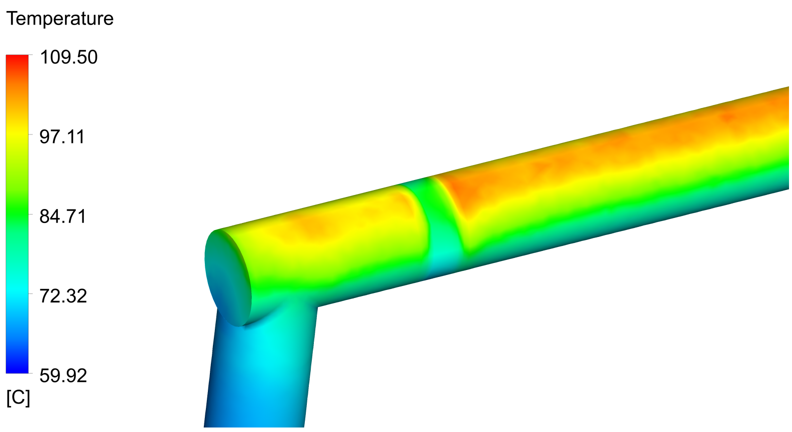

Figure 10: Impact of connection pipe on the fluid

5 CONCLUSIONS

A coupled fluid-solid simulation of one PFU was performed. Mesh sensitivity was checked on a reduced model to obtain mesh parameters for the full PFU model. Boundary and operating conditions correspond to the SN scenario. The maximum temperature on mono-block is 219.7 °C, in the fluid 109.5 °C and on the pipe 112.7 °C. The pressure drop is 4.82 bar, which is similar to the preliminary hydraulic simulations. Twisted tapes are enhancing velocity to increase heat transfer from the pipe to fluid.

The resulting temperature distributions in the solids will serve as an input for further thermo-mechanical simulations.

ACKNOWLEDGMENTS

This work has been carried out within the framework of the EUROfusion Consortium, funded by the European Union via the Euratom Research and Training Programme (Grant Agreement No. 101052200 - EUROfusion). Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or the European Commission. Neither the European Union nor the European Commission can be held responsible for them. The financial support from the Slovenian Research Agency, grant P2-0405, is also gratefully acknowledged.

REFERENCES

[1] G. M. Polli, "Divertor Tokamak Testing Facility (DTT): A Test Facility for the Fusion Power Plant", Paper presented at the OMC Med Energy Conference and Exhibition, Ravenna, Italy, September 2021. 978-88946678-0-6

[2] ENEA: DTT Project. Accessible: https://www.dtt-project.it/index.php/science/dtt-project.html, View date: 15.7.2022.

[3] ENEA, Divertor Tokamak Test facility – Interim Design Report, (2019)

[4] S. Roccella, G. Dose, F. Giorgetti, M. Angelucci, E. Martelli, Design of a standard divertor module: Deliverable DIV-IDTT.S.01-T001-D003, (2021)

[5] M. Angelucci, E. Martelli, S. Roccella, Report on Hydraulic analysis of the divertor module 2021: Deliverable DIV-IDTT.S.02-T003-D001, (2021)

[6] ITER material properties handbook (Appendix A, Materials Design Limit Data). From Reactor Engineering Division R4 JSI collection, View date: 17.5.2022.

[7] ANSYS: software ANSYS. Accessible: https://www.ansys.com/ products, View date: 16.5.2022.