International Conference
Nuclear Energy for New Europe

EU DEMO Upper Port Transfer Cask CFD Analysis

Martin Draksler, Boštjan Končar
Reactor Engineering Division, Jožef Stefan Institute
Jamova cesta 39
SI-1000, Ljubljana, Slovenia
martin.draksler@ijs.si, bostjan.koncar@ijs.si

Christian Bachmann
Eurofusion – Programme Management Unit
Boltzmannstrasse 2,
85748 Garching, Germany
christian.bachmann@euro-fusion.org

ABSTRACT

An analysis of the EU DEMO upper port transfer cask with a hot BB segment inside is carried out to determine the temperatures of the BB segment and the cask over the time. Since the cask is filled with air at 95 kPa, the natural circulation of the air is expected to establish, promoting the cooling of the BB segment by transferring the heat to the cask walls. It is assumed that the remote maintenance will take place 1 month after plasma shutdown. In addition to a transient 3D CFD analysis, a simplified theoretical model, based on the energy conservation law, was developed. Both models were used to predict the time-dependent temperature trends of a BB segment that has been placed into an empty cask. The analysis described here shows that naturally circulating air in the cask can provide a sufficient cooling mechanism for decay heat removal from the BB segment to stabilize its temperature within few days if the external cooling of the cask is sufficiently effective (e.g. either rather strong natural convection is established and/or the air temperature is sufficiently low).

1 INTRODUCTION

The DEMOnstration fusion power reactor (DEMO) is identified as a key step following ITER towards the exploitation of fusion power [1]. As such, DEMO must demonstrate the adequacy of implemented technologies for safe electricity production, developed on consideration that regular, rapid and reliable maintenance strategy of the plant can be carried out remotely [2]. With main requirements of DEMO to achieve operation with a closed fuel-cycle (i.e. tritium self-sufficiency) and reliable long-pulse plasma operation [3], the majority of the fusion energy transported by neutrons has to be absorbed in the breeding blanket (BB) [4]. Very high neutron fluence (more than one order of magnitude higher than in ITER [3]) will result in a high irradiation of in-vessel components (IVC) and high decay heat after the plasma shutdown. In [5] it is shown that during in-vessel maintenance, i.e. after a cool-down period of 4-8 weeks, the decay heat of the BB is reduced sufficiently to avoid excessive temperatures that would impede the remote handling operation. Once the BB temperature is sufficiently low, the segment is picked by the BB transporter and moved to the transfer cask.

In this article, natural convection cooling of the BB segment in a closed transfer cask, filled by air at 95 kPa is analysed. It is assumed that the remote maintenance will take place 1 month after plasma shutdown. The homogenized material properties are applied to BB segments, with consideration of the Water-Cooled Lithium-Lead (WCLL) concept. The applied heating power per outboard segment due to decay heat is 10 kW [6]. Based on our previous study [7], it is conservatively assumed that the BB temperature is 100°C when it would be picked by the BB transporter. A conjugate heat transfer model is applied to solve the coupled heat transfer in the solid and fluid (circulating gas). Heat transfer model considers also the radiation heat exchange that is modelled with the Surface-to-Surface (S2S) radiation heat transfer model [8]. The ANSYS Fluent code 2021.R2 [8] was used to run transient CFD simulations over a time of several hours. In addition to a transient 3D CFD analysis, a simplified theoretical model, based on the energy conservation law, was developed. Both models were used to predict the time-dependent temperature trends of a BB segment that has been placed into an empty cask.

The aim of this task is to carry an analysis of the upper port transfer cask with a hot BB segment inside to determine the temperatures of the BB segment over the time. Since the cask is filled with air at 95 kPa, the natural circulation of the air is expected to establish, promoting the cooling of the BB segment by transferring the heat to the cask walls.

The structure of the paper is as follows: in Section 2, the CFD simulation setup is briefly presented. This is followed with the description of the simplified lumped model. The results of analysis are discussed in Section 3 and the main conclusions are drawn in Section 4.

2 SIMULATION MODEL

2.1 Geometry and applied boundary conditions

For the purpose of this analysis, a CAD geometry of the transfer cask and the BB segment [9] has been substantially simplified (see Fig. 1). For example, complete BB transporter mechanism (e.g. rails, trolley, trunk and gripper) is removed from the CFD model. The cask is modelled as a shell structure with thickness of 5 mm without any liners. The BB segment is considered as a homogeneous structure, i.e. the homogeneous material and thermophysical properties are applied, with consideration of the Water-Cooled Lithium-Lead (WCLL) BB design concept that is fully drained. The material composition of the BB segment is adopted from [10]. The homogenized material properties and geometrical data used in this analysis are reported in Table 1.

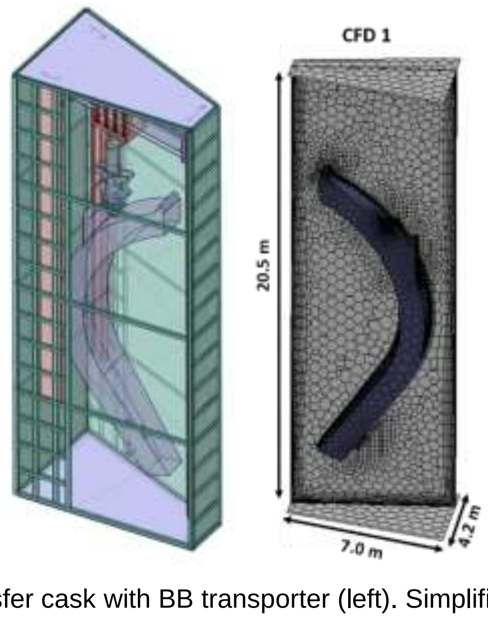

Figure 1: CAD model of transfer cask with BB transporter (left). Simplified CFD model with discretization mesh (right).

Table 1: Geometrical data and homogenized material properties used in the CFD study

	Volume $[\mathrm{m}^3]$	Surface $[\mathrm{m}^2]$	ρ $[\mathrm{kg/m}^3]$	C_p $[\mathrm{J/(kg\ K)}]$	λ $[\mathrm{W/(m\ K)}]$	Mass [kg]	Heat capacity $[\mathrm{MJ/K}]$	View factor
Outboard lateral* BB	21.8	79.6	2189.58	572.2	7.22	47805	27.3	0.97
Cask	2.4	433	7854	434.0	60.5	18520	5.3	0.17
Air (Cask interior)	381.9	-	Ideal gas	1006.4	0.02	410	0.4	-

* Fully drained

2.2 Considered scenario (decay heat and initial conditions)

It is assumed that the remote maintenance will take place 1 month after plasma shutdown. From data reported in [6], it has been estimated that 30 days after plasma shutdown the decay heat in 80 BB segments would be ~571 kW, which yields ~7 kW per segment. In this study, it is conservatively assumed that the heating power per outboard segment due to decay heat is 10 kW. Based on our previous study [1], it is also conservatively assumed that the BB temperature is 100°C when it would be picked by the BB transporter.

2.3 CFD code and setup

Transient simulations of the natural convection cooling by air inside the cask have been performed with the ANSYS Fluent code 2021.R2 [8]. The flow is considered laminar [11]. The radiation heat exchange is modelled with the Surface-to-Surface (S2S) radiation heat transfer model [8]. Emissivity of the surfaces, participating in the radiation heat exchange between the blanket segment and the cask interior) is set to 0.3 [13]. Up to 20 energy iterations were performed after individual radiation iterations, whereas the maximum number of radiation iterations was increased to 100. The residual convergence criteria for S2S model, continuity, momentum and energy equations were set to 10^{-5}. Simulations were run using the Coupled transient solver with 2^{nd} order schemes for temporal and spatial discretization.

In order to achieve the solver convergence at the beginning of the simulation, when the air is in rest and the natural convection is not yet established, very small simulation time step is needed. The simulation is initiated with the time step dt set to $0.1\ \mu s$, which is gradually increased to $0.1\ ms$ [11]. To speed up the solution of the transient, the so-called frozen-flow treatment has been used [7],[11]. In practice this means that the full set of equations (Navier-Stokes equations and heat transfer equations) was solved for 25 time steps, then the flow field was frozen for the next 25 time steps when only heat transfer equation was solved. Afterward, the full set of equations was solved again for the following 25 time steps. These steps were automatically repeated until the simulation was stopped. In the case of frozen-flow field, the simulation time step was increased from $0.1\ ms$ s to $0.1\ s$, which enables much faster progression of the transient simulation in physical time. Such approach has been tested and verified against the solution obtained without the frozen-flow assumption [7],[12].

For the purpose of this study, two CFD simulation cases were developed. In the first case (CFD1), only the interior of the cask has been considered in the analysis. As such, the no-slip wall boundary condition has been applied to the fluid surfaces in contact with the cask's internal walls. To consider the external cooling of the cask by surrounding air, the so-called convective heat transfer boundary condition (with specified external heat transfer coefficient $htc_{ext} = 5\mathrm{W/(m^2K)}$ and external heat-sink temperature set to 35°C) has been used.

The results discussed here were obtained with the conformal polyhedral mesh with hex-core, generated with the Ansys Fluent Meshing tool [8]. The surface element size (min/max) was set to 0.02/0.32 m, and 10 boundary layers were generated at solid walls around the BB segment and inside the (inner) cask walls. The total number of cells is 165000.

To gain the confidence in obtained results, another high-resolution mesh with 12 million elements has been tested. In this simulation case (CFD2), the cask is explicitly modelled and the external cooling is applied to the external surface of the cask. Since the cask wall with thickness equal to 5 mm is considered explicitly as a solid domain, the mesh is substantially refined at the cask's internal walls (2 cm wide cells) in order to achieve 3 cells across the cask wall thickness. It has been shown [11] that for the simulation time of one hour practically the same cooling trends of the BB segment were obtained with both tested meshes. As such, only the coarse mesh (without cask) has been used for further analysis.

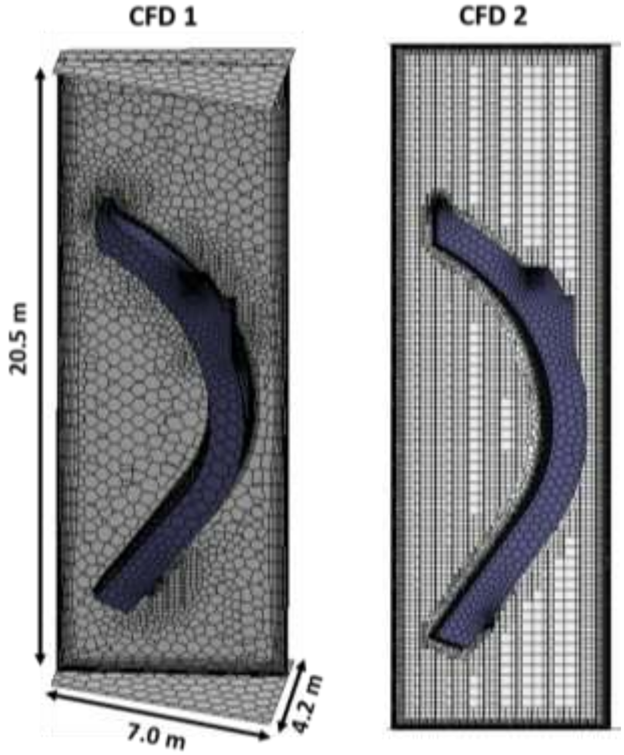

Figure 2: Simplified CFD model with discretization mesh (coarse mesh – left, refined mesh - right).

2.4 Lumped model

A 3D transient conjugate heat transfer simulation of natural convection cooling and radiation heat transfer is computationally very demanding, requiring a considerable amount of computational time and CPU resources. For example, 1024 CPU hours were needed to simulate 1 hour of physical time using a coarse mesh with 4M cells [14]. In order to be able to make predictions for longer periods of time (i.e. few days or even weeks), a faster model is needed. For this purpose, a simplified theoretical model, based on the energy conservation law, has been developed (see Figure 3). The change of temperature with respect to time $\partial T/\partial t$ equals the net change of system's energy per unit time due to decay heat, convection and radiation:

$$C_i \cdot \frac{dT_i(t)}{dt} = Q_{decay\ heat}(t) - Q_{losses}(T_i), \tag{1}$$

where $Q_{decay\ heat}(t)$ is the decay heat per unit time, and $Q_{losses}(T_i) = Q_{conv}^{i}(T_i) + Q_{rad}^{i\leftrightarrow j}(T_i, T_j)$. The C_i denotes the total heat capacity of the component under consideration (e.g. the BB segment, the air and/or the cask). Variable T_i denotes the time-dependent temperature, which is considered as the average temperature. The same equation is used also to compute the net change of energy for air and the cask.

The energy exchange due to convection and radiation from a surface S_i is defined by Eqs. (2) and (3), respectively:

$$Q_{conv}^{i}(T_i) = HTC_i \cdot (T_i - T_{bulk}) \cdot S_i \tag{2}$$

$$Q_{rad}^{i\leftrightarrow j}(T_i, T_j) = A \cdot \sigma \cdot \varepsilon \cdot \left(VF_i \cdot T_i^4 \cdot S_i - VF_j \cdot T_j^4 \cdot S_j\right). \tag{3}$$

Variable T_i denotes the volume averaged temperature of the component under consideration. The term htc_i describes the heat transfer coefficient [$\mathrm{W/(m^2\ K)}$] on i-th surface (e.g. BB external surface, cask's internal/external wall) with the temperature T_i and surface area S_i. The radiation heat exchange between two surfaces (S_i, S_j) is described by the Stefan–Boltzmann law, where $VF_{i\rightarrow j}$ is the view factor from surface S_i to S_j, ε is the emissivity and σ is the Stefan-Boltzmann constant. The constant A in the eq. (3) is used for additional calibration of the term to consider also the reflection of radiation from receiving surfaces.

The system of governing equations (energy conservation equations for BB segment, air and cask) is solved numerically using the Wolfram Mathematica 9.0. Graphical depiction of the lumped model is shown in Figure 3.

Though the model considers transient behavior of heat loads/sinks, one should be aware of major limitations of the lumped model, which are: (i) Homogeneous temperature across BB sergment is assumed at any given time instant; (ii) Inputs from CFD simulations are required for the wall heat transfer coefficient (htc) and view factors (VF) for radiation heat transfer.

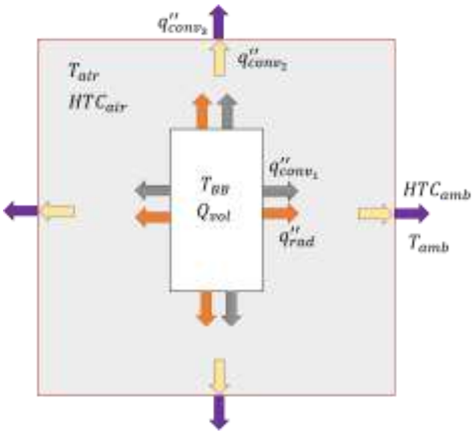

Figure 3: Graphical depiction of the lumped model.

3 RESULTS

In this section, the results of analysis are presented and discussed. It is assumed that the BB segment is fully drained.

3.1 Temperature contours

Contours of BB temperature, reported 1 hour after beginning of simulation are shown in Figure 4. It is noted that the temperature distribution across the structure of BB segment is nonuniform, i.e. the segment is hotter in the interior.

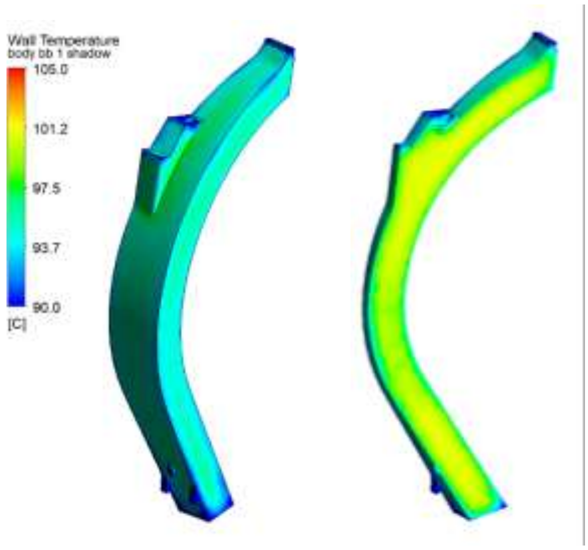

Figure 4: Temperature distribution in BB segment after one hour of simulation time.

The time trends of maximum and average BB temperature, obtained with both tested meshes, are shown in Figure 5. Both simulations yield rather very similar time-trends of BB temperature despite the differences in applied boundary conditions for the cask and despite that the resolutions of used numerical meshes are very different.

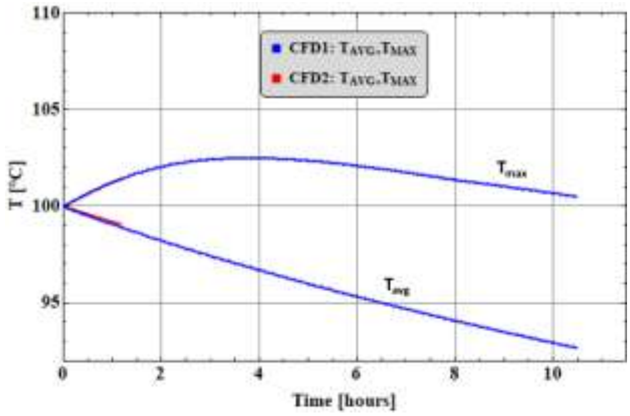

Figure 5: Maximum and average temperature of the BB segment for both CFD simulations.

Figure 6 shows various predictions with the lumped model (solid lines), which are presented together with the CFD results (symbols). Analysis with the lumped model shows that the BB average temperature would increase in time if no cooling is provided (yellow line). The blue symbols (CFD1 result) indicate that the BB temperature would remain rather stagnant if only radiation heat transfer is considered for the cooling of the BB segment. This solution has been used for the calibration of the lumped model (the coefficient A in eq. (3) is implemented to consider also reflection of the incident radiation). The calibrated lumped model has been further used for predictions of the averaged BB temperatures with various applied cooling rates at the BB external walls. This parameter, htc_{BB}, is used to describe the effectiveness of the natural convection cooling at the BB segments.

For lumped model prediction, the applied heat transfer coefficient at cask's internal wall (htc_{Cask}) is set to $2.5\ \mathrm{W/(m^2K)}$, while the cask is externally cooled by the $htc_{ext} = 5\ \mathrm{W/(m^2K)}$ – the same value is used also in CFD simulations.

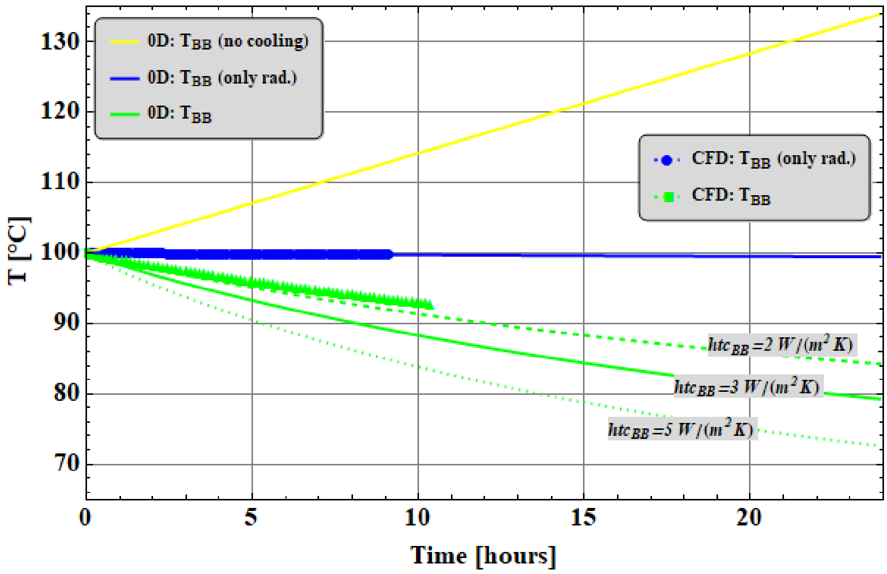

Figure 6: Predicted BB average temperature by CFD simulations (symbols) and lumped model (lines). Different styles of green lines correspond to lumped model predictions with different convective cooling rates (htc_{BB}), applied at the BB walls.

Based on the CFD results, corresponding heat transfer coefficients at the BB external walls (htc_{BB}^{Total}) and the cask internal walls (htc_{Cask}^{Int}, CFD2 only) were assessed. The obtained values are reported in Table 2. The htc_{BB}^{Total} is evaluated based on the total removed heat from the BB segment due to the natural convection and thermal radiation. The contribution of the natural convection cooling is approximately 40% [15]. As such, the corresponding htc_{BB}, representing only the natural convection cooling is estimated to $\sim 2W/(m^2K)$.

Table 2: Estimated heat transfer coefficients from the CFD simulations

	CFD 1		CFD2		
Elapsed time	htc_{BB}^{Total}	htc_{Cask}^{Ext} (applied)	htc_{BB}^{Total}	htc_{Cask}^{Int}	htc_{Cask}^{Ext} (applied)
$0.5\ h$	4.9	5.0	4.0	2.6	5.0
$\sim 1\ h$	5.1	5.0	4.3	2.6	5.0
$\sim 2\ h$	4.9	5.0	4.3	2.4	5.0
$\sim 5.5\ h$	4.8	5.0	-		

4 CONCLUSIONS

An analysis of the upper port transfer cask with a hot BB segment inside is carried out to determine the temperatures of the BB segment over the time. Since the cask is filled with air at 95 kPa, the natural circulation of the air is expected to establish, promoting the cooling of the BB segment by transferring the heat to the cask walls. It is assumed that the remote maintenance will take place 1 month after plasma shutdown. In addition to a transient 3D CFD analysis, a simplified theoretical model, based on the energy conservation law, was developed. Both models were used to predict the time-dependent temperature trends of a BB segment that has been placed into an empty cask. In this study, it is conservatively assumed that the heating power per outboard WCLL BB segment due to decay heat is 10 kW. Based on our previous study [7], it is conservatively assumed that the BB temperature is 100°C when it would be picked by the BB transporter.

Properly refined and calibrated lumped model represents an accurate and fast prediction tool which can provide rather accurate results, similar to the detailed CFD simulations as long as the assumptions used in the model are in line with CFD simulations. However, it should be kept in mind that the lumped model assumes that the temperature of the structure is uniform at any given time instant.

This study has shown that naturally circulating air in cask can provide a sufficient cooling mechanism for decay heat removal from drained BB segment to stabilize its temperature within few days if the external cooling of the cask is sufficiently effective (e.g. either rather strong natural convection is established and/or the air temperature is sufficiently low). The heat transfer coefficient at the external surfaces of the BB segment due to the natural convection cooling is estimated to $\sim 2\ \mathrm{W/(m^2K)}$.

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.

The CAD model of the transfer cask with the BB segment inside has been provided by dr. Rocco Mozzillo, within the EUROfusion WPDES project.

REFERENCES

[1] A.J.H. Donne et al. European Research Roadmap to the Realisation of Fusion Energy (2018), https://www.eurofusion.org/fileadmin/user_upload/EUROfusion/Documents/2018_Research_roadmap_long_version_01.pdf.

[2] https://www.euro-fusion.org/programme/demo/.

[3] IRSN, “Nuclear Fusion Reactor //Safety and Radiation Protection Considerations for Demonstration Reactor that Follow the ITER facility”, online publication, https://www.irsn.fr/EN/Research/publications-documentation/Scientific-books/Documents/ITER-VA_web_non_imprimable.pdf.

[4] G. Federici, et al., An overview of the EU breeding blanket design strategy as an integral part of the DEMO design effort, Fus. Eng. Des. 141 (2019) 30–42, https://doi.org/10.1016/j.fusengdes.2019.01.141

[5] M. Draksler et al., Assessment of residual heat removal from activated breeding blanket segment during remote handling in DEMO, FED 173 (2021), https://doi.org/10.1016/j.fusengdes.2021.112891.

[6] T. Eade, et al., Activation and decay heat analysis of the European DEMO blanket concepts, Fus. Eng. Des. 124 (2017) 1241–1245, https://doi.org/10.1016/j.fusengdes.2017.02.100.

[7] M. Draksler et al., Assessment of residual heat removal from activated breeding blanket segment during remote handling in DEMO, FED 173 (2021), https://doi.org/10.1016/j.fusengdes.2021.112891.

[8] Fluent User Guide, version 2021.R2

[9] Private communication with Rocco Mozzillo, email from July 28 2022. Geometry shared via Dropbox.

[10] A. Valentine, Nuclear heat in FW/BB during ex-vessel LOCA transient, Report, 2021, *EFDA_D_2NDPJ2*

[11] M. Draksler, A. Bashavaraj, P. Črne. CFD assessment of natural gas flow in the VV and in the upper port transfer cask: 18/01/2023: associated deliverable DES-ENG.MECHENG.MECHAN-T011-D002, 2023. 68 str. EFDA_D_ 2NL5SQ.

[12] M. Draksler, CFD assessment of air flow in VV during in-vessel-component transport, EFDA_D_2NEUJ7, Report, 2020.

[13] C. Bachmann, Task specification: CFD assessment of natural gas flow in VV during ex-VV LOCA, Private communication, 2021.

[14] M. Draksler, J. Justin, C. Bachmann and B. Končar. Thermal Loading of DEMO Breeding Blanket During Maintenance Conditions. In Proceedings of NENE2020, Portorož, Slovenia. September 7 -10, 2020.

[15] J. Kaufman, Hlajenje segmenta oplodne obloge reaktorja DEMO v transportni komori, Zaključna naloga, FMF UL, 2023.