NUMERICAL ANALYSIS OF COOLANT MIXING IN THE PRESSURE VESSEL OF VVER-440 TYPE NUCLEAR REACTORS

I. Boros, A. Aszódi
Budapest University of Technology and Economics
Institute of Nuclear Techniques
boris@reak.bme.hu, aszodi@reak.bme.hu

ABSTRACT

The precise description of the coolant mixing processes taking place in the reactor pressure vessel (RPV) of pressurized water nuclear reactors has an essential importance during power operation, as well as in case of incidental or accidental conditions. In this paper the detailed CFD model of the pressure vessel of a VVER-440 type reactor and calculations performed with this RPV model are presented. The CFD model of the pressure vessel contains all the important internal structural elements of the RPV. Sensitivity study on the effect of these elements was also carried out. Both steady-state and transient calculation were performed using the CFD code CFX-5.5.1.

The results of the steady-state calculations give the so called mixing factors, i.e. the effect of each single primary loop at the core inlet. The mixing factors can be given for nominal circumstances (i.e. all main coolant pumps are working) or in case of less than six working MCPs. In order to validate the model the calculated mixing factors are compared with the values measured in the Paks NPP.

The results of the transient calculations show the dynamical behavior of the primary coolant. With these calculations coolant mixing processes during boron dilution transients or during transients caused by enter of cold water into the core can be analyzed. The mixing of coolant fed by the high pressure injection system into the pressure vessel has been also examined. This calculations are particularly interesting, since the temperature of the coolant fed by these safety systems is much lower than the temperature inside the RPV, so it has an important role on thermal stress of the vessel.

1. INTRODUCTION

The distribution of the temperature and boron concentration has major influence on the neutron-kinetical behaviour of the reactor and the thermal stresses affecting the pressure vessel. Therefore coolant mixing processes taking place in the reactor pressure vessel (RPV) are of primary importance during normal power operation, as well as at incidental conditions.

The possibilities for the determination of coolant mixing with measurement are very limited because the experimental facilities are very expensive and difficult to build. In the past usually analytical methods were used, but their applicability is also restricted because of the complicated geometry of the pressure vessel. The Computational Fluid Dynamics (CFD) methods provide an effective tool for the 3D simulation of the coolant mixing processes both for normal operation and incidental conditions. The advantages of CFD codes are the lower costs (compared with experimental methods) and the accuracy (compared with analytical methods), while the large computational demand and long simulations are the disadvantages.

In order to determine these mixing processes inside the RPV, a detailed 3D model of the vessel has been built and steady-state and transient calculations were performed. The CFD code CFX-5.5.1 [1] has been used for the simulation presented in the paper.

2. THE DETAILED CFX MODEL OF THE RPV

The CFX model of the pressure vessel extends from the inlet nozzles to the outlet nozzles, and it includes the following elements: inlet and outlet nozzles, downcomer, lower plenum, model of the core (see Fig. 1 and Fig. 2).

From the point of view of the coolant-mixing there is a special feature of the RPV: the existence of the control rod chamber. In this region, under the core there are 37 brake tubes, into which the fuel assemblies (coupled underneath the control rods) can slide in, when the control rods enter into the core. There are planar perforated plates under and above the control rod chamber. These plates and the large volume of the control rod chamber causes very good mixing in accidental conditions (in case of asymmetric flows). The wall of the fuel assemblies is closed, so inside the core there is no cross-flow. (Because of this the core can be modelled with body forces, which allow flow only in the vertical direction.)

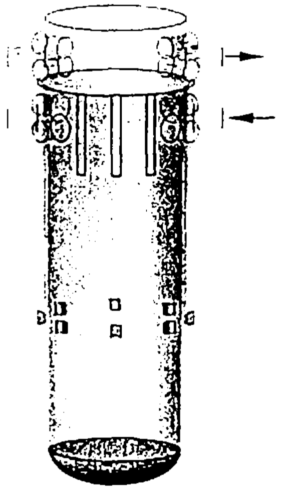


Figure 1. Geometry of the RPV of the VVER-440 type reactor

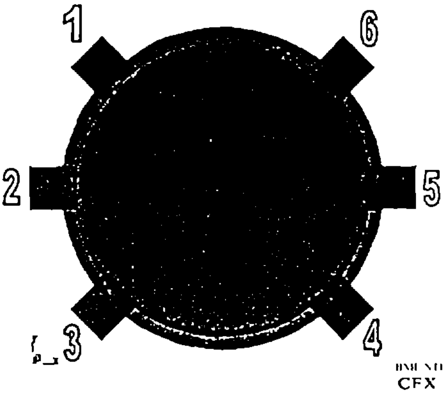


Figure 2. Arrangement of the inlet nozzles

The main thermal-hydraulical parameters used for the simulation during normal operation the followings:

- mass flow rate in one loop: 1460 kg/s,
- inlet temperature of the coolant: 266.6 °C,
- outlet temperature: 297.3 °C,
- pressure drop through the RPV: 2.8 bar (1.9 bar through the core),
- density: 768 kg/m3,
- dynamic viscosity: 1.E-4 kg/ms,
- specific heat capacity: 5115 J/kgK,
- thermal conductivity: 0.596 W/mK,
- thermal expansivity: 2.1E-3 1/K.

Calculations performed with a simplified model of the RPV showed the necessity of buildin of a detailed RPV model including all the important structural elements inside the vessel [4 During the model development it has to be investigated, which structural elements a essential for the calculations, or what approximation can be done instead of exact modellin The limits of the computational capacity also have to be taken into account.

The effects of the main structural elements upon the coolant mixing have been examine separately. A parameter study (with steady-state test calculations) was carried out to find o whether the elaboration of the model of a structural element is neccessary. The paramet study included the achieving of simplified and detailed models of the element and th performing of steady-state test calculations with these models. The parameters of the te calculations were fitted to the thermal hydraulic parameters belonging to the normal operatio (see above). For the calculations the standard k–ε turbulence model has been used. Th number of the tetrahedral volume elements in the volume mesh was between 100 000 and 700 000.

The following internal structural elements have been investigated [5]:

<u>Alignment drifts:</u> there are eight co-planar alignment drifts – enclosing an angle of 45 degree with each other – in the RPV seated on the vessel wall at the cleavage plane (see Fig. 3, 4 Their task is to fix the reactor pit coaxial to the RPV.

Figure 3. Photo of an alignment drift

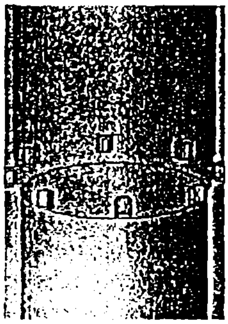


Figure 4. The arrangement of the alignment drifts in the model

There are alignment drifts under all inlet nozzles and two additional azimuthal one between the outermost nozzles. The drifts fill a noticeable part of the flow region, so they have impact on the coolant flow in the downcomer. On the drifts there are vertical holes.

The results of the calculations showed (see **Figure 5**) that under the drifts there is a region in which the coolant is stagnating (with near to zero effevctive velocity). This phenomenon cannot be experienced at the bottom of the downcomer (i.e. the effect of the drifts is local during normal operation), but these stagnating regions can play an important role during transient processes, at which the coolant mixing in the downcomer is very important (e.g. at Pressurized Thermal Shock).

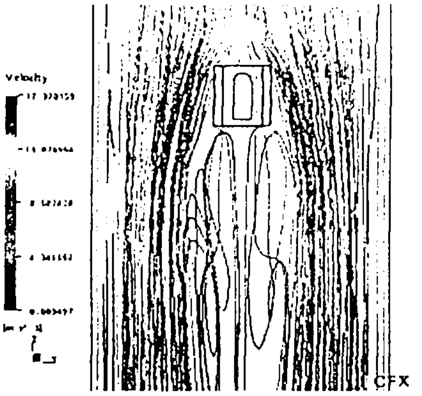


Figure 5. Stagnating region under the drift (streamlines coloured by velocity)

The baffles of the hydro-accumulators: there are three semicircular profile baffles, each 1.9 m long in the upper part of the downcomer, welded to the vessel wall (see **Figure 6**). The task of the baffles is to guide the water of the hydro-accumulators downwards in the downcomer during an incident. However, the baffles affect the mixing of the normal primary coolant as well, because the baffles guide it also downwards. This phenomenon can have an important influence on nuclear safety. (E.g. when lower temperature coolant enters the vessel from one of the neighboring nozzles.)

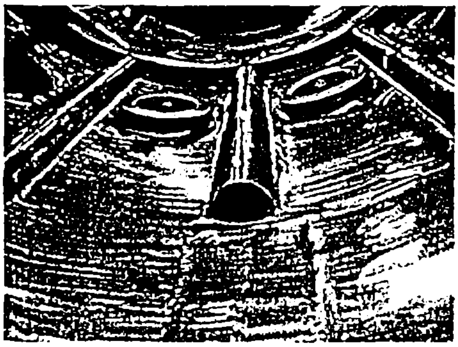


Figure 6. Baffles of the hydro-accumulators

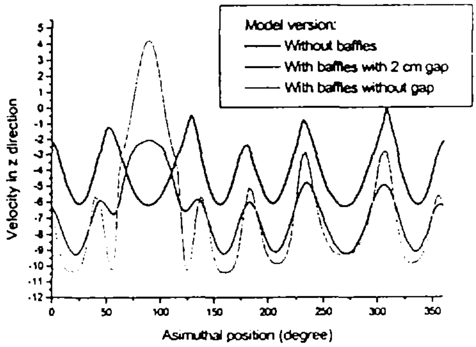


Figure 7. Calculated coolant velocity in z direction in the downcomer (r=1.7 m, z=7 m)

According to the performed calculations the baffles considerably influence the coolant flow the downcomer (see Figure 7). Under the baffles there is a relatively large stagnating regio where some eddies can be observed. The baffles effect the coolant flow even at the core inle These results are in good agreement with the experimental ones: the effect of the stagnatir region can be observed on the mixing factors measured at Paks NPP [3].

Elliptical perforated plate: the elliptical perforated plate, situated in the lower plenum has a effect on the coolant flow: it smoothes the differences in the velocity of the coolant. Th elliptical plate of a VVER-440 type reactor contains 1344 holes with a diameter of 4 cm (se **Figure 8**). The plate has been modelled with body forces. In order to find the suitable bod forces to describe the effect of the plate on the coolant flow, a simplified model of th elliptical plate has been built.

With the help of the performed steady-state calculations, the velocity profile after the plat and the pressure drop on the plate has been determined. The small jets after the perforate plate are insignificant at a distance of 10 cm after the plate (see **Figure 9**). The calculate values have been used to specify the parameters of the body forces, which describe the flo resistance of the elliptical plate with cosine-functions(see **Figure 10**), and which is used in th whole RPV model (see next chapter).

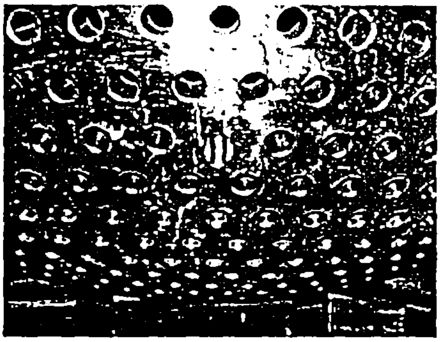


Figure 8. Elliptical perforated plate

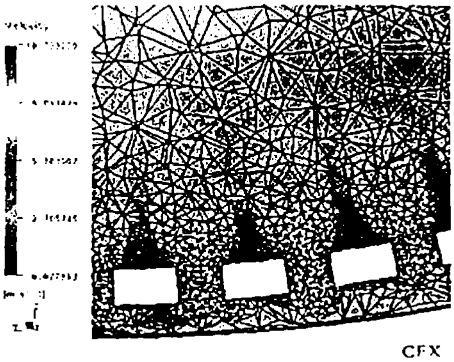


Figure 9. Velocity field near to the plate

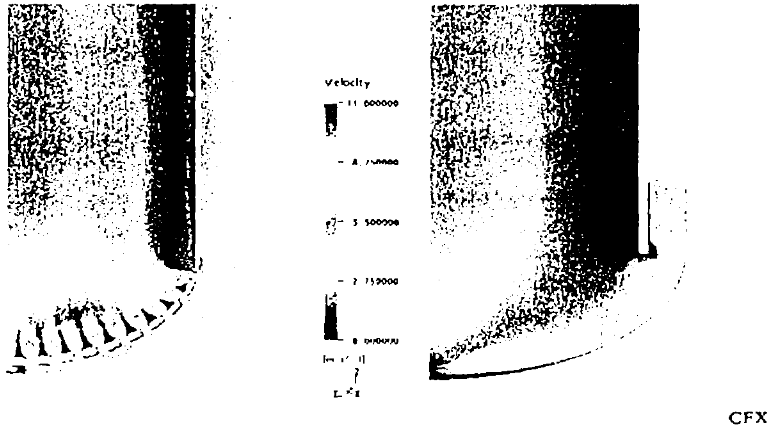


Figure 10. Velocity field near to the elliptical plate in the case of the detailed and simplified plate model

Brake tube chamber: the 37 brake tubes under the core, have also an important influence on the coolant mixing in the RPV. The coolant flows into the brake tubes not through the bottom of the tubes, but through small holes situated on the lower part of the tube mantle (see Figure 11). In the calculations introduced below, the brake tubes were modelled by simple tubes, cut out from the flow domain.

At the moment the investigation of different brake tube chamber models, the building of the direct model of the control rod chamber and running of test calculations are in progress.

These structural elements form the detailed model of the RPV (see Figure 12). The core and the structure of the control rod drives are not modelled yet. This model is suitable to investigate mixing processes, which are particularly important in the downcomer. Both steady-state and transient calculations were performed with this model. In the followings some results of these calculations will be introduced.

Figure 11. Photo of the brake tube chamber

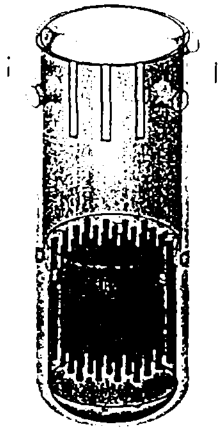


Figure 12. The detailed CFD model of the reactor pressure vessel

3. CALCULATIONS WITH THE RPV MODEL

3.1. Investigation of the mixing factors during normal operation

With the RPV-model introduced above, steady-state calculations were performed to examine the coolant mixing in the RPV with the help of additional scalar variables. In the centre of the interest were the mixing factors, which give the share of one primary loop at the inlet of a certain fuel assembly, so it demonstrates the coolant mixing in the vessel from the inlet up to the core. The calculation of the mixing factors give possibility for the validation of the model because experimental values are available, which was measured at the NPP Paks [3].

For the calculations, the nominal mass flow rate of the primary loops was given as inlet boundary condition, and the pressure above the core as outlet boundary condition. Six different scalar components were defined. The concentration of a scalar component in one loop was 1 (kg/m^3), in the other loops 0. So the concentration of the scalar components in t different fuel assemblies at the core inlet gave immediately the mixing factors for each loop

The model of the pressure vessel contained 2 700 000 volume elements. The brake tubes w closed out from the flow domain. The results show qualitatively good accordance with t measured data [3], however numerical difference can be observed (see Figure 13). The reas of this – with high probability – is the lack of the lower perforated plate of the brake chamb In this consideration the modification of the model is planned.

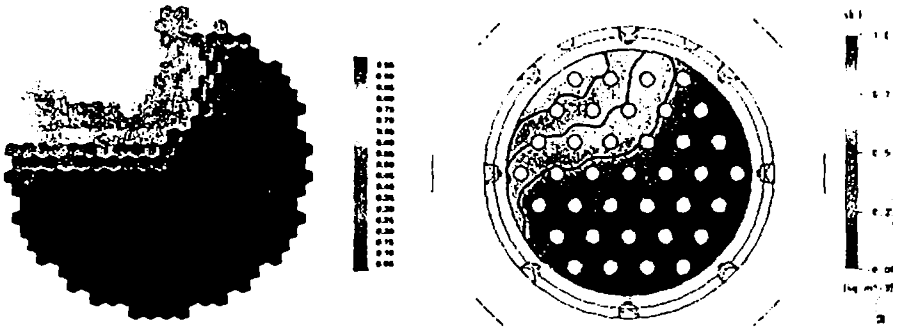


Figure 13. Measured (left) [3] and calculated (right) mixing factors of the loop No. 1

3.2. Investigation of the entry of coolant with lower temperature into the RPV

In order to demonstrate the effect of the structural elements the simulation of the entering o coolant with lower temperature into the pressure vessel have been performed. As initia condition the coolant flow was assumed to be nominal in the RPV (123 bar pressure, 150 kg/s mass flow rate per MCP). The beginning of the transient is the entry of coolant into th RPV on the loop No. 6 with a temperature of 333 K instead of the original 539 K (see Figur 14).

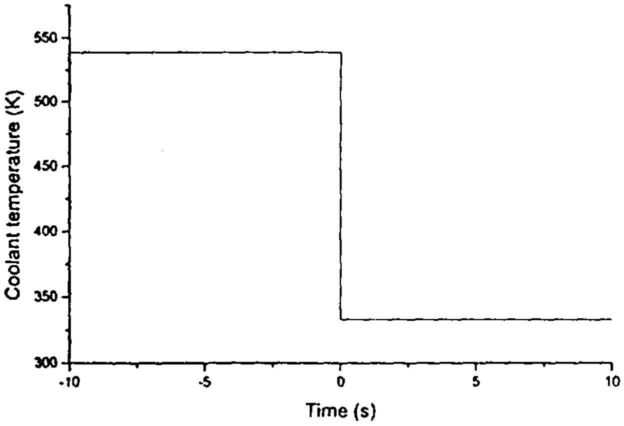
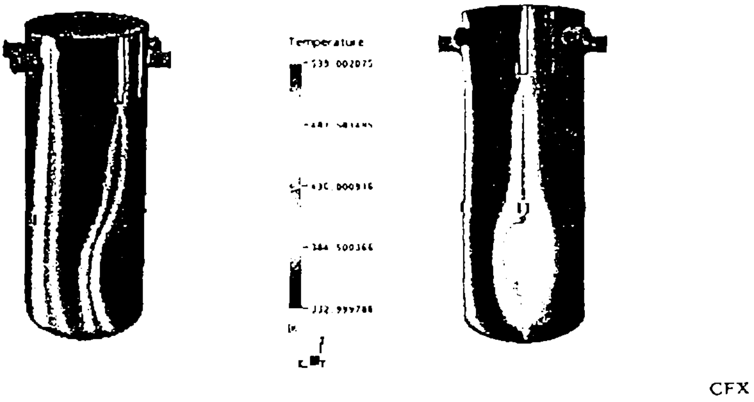


Figure 14. Coolant temperature at the inlet nozzle No. 6

The investigated geometry contained only the downcomer and the lower plenum with the internal structural elements (the outlet boundary was the core inlet). This model contained 1 900 000 volume elements. The total time of the simulation was 10 seconds with a timestep of 0,02 s.

The results show that the stagnating region – the common effect of the HA-baffles and the alignment drift – under the baffles fills up very slowly with colder coolant: after 10 seconds the coolant temperature in this region is 480 K (see Figure 15, 16 and 17).



t = 5 s t = 10 s

Figure 15. Coolant temperature in the downcomer

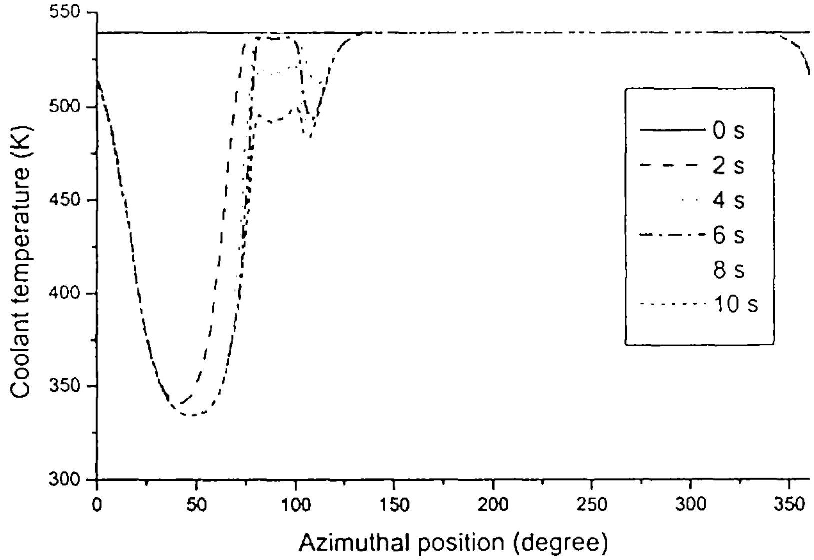


Figure 16. Azimuthal distribution of coolant temperature in the downcomer (*z*=3.5 *m*, under the alignment drifts)

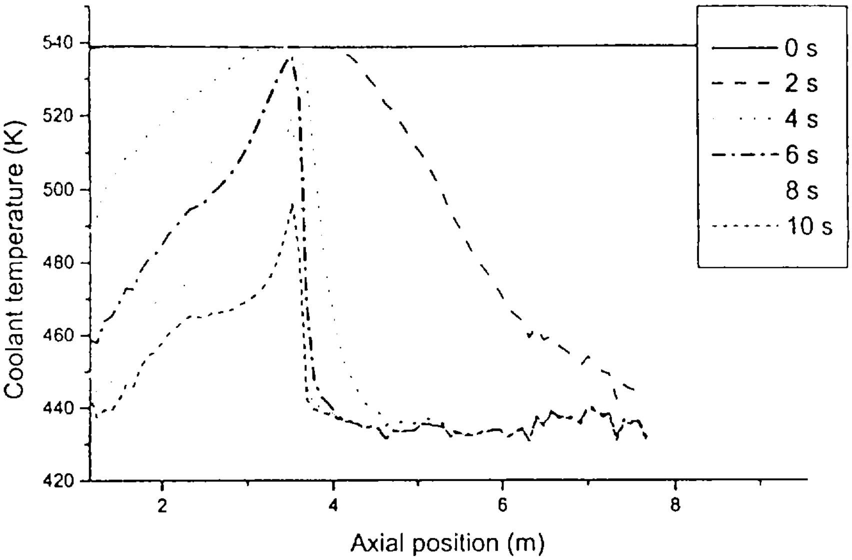


Figure 17. Axial distribution of coolant temperature in the downcomer (r=1.7 m, in the line c the middle HA-baffle)

3.3. The investigation of the accidental start-up of the High Pressure Injectio System

The accidental start-up of the High Pressure Injection System (HPIS) has been investigate with transient calculation with the detailed RPV model. This transient has in practice very lo probability, but it is very interesting from the point of view of the thermal stresses of th vessel wall. (The temperature of the coolant transferred into the RPV by the HPIS is muc lower than the temperature of the primary coolant.)

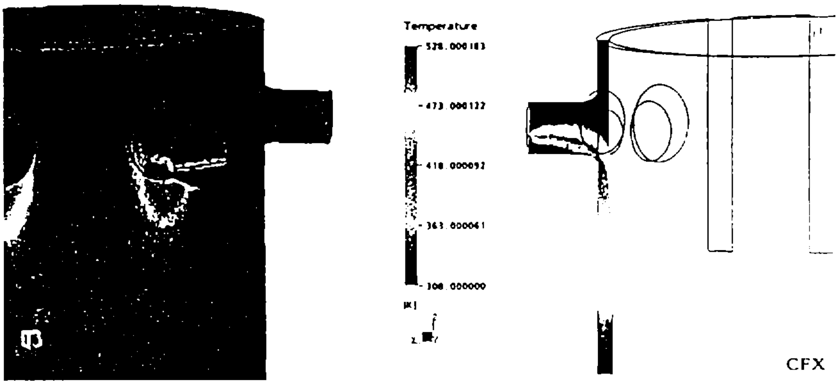
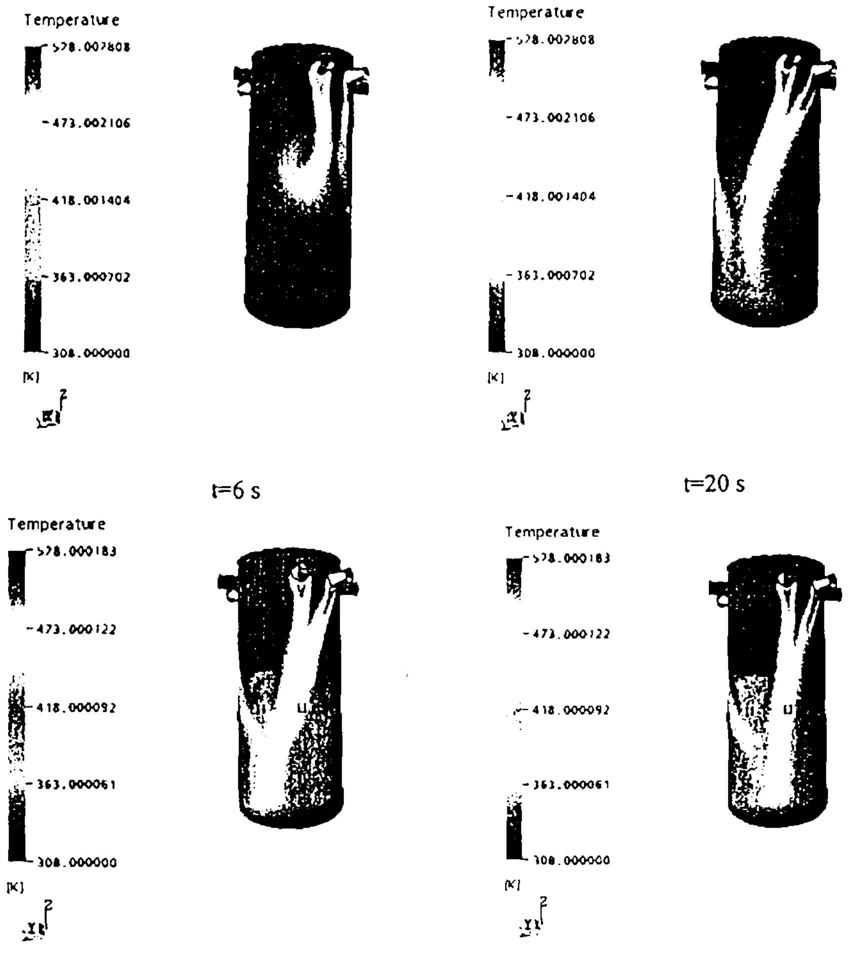


Figure 18. Inlet boundary of the transient simulation

The assumed initial conditions of the transient: the reactor is in hot shut down condition, i.e. the main coolant pumps are stopped, the coolant mass flow rate in the loops is 50 kg/s, which corresponds to the natural circulation developing in the primary circuit. The temperature of the coolant is 255°C, the pressure is 12.3 MPa.

The transient is initiated by the accidental start-up of the pumps of the HPIS. The pumps transport coolant with lower temperature (35°C) into the cold legs of the primary loops No. 2, 3 and 5. (The pumps of the HPIS are connected to these loops.) The mass flow rate of the HPIS-coolant is 20 kg/s which corresponds to the real characteristic of the pumps of the HPIS at this pressure.



t=6 s

t=20 s

t=40 s

t=60 s

Figure 19. Temperature distribution in the downcomer during the transient

At the inlet nozzles the total thermal stratification of the coolant was assumed (see Figui 18). In the calculation buoyancy was taken into account, and because of the large coolai temperature differences the thermal parameters of the coolant were approached by polynon as a function of the temperature [2]. The duration of the simulation was 80 s, with a timeste of 0.2-0.5 s.

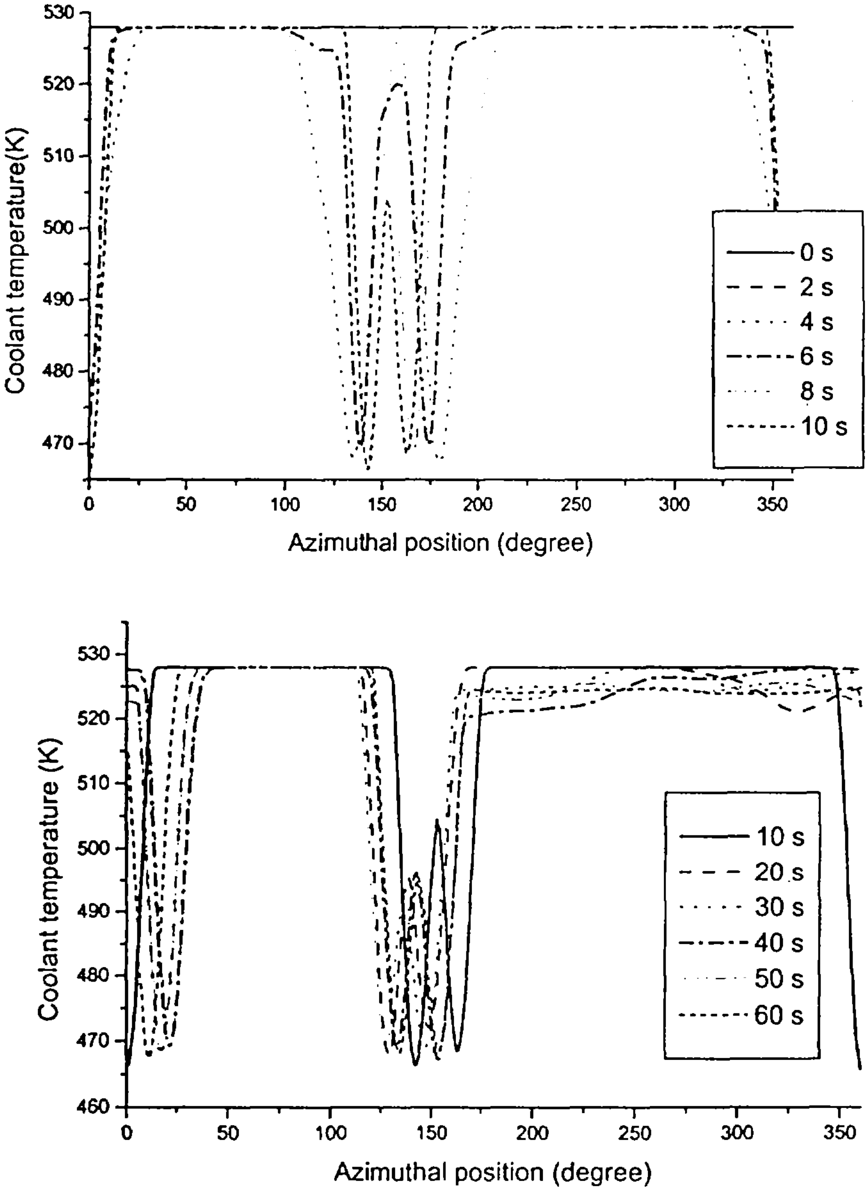


Figure 20. Azimuthal temperature distribution in the downcomer (z=7 m)

According to the performed calculation the coolant with lower temperature flows in plumes downward in the downcomer (see **Figure 19**, **20** and **21**). The coolant temperature at the wall of the vessel directly under the inlet nozzle is about 130°C. During the transient the plumes are adhering and oscillating together while the vessel is slowly filled with cold coolant.

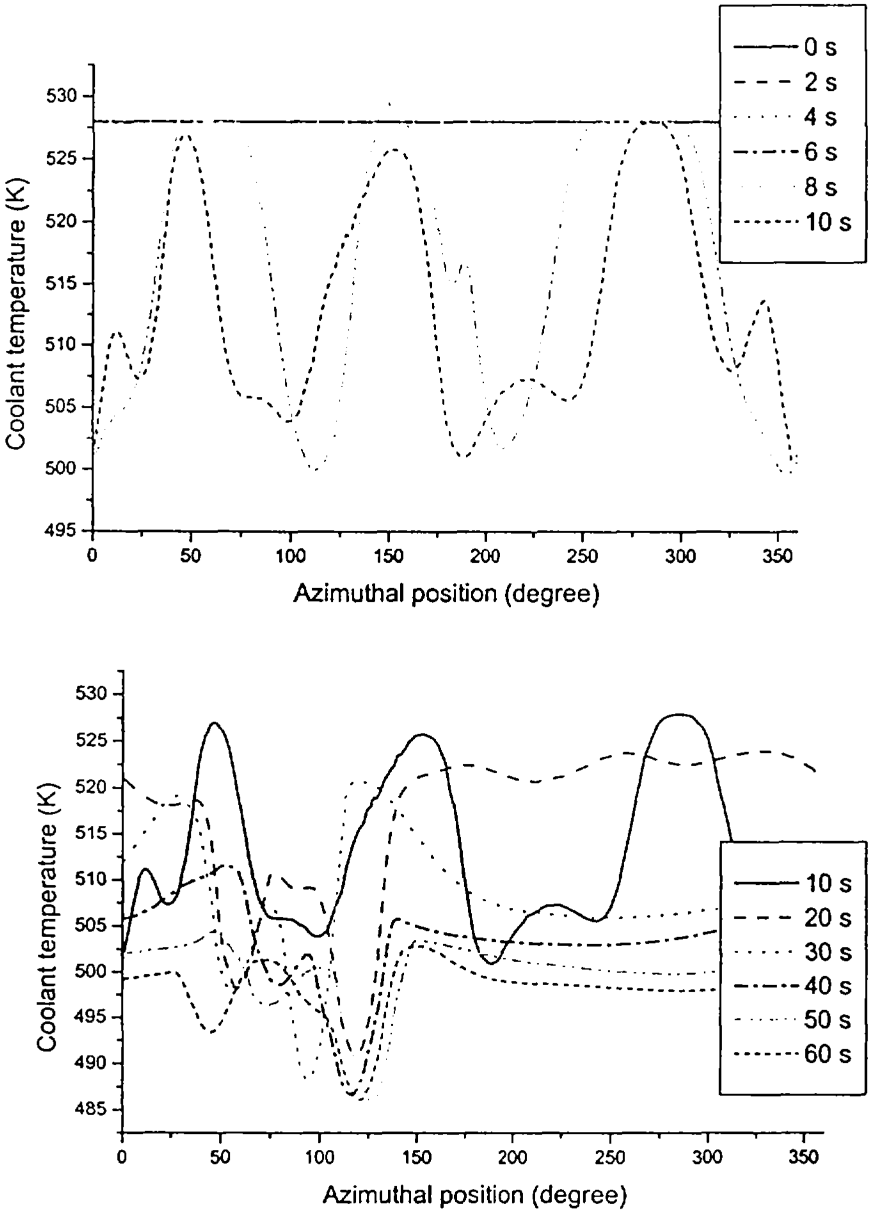


Figure 21. Azimuthal temperature distribution in the downcomer (z=3.5 m)

3.4. Investigation of a boron dilution transient

In order to investigate the effect of the control rod chamber on the coolant mixing a boron dilution transient has been simulated with the detailed RPV model. As initial condition the reactor was assumed to be in hot shot down condition, with stopped MCPs. (There is no natural circulation assumed in this case.) The coolant temperature is 266 °C, the pressure is 123 bar. The boron dilution occurs during the start-up of the first MCP: the pump transports about 10 m^3 diluted coolant into the RPV in 8 seconds (see **Figure 22**). The mixing of the diluted slug was demonstrated with the help of a mixing scalar component.

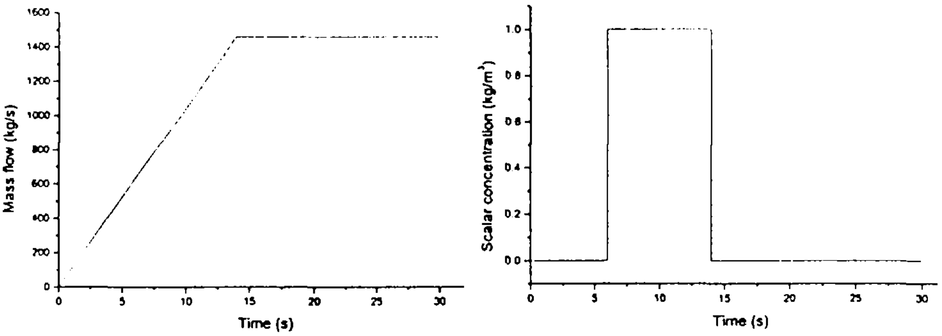


Figure 22. Mass flow and scalar concentration during the transient in the loop No. 1

The investigated model included the control rod chamber as well. The volume mesh of the flow domain contained 2 200 000 volume elements. The transient simulation was carried out for 30 seconds with a timestep of 0.1 s.

The results show that the coolant flows in the downcomer in two main jets, while under the affected inlet nozzle the borated coolant remains in a stagnating region. The effect of the HA baffles and the alignment drifts can be also observed in the coolant flow. According to the calculation the diluted slug reaches the core in the 17. s of the simulation (see **Figure 23, 24**).

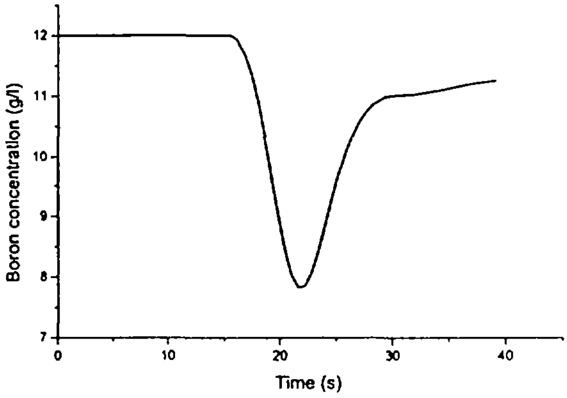
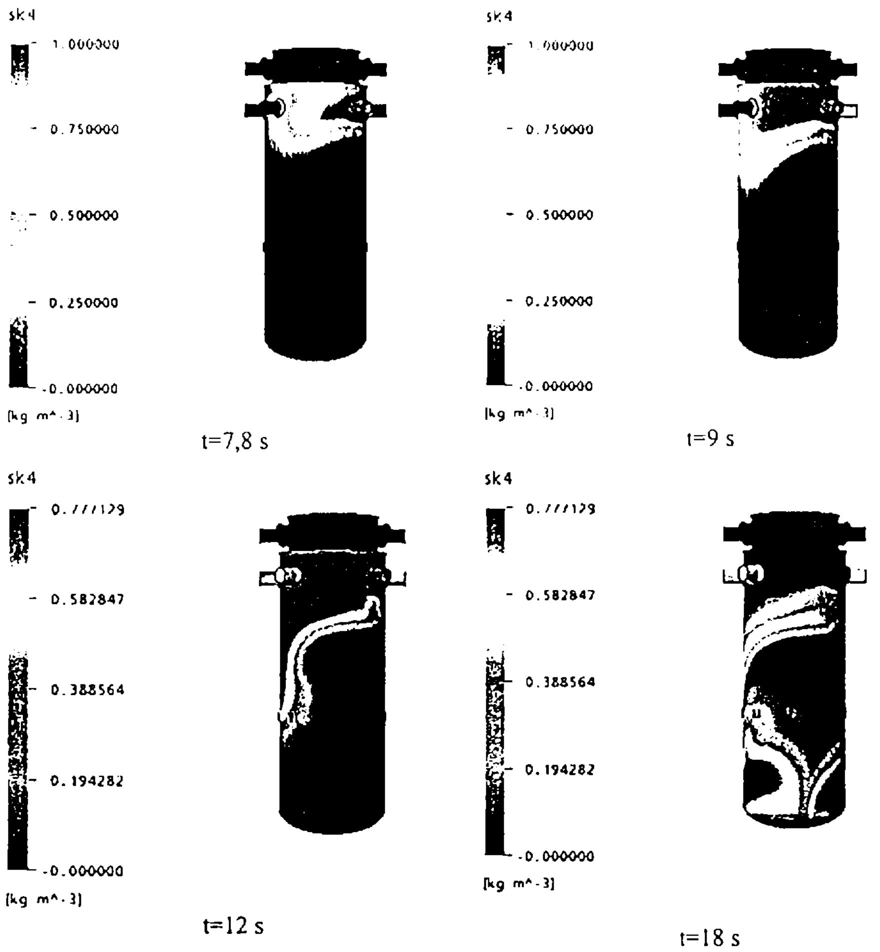


Figure 23. Average boron concentration at the core inlet

One of the most interesting results is the boron concentration distribution at the core inlet. Due to the effect of the brake tubes (they guide the coolant upwards in the control rod chamber) the diluted slug reaches the core in one plume, at the centre of the core inlet (see **Figure 25**).



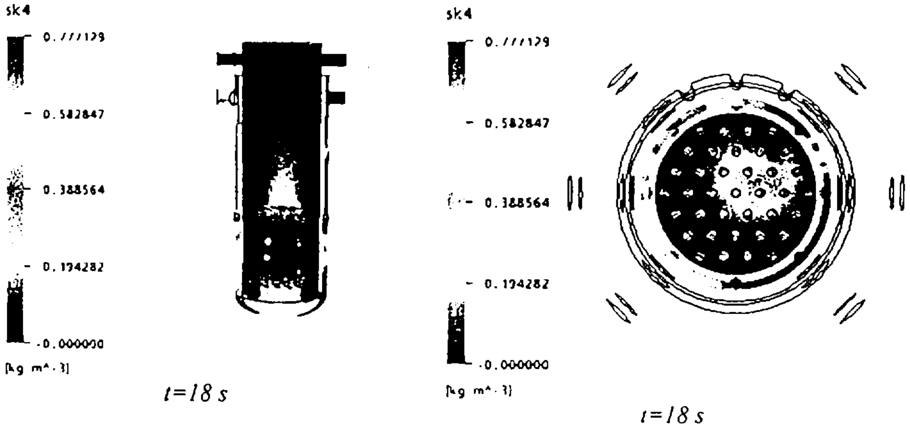
t=7,8 s

t=9 s

t=12 s

t=18 s

Figure 24. Scalar concentration in the downcomer



t=18 s

t=18 s

Figure 25. Scalar concentration in the control rod chamber and in the core

4. SUMMARY

In the article the present state of the model development of the RPV of a VVER-440 typ reactor was presented. To fulfil the parameter study of the main structural componen detailed models were developed, that can determine the effect of these elements on th coolant flow. Compared to the available experimental data the calculations give qualitatively good result.

For the performed calculations the following computers were used:
- a PC with 1800 MHz Intel Pentium IV. processor, 2 GB RAM, Windows NT OS,
- two PCs with 2400 MHz dual Intel Xeon processor, 2 GB RAM, SuSe Linux OS.

In the future the model has to be extended up to the outlet nozzles. With the help of th detailed model we intend to perform calculations on PTS and boron dilution transients. large interest is shown in the coupling of CFD methods with other computer codes, e.g. wi 1D system codes or with 3D neutron-kinetic codes.

5. ACKNOWLEDGMENTS

The authors acknowledge the help of I. Kiss and S. Máté in the Maintenance Training Cent of the Paks NPP and the data provided by J. Elter (Paks NPP).

6. REFERENCES

[1] "CFX-5.5.1 User Manual", AEA Technology, 2002

[2] M. A. Mihejev: "Basics of practical heat transfer calculation" (in Hungaria Tankönyvkiadó, Budapest, 1990

[3] J. Elter: "Experimental investigations and computational method for determination inlet coolant temperature for V-213 reactor type", PhD thesis (in Hungaria Budapest, 1993

[4] I. Boros, A. Aszódi: „Numerical Analysis of Coolant Mixing in the RPV of VVE 440 Type Reactors with the Code CFX-5.5.1", IAEA Technical meeting on Use Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Syste including Containment, J4-TM-25218, Pisa, Italy, 11-14 November 2002, (Proceedings

[5] I. Kiss, S. Máté: Maintenance Training Centre of the Paks NPP, 2002, perso communication.