INTEGRATED ASSESSMENT OF THERMAL HYDRAULIC PROCESSES IN W7-X FUSION EXPERIMENTAL FACILITY

T. Kaliatka, E. Uspuras and A. Kaliatka
[1] Lithuanian Energy Institute, Breslaujos str. 3, LT-44403 Kaunas, Lithuania

tadas@mail.lei.lt, uspuras@mail.lei.lt, algis@mail.lei.lt

ABSTRACT

Energy received from the nuclear fusion reaction is one of the most promising options for generating large amounts of carbon-free energy in the future. Nowadays world needs new, cleaner ways to supply our increasing energy demand, as concerns grow over climate change and declining supplies of fossil fuels. Power plants based on nuclear fusion reaction would have a number of advantages. However in this technology existing physical and technical problems are complicated. The problems, related to the safety of humans and surrounding environment are occurring in addition to the all physical and engineering issues, given to nuclear fusion devices. Several experimental nuclear fusion devices around the world have been already constructed and a few are under construction. However, the processes in cooling system of in-vessel components, vacuum vessel and pressure increase protection system of nuclear fusion devices are not widely studied. The most amount of radioactive materials is concentrated in the vacuum (plasma) vessel of the fusion device, but this component cannot withstand to a pressure even slightly above atmospheric. Any rupture of cooling system pipe of in-vessel components, leading to ingress of water in the vacuum vessel may lead to sharp pressure increase and possible damage of vacuum vessel. Therefore, systematic and detailed experimental and numerical studies, regarding the thermal-hydraulic processes in cooling system and vacuum vessel, are important and relevant.
In this article the study of thermal-hydraulic processes in cooling systems of in-vessel components, vacuum vessels and pressure increase protection system of nuclear fusion devices will be provided. “Ingress of Coolant Event” experimental facilities were modelled using RELAP5 program code and the calculation results were compared with the experimental data. Using the experience gained from the modelling of “Ingress of Coolant Event” experiments, the numerical model (including cooling system of in-vessel components, vacuum vessel and pressure increase protection system) of Wendelstein 7-X experimental fusion device was developed. The integrated analysis of the thermal-hydraulic processes occurring in the cooling system of in-vessel components, vacuum vessel and pressure increase protection system of Wendelstein 7-X experimental fusion device was performed in the case of loss of coolant.
The provided analysis showed that RELAP5 code is capable of modelling the processes in cooling systems of in-vessel components, vacuum vessels and pressure increase protection system of nuclear fusion devices in the case of loss of coolant event. Integrated analysis of W7-X device in the case of loss of coolant event showed that the design of facility prevents the failure of vacuum vessel.

KEYWORDS
Wendelstein7-X, RELAP5, Thermal-hydraulics, Fusion, Loss of coolant event.

1. INTRODUCTION

During the 60-year history of the nuclear fission reactors currently operating nuclear fission reactor power is 400 GW in the world [1]. Unfortunately A large amount of radioactive waste generated by the operation and safety aspects is the key factors, which force humanity to look for alternatives of nuclear fission energy. One of the alternatives to the energy produced in fission reactors is energy released during nuclear

fusion reaction and the application of this energy to electricity. Fusion power is the power generated by nuclear fusion processes. In fusion reactions, two light atomic nuclei fuse to form a heavier nucleus. In doing so they release a comparatively large amount of energy arising from the binding energy due to the strong nuclear force which is manifested as increase in temperature of the reactants. However in order to achieve efficient use of energy from the fusion reaction, a number of fusion physics and engineering problems still need to be solved. Key problem among them is how to maintain a stable high temperature ($T>10^8$ K), of the plasma in the vacuum vessel for a long time. In addition to all these physical and engineering questions, nuclear fusion devices must be safe for humans and the surrounding environment. In order to prove safety of nuclear fusion devices the various physical and numerical experiments should be provided. Only when physical and engineering problems will be solved and the safety of nuclear fusion devices will be proved – these devices will be used in the world energy generation.
One of research fusion facilities is stellarator type Wendelstien 7-X, which presently under construction at the Max-Planck-Institut für Plasmaphysik, Greifswald, Germany [2]. The superconducting magnet system enables continuous operation, limited by the cooling water system whose capacity to remove the plasma heat load onto the wall components is designed for 30 minutes of full power operation. W7-X torus is presented in Figure 1.

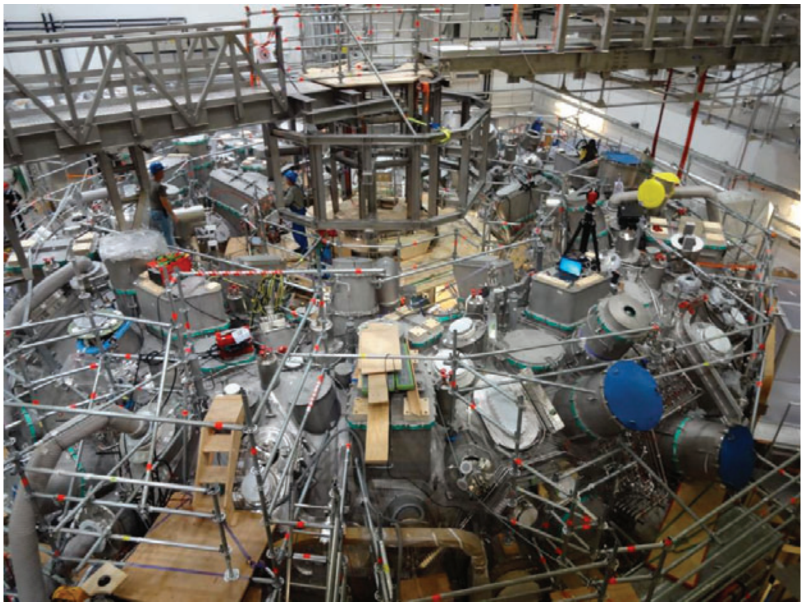


Figure 1. Picture of torus of W7-X facility which is under construction taken at the end of 2013.

The ingress of water during the W7-X no-plasma “baking” operation mode into the plasma vessel represents one of the critical failure events, since primary and secondary steam production leads to a rapid increase of the inner pressure in the vacuum (plasma) vessel. In this article the integrated analysis of the Loss-of-Coolant Accident (LOCA) in W7-X facility is presented. A rupture of the 0.04 m target module cooling pipe could lead to the loss of vacuum condition up to an overpressure in the vacuum vessel, damage of in-vessel components (for example: the bellows of the ports) [3, 4]. The pressure behavior in the plasma vessel depends on the amount of discharged water through the leak. Thus, the processes both in the target modules cooling system and plasma vessel should be modelled. Pressure increase protection system is used in order to prevent the vacuum vessel from the failure due to high pressure inside vacuum vessel. This system has special burst discs which opens then pressure in vacuum vessel reaches certain limit. During this action steam from vacuum vessel is transferred to atmosphere and the pressure in vacuum vessel must be reduced to atmospheric. The process (mainly condensation and heat transfer) in pressure increase protection system have the impact to target modules cooling system and vacuum vessel.

It should be noted that during “baking” operation mode no plasma in the plasma vessel exists, i.e. no special models for plasma simulation are required.
The analysis of LOCA event was performed by employing the computer models, based on thermal hydraulic system code RELAP5 Mod3.3. RELAP5 [5] is a “best estimate” system code for the analysis of all transients and postulated accidents in light water reactor systems, including both large and small-break loss-of coolant accidents as well as the full range of operational transients. RELAP5 code was chosen for the analysis, because the W7-X facility divertor cooling system is filled with water (coolant accepted in RELAP5 code) and can be described by RELAP5 components (pumps, valves, pipes, heat structures, etc.). However, there is no possibilities to model vacuum conditions in this code (lowest possible pressure is 700 Pa). In order to make sure that RELAP5 code is suitable to model the processes in vacuum vessel during LOCA event, the RELAP5 code was validated against Ingress of Coolant Event (ICE) experiments. Experience and modeling recommendations gained from the modeling of ICE experiments was used in order to develop the complex model (targets cooling system, vacuum vessel and pressure increase protection system) of W7-X and provide the analysis of LOCA event. The results of the presented studies are used to optimize the design of the coolant circuits of W7-X, which is now under construction, and to define protection measures and instructions in order to ensure safe operation.

2. MODELING OF PROCESSES IN VACUUM VESSEL

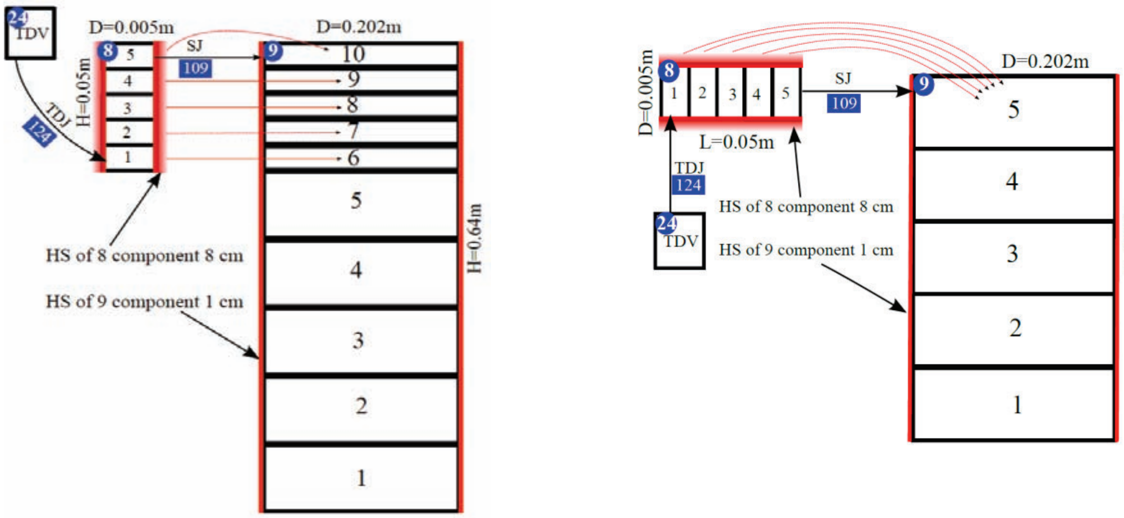
The analysis of processes in vacuum vessel was performed using the “best estimate” system code RELAP5 Mod3.3 [5]. The one dimensional RELAP5 code is based on a non-homogeneous and non-equilibrium model for the two-phase system that is solved by a fast, partially implicit numerical scheme to permit economical calculation of system transients. In addition, RELAP5 can be used to solve many plant thermal-hydraulic problems. The code includes many generic models allowing to simulate general thermal hydraulic systems. The models include pumps, valves, pipes, heat releasing or absorbing structures, reactor point kinetics, electric heaters, jet pumps, turbines, separators, accumulators, and control system logic elements. However, as it was mentioned in the previous paragraph RELAP5 is not applicable for the low pressure, which is in the vacuum vessel of fusion reactors. In order to use RELAP5 code for the analysis of the processes in vacuum vessel of real fusion facility (W7-X in our case) validation process must be done. For the validation of the models which represents the processes in the vacuum vessel the experimental data received from (ICE) experimental facility were used.

2.1. Modeling of ICE Experiments

There are few experimental investigations of water injection in the low pressure (vacuum) vessel performed. In some of these experiments the walls of vacuum vessel are heated or special hot steel structure are installed inside vacuum vessel [6]. These experiments reflects the situation to one of the operation mode (“backing”) of real fusion device. For the analyse the processes, which occur, in the case when water is injecting in vacuum vessel with hot surface inside, the experiment described in article [6] was chosen. The experimental facility consists of two main parts: a) water tank, and b) cylindrical vacuum vessel. The length and diameter of the water tank are, respectively, 1.41 m and 0.202 m. The vacuum enclosure contains the electrically heated block at the top (during experiment heated to 573 K temperature and using 870 W for auxiliary heating) and the nozzle at the bottom. The enclosure is 0.202 m in diameter and 0.69 m in length. The enclosure wall temperature is maintained equal to the incoming water temperature (353 K in analysed experiment). During analysed ICE experiment water pressure $3 \cdot 10^5$ Pa, pressure in VV is $8 \cdot 10^3$ Pa. The diameter of the nozzle is 1 mm. Water flow rate from the nozzle to VV is 1.34×10^{-2} kg/s. Water is injected from nozzle to the VV during all experiment – 100 s. These parameter values were set as initial values for the RELAP5 computation model.
In order to simulate ICE experimental facility two RELAP/Mod3.3 models were developed (Figure 2). Geometrical data, boundary conditions and other parameters in these models was chosen according to the

selected experiment. Both models consist of two pipe components. Pipe component “8” is used to model the heated block presented in the experiment. In the first model the interaction between the water jet and heated block was modelled using small vertical (Figure 2 *a)*) pipe. The second model has horizontal pipe (Figure 2 *b)*). The vacuum vessel (VV) in bout models is presented as vertical pipe element. The component “8” is very small (diameter = 0.005 m, height or length = 0.05 m) and it has heat structures (HS). The surface area of HS of component “8” is the same as in the experiment [6], the width is 0.08 m (according to experimental value 0.084m). The inner surface layer of HS is associated with component “8”, the outer surface associated with component “9”. This construction of HS allows to model heat transfer form heated block (component “8”) to VV (component “9”). The HS of component “8” has also heating of 870 W which is the same as the auxiliary heating in the ICE experiment. The dimensions of pipe, which represents the VV, is the same as VV in the experiment (diameter = 0.202 m, height = 0.64 m). This component also has HS. Width of this HS is 0.001 m and the outer surface of this HS is isothermal.

Components “8” and “9” are connected together through the single junction (SJ) “109”. Components “8” and “9” have internal nodes. Component “8” has 5 nodes. Component “9” depending on azimuthal angle of component “8”, if this component is vertical component “9” has 10 internal nodes, if horizontal - component “9” has 5 internal nodes. Water injection through the nozzle is modelled by time dependent junction (TDJ) “124”. In this junction the mass flow rate, measured in the ICE experiment, is given. The injected water parameters are indicated in the time dependent volume (TDV), this component models the water tank.



a) *b)*

Figure 2. Scheme of nodalization for RELAP5 code: a) Vertical heated tube, b) horizontal heated tube.

Calculation using RELAP5 was provided for both nodalization cases (component “8” vertical and horizontal). This was done in order to analyze the processes in VV and to clarify which case of nodalization is suitable for the modelling of ICE experiment with heated surface in VV. The comparison of ICE experimental data and calculation results are presented in Figure 3 and Figure 4. Experimental data was taken from article [6]. The comparison of calculated and measured pressure in the VV of ICE experiment is presented in Figure 3. As it is seen form the figure, calculation results using nodalization scheme with vertical component “8”, is more close to experimental data. The comparison of temperatures in the hot surface of component “8” and measurements in the heated block is presented in Figure 4. The measured temperatures (centre and periphery of heated block surface) in heated block and calculated temperatures of hot surface of component “8” are more similar in the case where component “8” is in vertical position. In the case where component “8” was modelled as horizontal pipe temperature in the first node of component “8” is similar to the temperature measurements in the heated block, but the second node has much higher temperatures. This is because in horizontal pipe (component “8”) contact time of water with the hot surface is very short and due to auxiliary heating, the modelled heated block is heated more then it could transfer to water.

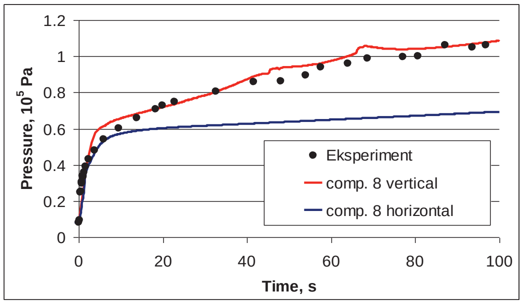


Figure 3. Comparison of calculated and measured pressure in the VV of ICE experiment

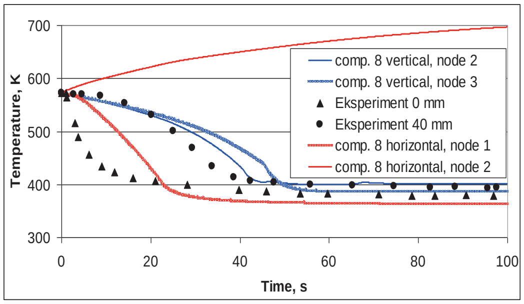


Figure 4. Comparison of the temperatures on the hot surface of pipe component “8” and measurements of the surface on heated block

2.2. Development of W7-X cooling system, vacuum vessel and pressure increase protection system model

The main parameters of W7-X are: average major radius 5.5 m, average minor plasma radius 0.53 m, total weight 725 t (Figure 1). W7-X facility is composed of 10 so-called divertor units located in the vacuum vessel with the bean-shaped cross section. The maximal height of the torus is 7. Each divertor unit is assembled from 12 separate horizontal and vertical target modules capable to remove maximum 10 MW/m^2 convective stationary power load [3].

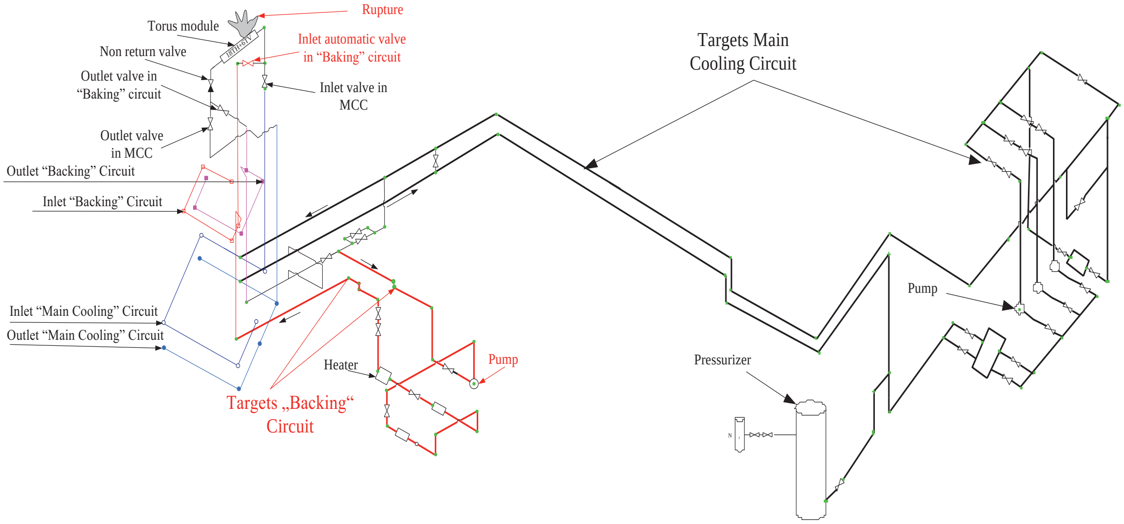


Figure 5. Simplified scheme of main cooling and “baking” circuits.

The W7-X facility target modules cooling system consists of two coolant circuits: the Main Cooling Circuit (MCC) and the so-called “baking” circuit. The MCC is used for cooling of the target modules when the W7-X facility is under normal operation. Before plasma operation, the target modules and other in-vessel components must be heated up in order to ‘clean’ the surfaces by thermal desorption and the subsequent pumping out of the released volatile molecules. The “baking” circuit is mainly used for this purpose. Both MCC and “baking” circuits are connected together and supply water to the same target modules. During operation of W7-X in the “baking” mode, the heat, necessary for target modules heating is generated in electrical heater. There is only one pump for all target modules loops in the “baking” operation mode. For the “cooling” operation mode there are three pumps available in the MCC. A simplified scheme of the circuits is shown in Figure 5.

The maximum water temperature is 433 K, the water pressure is about 10^6 Pa under the W7-X “baking” operation mode. The corresponding mass flow of water in the “baking” circuit is 177 m^3/h (44.6 kg/s), the flow velocity through the cooling tubes of the target modules during “baking” operation mode is about 1 m/s [4].

The both MCC and “baking” circuit in RELAP5 model are model in details. The measurements (pipe lengths, elevations, pump parameters, heater power and valves parameters) and the configuration of pipes (necessary for evaluation of form loss coefficients) were taken from the drawings provided by the W7-X design office [7]. The W7-X cooling system supplies 10 divertor units. The divertors can be grouped into two groups – upper and lower divertor units. One single upper divertor unit, connected with lower divertor unit creates one torus segment. Thus, each torus segment is comprised of 18 horizontal (9 upper and 9 lower) target modules and 2x3 vertical target modules. In the developed RELAP5 nodalization, four torus segments are modelled as equivalents (with the corresponding water volume and hydraulic resistance). One single torus segment is modelled in more extended format (Figure 6). In this torus segment, the upper and lower, horizontal and vertical target modules were modelled separately. From horizontal target modules two single target modules were selected: one in upper and one in lower position. These elements of single target modules allow to model rupture of single target module. In Figure 5 only one torus segment is shown. Each torus segment is connected to the MCC and “baking” circuit using valves, which are at the segment inlet and outlet (Figure 6). The valves at the inlet to torus segment from “baking” circuit are automatic. On the outlet of each torus segment check valve is installed. In case of pressure increase in the VV (it indicates the injection of water into vessel through rupture in cooling circuit) the automatic valves are closing, reducing the discharge of water.

The complicated three-dimensional geometry of the vacuum vessel volume in the stellarator (Figure 1) in the developed model is simplified (Figure 6). Whole volume (108 m^3) of the W-7X VV was modelled using three pipe components. Two pipe components are modelled as horizontal pipes and one as vertical pipe. Each pipe component is divided into 5 inner nodes. The end of one horizontal pipe is connected to the middle of the vertical pipe and it is connected to the beginning of the other horizontal pipe. Ends of the horizontal pipes are open and joined together, simulating the closed circle of torus geometry. The inner surface area (215.3 m^2) and wall thickness (0.017 m) of vessels structures in the model correspond to the available design data [3, 4]. One additional small volume (0.026 m^3) (pipe component) models the volume into which the water is released from the ruptured pipe. It is defined for the aims of simulation – it helps to more realistically model the steam – water mixture flowing from the ruptured pipe into the volume on the back side of divertor.

In the VV model of water from the cooling system is injected in the top (ingress of coolant event) of the vertical pipe. This nodalization scheme was chosen according to the experiences and recommendations gained from the modelling of ICE experiments.

The VV of W7-X facility is connected to the pressure increase protection system. This protection system consists of two burst disks, venting pipe lines and tank for condensed water in venting pipes (draining tank). Burst disks connect the VV with the venting pipe lines. These disks are installed to protect the VV from overpressure. The opening pressure of the first burst disk is $1.1 \cdot 10^5$ Pa, while the opening pressure of the second burst disk is $1.2 \cdot 10^5$ Pa (the absolute pressure). Both burst disks are installed on the pipelines of 0.3 m inner diameter that are connected to the main venting pipeline of 0.5 m inner diameter. The exit of main venting pipeline (chimney) is outside the building above the roof level. In case of a loss of coolant accident and the water ingress in the VV the opening pressure of the burst disk would be reached and after disk opening steam would enter venting piping and would be directed outside the building. The surface of venting pipe walls is colder then the steam released from the burst disk. The steam would be condensing on the colder surfaces of the venting piping. Therefore the venting piping is designed with an inclination, which ensures that water flows to the drainage tank (see Figure 6). Capacity of the drainage tank is 100 litres.

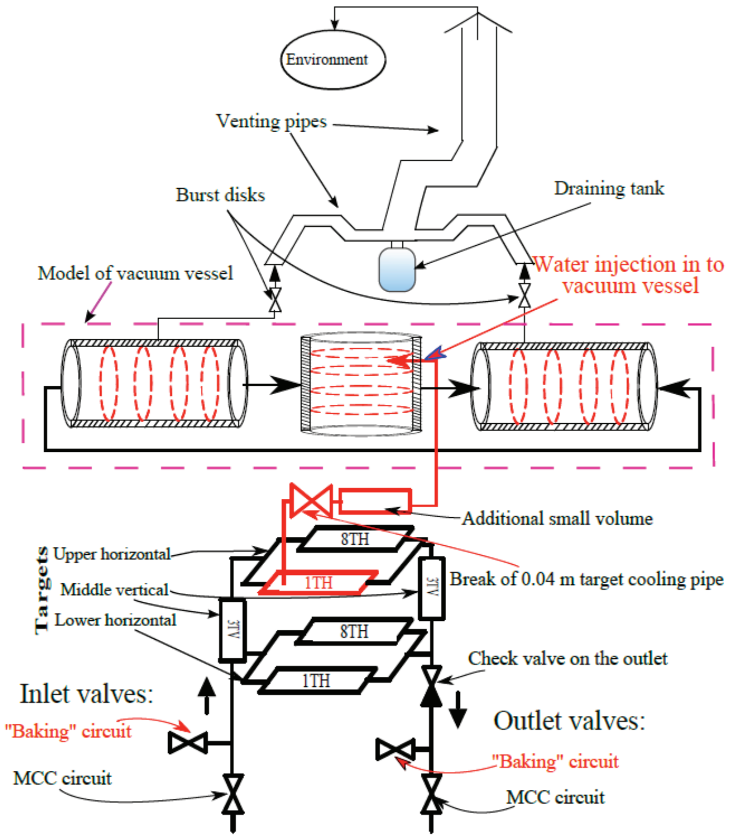


Figure 6. Nodalization scheme of affected target module and connection to the vacuum vessel.

The burst disks are simulated using RELAP5 valve components. Valves are closed at the beginning of simulations. The first valve opens if pressure difference in VV and venting pipelines exceeds $1 \cdot 10^4$ Pa. If pressure difference between VV and venting pipelines are still increasing and reach $2 \cdot 10^4$ Pa, the second valve opens. The flow area of these valves (when they are open) is equal to the area of the W7-X burst disks area (0.049 m^2). The environment is simulated in RELAP5 model using time dependent volume element with atmospheric pressure. Due to limitation of RELAP5 code it was assumed that the pressure in the modelled VV is equal to 1000 Pa. It was assumed during the modelling that the venting pipes of pressure increase protection system are made from stainless steel with wall thickness 0.001 m, initial pressure in the system – atmospheric and system is initially filled with air. The initial temperature of walls of venting pipes is 288 K in all pipes, except the top part of venting pipe, which is outside the building roof, and there minimal possible (due to limitation of RELAP5 code) initial temperature is 273 K.

The walls of the vacuum vessel and venting piping were modelled as a heat structure. The surface area of HS of vacuum vessel and venting piping are the same as the surface of W7-X experimental facility. The outer surface associated with the surrounding environment. This construction of HS allows to model heat exchange between the VV and venting piping and the surrounding environment. Environment temperature in W7-X hall (VV and part of venting piping) is 288 K temperature, air velocity ~1 m/s; environment temperature outside W7-X hall (part of venting piping) is 273 K temperature, air velocity ~3.5 m/s.

3. INTEGRATED ANALYSES OF THERMAL HYDRAULIC PROCESS

It was assumed during the modelling that rupture occurs in the 0.04 m diameter feeder pipe, connecting single upper horizontal target module (Figure 6). The following assumptions are used:

- Double ended guillotine rupture occurs at time moment t = 0 s. Break fully opens within 0.01 s.
- To reduce the discharge of water the automatic valves on the inlets to each torus segments are closing. Signal for automatic actuation of valves is generated when pressure in plasma vessel reaches $2 * 10^3$ Pa. The calculations (see below) show that pressure reaches $2 * 10^3$ Pa in plasma vessel ~ 1 s after the rupture occurred. Assumed delay between parameter reaches the set-point and signal generation – 0.5 s. Delay between signal generation and start of valve actuation – 1 s. Time to full closure of automatic valve on target module inlets – 5 s. Thus, 7.5 s after the rupture the torus segment inlet automatic valves are fully closed.
- Another measure to reduce the discharge of water from rupture to plasma vessel – automatic trip of pump in “baking” circuit. It was assumed that signal for automatic pump trip is the same as for closure of automatic valve – when pressure in plasma vessel reaches $2 * 10^3$ Pa (t = 1 s). The delay between the parameters reaches the set-point and pump trip – 1 s. Thus, trip of pump in the “baking” circuit begins ~ 2 s after the rupture.
- When the pressure in plasma vessel exceeds $1.1 * 10^5$ Pa (the absolute pressure), the pressure increase protection system is activated. First burst disk opens and steam from VV is discharged through venting pipes to the environment. Also some of the steam is condensed and drained to draining tank.

The discharge of coolant through the rupture is presented in Figure 7. The water flow rate from effected target cooling system to vacuum vessel reaches 28 kg/s for some seconds. Ingres of coolant increase the pressure in vacuum vessel and activate closure of automatic valves. After close of the inlet automatic valve the discharge of coolant through the rupture slightly decreases, but the water from other target modules in this torus segment is discharged until the pressure in piping decreases down to the pressure in vacuum vessel.

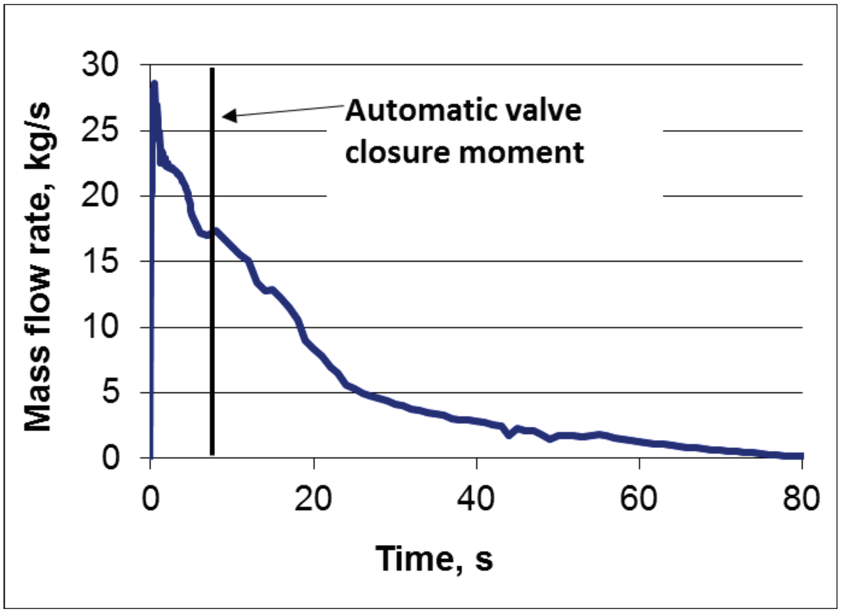


Figure 7. Water flow rate through the rupture to vacuum vessel

Then the pipe rupture accrues the pressure in "baking" circuit decreases (Figure 8) and circulation pump is switched off. However, after the automatic valves at inlets of torus segments start closure, the pressure in the "baking" circuit starts increasing. This is because inertia of pump impeller – after the pump trip the impeller is still rotating for about 1.5 minutes. The water is supplied into pipelines that cause the pressure increase in system upstream the valves (Figure 8). After the automatic valves are fully closed, the pressure in "baking" circuit starts to decrease slowly, because of pressuriser which controls the pressure in targets cooling system. After automatic valves closure the water is discharged only from the affected torus segment. The pressure in the affected torus segment slowly decreases to pressure in VV, when pressure in other (intact) torus segments remains initial pressure, because closed automatic valves isolated them.

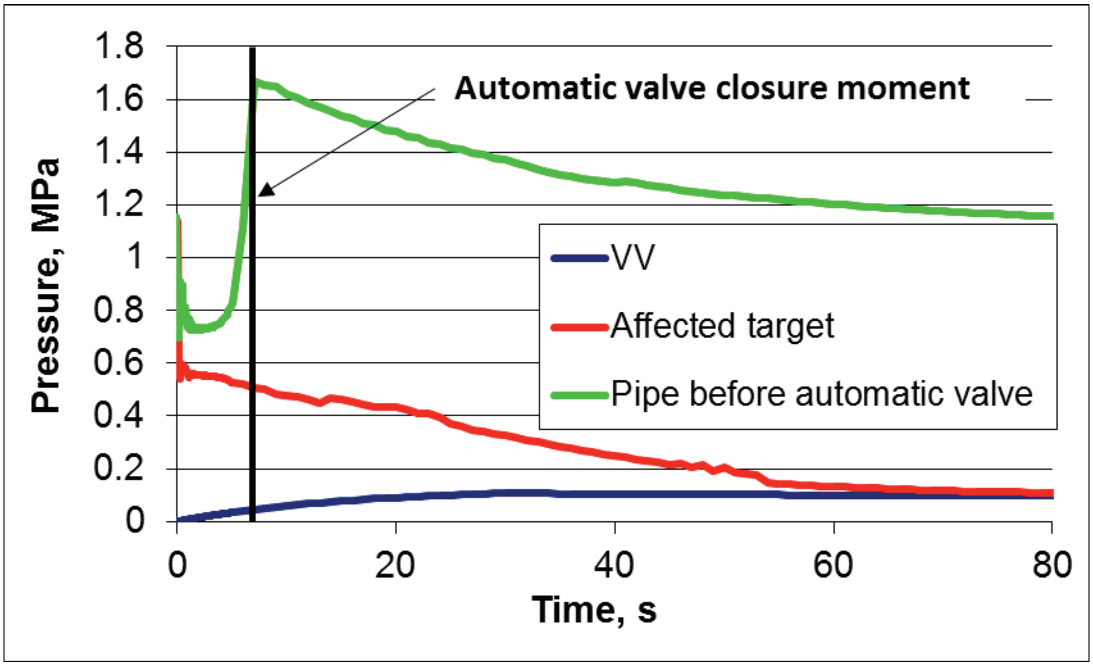


Figure 8. Pressure in cooling system and vacuum vessel.

After pipe rupture takes place, the steam – water mixture from the broken pipe is discharged into vacuum vessel through the small volume (Figure 6). This volume models the volume formed with vacuum vessel wall and target wall. Initially the conditions in the small volume are the same as in VV (air at vacuum condition). After the rupture, due to injection of coolant, the pressure inside VV starts to rise (Figure 8, Figure 9). When the absolute pressure in vacuum vessel reaches 1.1 * 10^5 Pa (32 s after the initiation of break), the first burst disk form the pressure increase protection system opens. After the opening of burst disk, steam is discharged to the venting piping and the pressure in VV starts decreasing till atmospheric pressure. The maximal absolute pressure peak in VV is ~1.1 * 10^5 Pa, which means that the diameter (0.25 m) of the installed burst disk is sufficient and pressure increase protection system is capable to prevent further pressure increase. The peak of the pressure in the venting piping is low, coming from the burst disk to the end of the pipe (Figure 9). The maximum value 1.04 * 10^5 Pa of the pressure is calculated in the node nearest the burst disk. In the venting pipes the pressure oscillations were received, but after ~3 s the pressure is stabilized to the initial pressure.

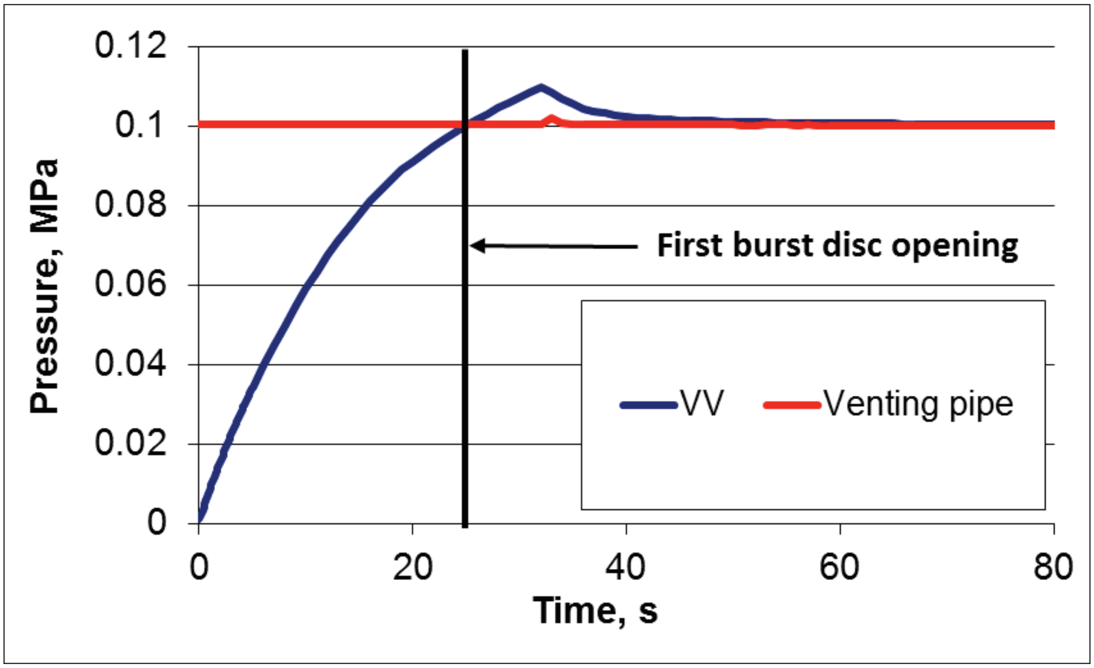


Figure 9. Pressure in vacuum vessel and pressure increase protection system.

The integral mass of water injected in the VV from the broken 0.04 m diameter feeder pipe, connecting single upper horizontal target module is presented in Figure 10. Water injection terminates ~100 s after the break. In total ~460 kg mass of water and steam is injected in the VV. As it was already mentioned, the burst disk opens ~32 s after the break. At this moment the water steam from VV releases to the pressure increase protection system. In the venting pipes of pressure increase protection system the biggest part of the steam is discharged to the environment, but some part of the steam are condensed in the venting pipe and drained to the draining tank. As it is shown in the presented figure, the mass of steam releasing through the burst disk continuously increasing until the water exists in VV (Figure 11). There is some imbalance between the mass of water discharged from the rupture of the affected torus segment to VV and steam mass discharge through the burst disk. This imbalance exists because some steam in the form of mist remains in the VV. In Figure 10 presented steam mass discharged through the end of the venting pipe. Until the calculation time ~5800 s the steam mass is increasing continuously, later (until ~12000 s) the mass remains constant. At the time moment ~5800 s the release through the burst disk decreases and all released steam are condensed not throwing it to environment. The small peak of the steam flow rate from the bust disk and to environment is shown in Figure 10 at time moment ~11000 s. This is because the bigger heat transfer coefficient and steam evaporation rate are observed when the liquid void fraction in VV vessel was 10 times smaller than just after the break. This leads to the increased steam flow rate. The water mass in draining tank was increasing continuously until the tank was filled with water (~11000 s after the break). However the condensation process did not stop on this time (steam is still going through the burst disk in to the venting pipes) and some water was collected in the horizontal pipe connected to the draining tank (Figure 12). The void fraction of the water in the horizontal pipe connected to the draining tank was ~16% at the end of the calculation.

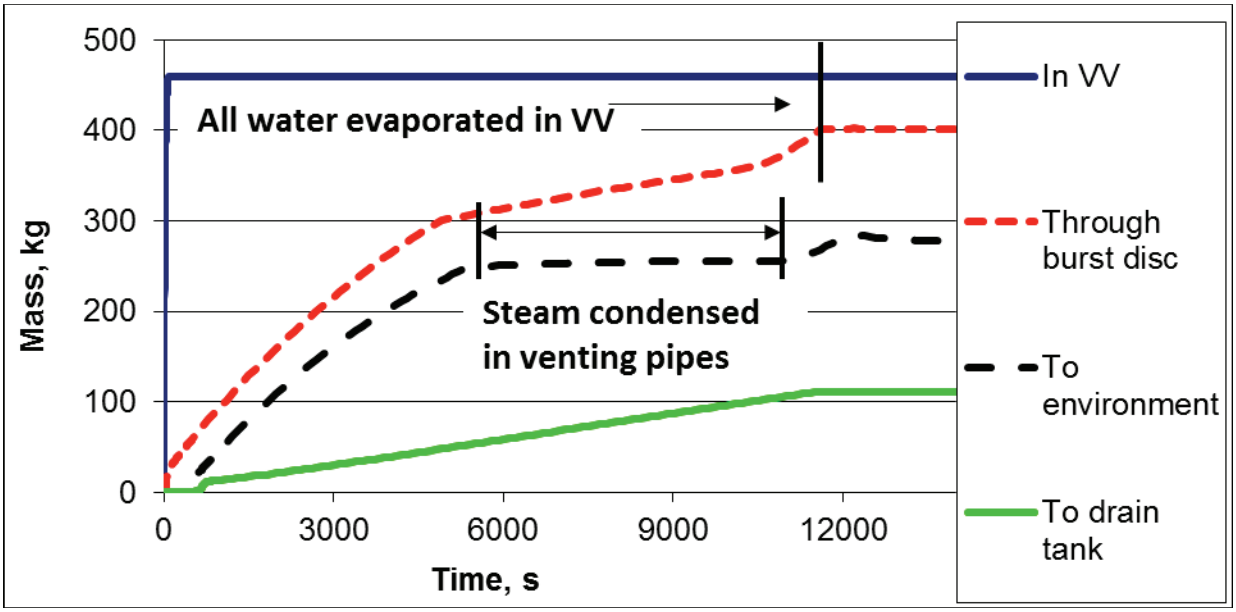


Figure 10. Integral mass of water injected in the vacuum vessel, steam through burst disk to pressure increase protection system, steam released from the venting pipes and steam condensate drained to drain tank.

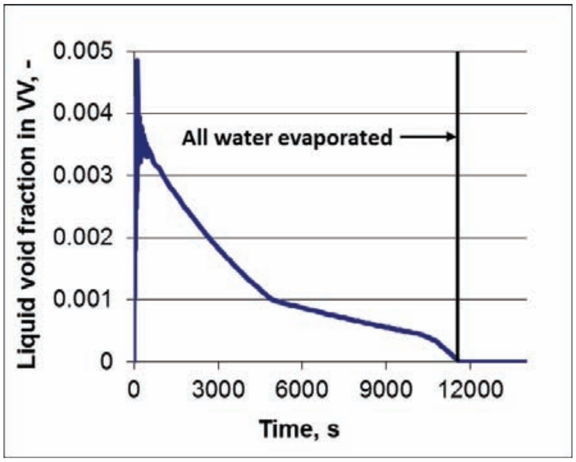


Figure 11. Void fraction of water in vacuum vessel

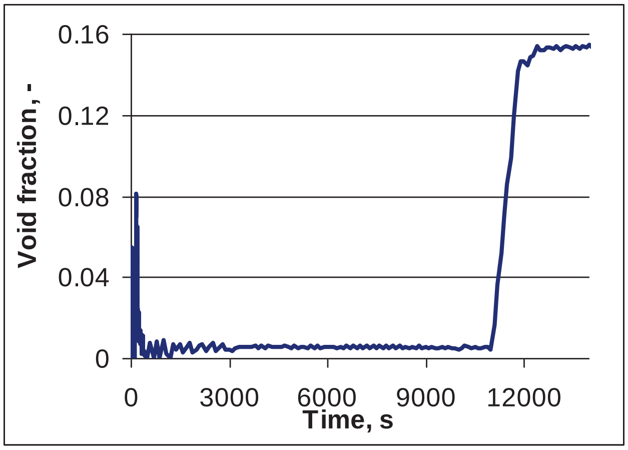


Figure 12. Void fraction of the water in the horizontal pipe connected to the draining tank.

In Figure 13 the temperature on vacuum vessel wall and the top of the venting pipe wall are presented. The vacuum vessel wall temperature are decreasing during time. The temperature of venting pipe wall, as it was mentioned, is increasing rapidly then the hot steam is going through it and temperature holds 373 K. As it is presented in Figure 10 there is time interval (5500-10500 s after the break) when all steam released through the burst disk is condensed and the discharge of steam from venting pipe to environment is equal 0 kg/s. At the time moment t = 9000 s the top part of the venting pipe walls cools down to temperature close to environment (273 K). The temperature is increasing again then the steam from the VV reaches the venting pipe (small peak of mass flow though the burst disk at time moment t=11000 s). At the end of calculation (all water in VV is evaporated and no steam release through the burst disk) the temperature of the top of venting pipe is decreasing to the environment.

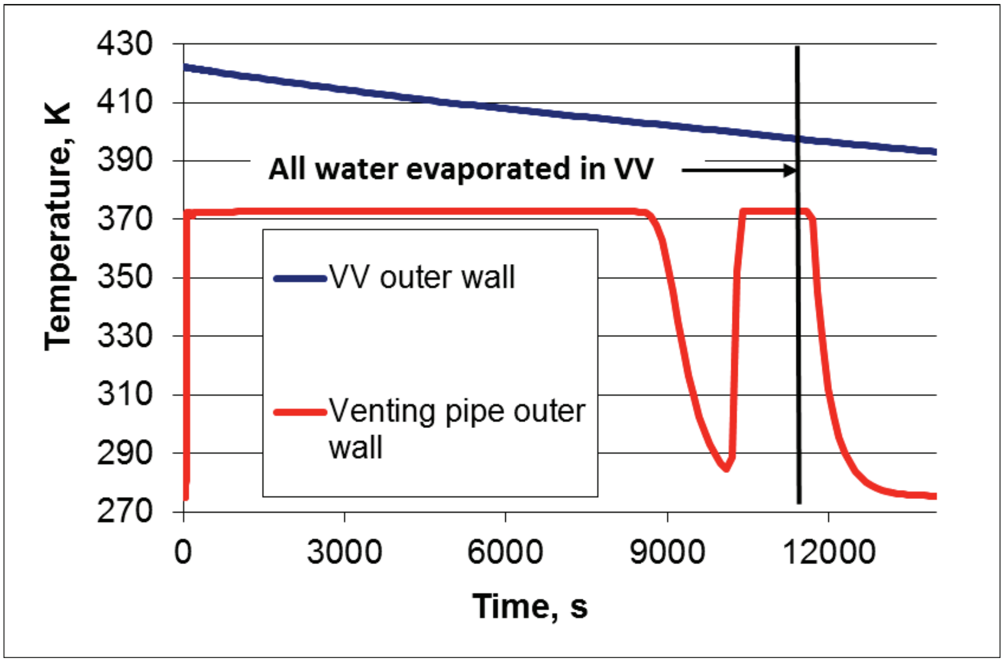


Figure 13. Vacuum vessel and venting pipe wall temperature

The obtained results show that the flow area of the burst disk (0.049 m^2) and capacity of the venting pipes are sufficient to prevent pressure increase inside the VV exceeding $1.1 * 10^5$ Pa in the case of simulated LOCA event. However, according to the calculation results the designed draining tank is too small in order to accommodate all condensed water.

4. CONCLUSIONS

1. The model of Ingress of Coolant Event (ICE) experiment facility was developed using the RELAP5 Mod3.3 code. The RELAP5 specific – limitation of minimum pressure was taken into account: the lowest possible pressure in a thermal-hydraulic system can not be below to the triple point of water. In RELAP5 simulation the lowest pressure in vacuum vessel can be in range 700 – 1000 Pa.
2. The comparison of ICE experiment calculation results with experimental data showed that, with some limitations to the initial conditions of the numerical simulation, the RELAP5 code can be applied to model the processes during water injection in to vacuum vessel of fusion devices.
3. Using the experience gained from the modelling of ICE experiments, the numerical model, including cooling system of in-vessel components, vacuum vessel and pressure increase protection system, of Wendelstein 7-X experimental fusion device was developed.
4. The provided analysis of W7-X facility showed that pressure increase protection system is sufficient to prevent pressure increase inside the vacuum vessel in the case of LOCA event.
5. The calculation results of LOCA event in W7-X facility showed that designed draining tank is too small in order to accommodate all condensed water. At the end of LOCA event the designed draining tank is full of water and void fraction of the water in the horizontal pipe connected to the draining tank was ~16%.

REFERENCES

1. *Nuclear power plants, world-wide* [interactive] [checked 2015-03-06]. Available in the internet: http://www.euronuclear.org/info/encyclopedia/n/nuclear-power-plant-world-wide.htm
2. H.-S. Bosch, V. Erckmann, R.W.T. Konig, F. Schauer, R.J. Stadler, A. Werner, “Construction of Wendelstein 7-X—Engineering a Steady-State Stellarator”, Plasma Science, IEEE Transactions, **Vol. 38**, pp. 265-273 (2010).
3. B. Streibl, “Manufacturing of the W7-X divertor and wall protection”, Proc. 23rd Symp. *Fusion Technology*, Venice, Italy (2004).
4. L. Topilski, “Consequences of the W7X in-vessel coolant pipe break at baking conditions”, 1-NBF-T0015, Max-Plank-Institut für Plasma Physik (2010).
5. C.D Fletcher, et. al., “RELAP5/MOD3 code manual user’s guidelines”, Report NUREG/CR-5535, Idaho National Engineering Lab., (1992).
6. T. Kunugi, et al., “Thermofluid experiments on ingress of coolant event”, *Fusion Engineering and Design*, pp. 67-72, (1998).
7. Private communication with Naujoks, D.; Boscary, J.; Topilski, L. 2013.