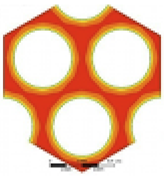
Portál pre odborné publikovanie ISSN 1338-0087

Flow simulations in fuel rod bundle

Jakubec Jakub · Elektrotechnika, Študentské práce

16.12.2011



This paper is focused on flow simulation in fuel rod bundle using code ANSYS CFX 12. ANSYS CFX, which is commercial computational fluid dynamics (CFD) code, applies method of finite volumes to solve Navier-Stokes equations describing laminar and turbulent behavior of fluids with Reynolds-averaged Navier-Stokes equations (RANS) method. In order to reach optimal mesh resolution for simulation of flow in fuel rod bundle, simple subchannel has to be investigated first. Next calculations are dealing with mixing processes in fuel rod bundle section with spacer grid and without it.

1. Intorduction

Detailed knowledge of the hydraulic processes is very important in the case of fuel rod bundles of nuclear reactors from the design and safe operation point of view. Investigations in this field can help to upgrade possibilities of modern types of reactors. Experiments and CFD codes can help to accomplished these tasks. Main goal of this work is to develop validated CFD models for some parts of the fuel assembly and in future fully functional and validated CFD model for whole fuel assembly.

2. Navier-Stokes equations

CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. ANSYS CFX is CFD code that applies method of finite volumes to solve Navier-Stokes equations. Navier-Stokes equations are differential equations describing motion of fluid substances. They consists of differential equations of mass conservation, momentum conservation and energy conservation.

Equation (1) expresses mass conservation law, which is often called the equation of continuity because it requires no assumptions except that the density and velocity are continuum functions. That is, the flow may be either steady or unsteady, viscous or frictionless, compressible or incompressible [1]. It is partial differential equation involving the derivatives of density (ρ) and axis velocity (u, v, w).

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad (1)$$

Momentum conversation law (2) is a fundamental law of nature, and it states that if no external force acts on a closed system of objects the momentum of the closed system remains constant. One of the consequences of this is that the center of mass of any system of objects will always continue with the same velocity unless acted on by a force from outside the system [2]. Variables in equations (2) are density (ρ), pressure in moving fluid (p), coefficient of viscosity (μ) and time (t).

$$\rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) = \rho\frac{du}{dt}$$
$$\rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) = \rho\frac{dv}{dt}$$
$$\rho g_z - \frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) = \rho\frac{dw}{dt} \tag{2}$$

Energy conservation law (3) is valid for a newtonian fluid under very general conditions of unsteady, compressible, viscous, heat-conducting flow, except that it neglects radiation heat transfer and internal sources of heat that might occur during a chemical or nuclear reaction [1]. Variables in equations (3) represents density (ρ), pressure in moving fluid (p), internal energy (û), velocity (V), nabla (∇), thermal conductivity (k), temperature gradient (∇T) and time (t).

$$\rho\frac{d\hat{u}}{dt} + p(\Delta.V) = \Delta(k\Delta T) + \phi \tag{3}$$

3. Subchannel simulations

To study the effect of the spacer grid and mixing processes in fuel rod bundle section of WWER-440 small subchannel was created. Subchannel was chosen for this calculations because it has less elements and nodes than rod bundle section and has lower requirements for computing performance and solving time. So the subchannel was used to find out the most appropriate mesh resolution properties for further calculations.

3.1. Model describtion

Subchannel was created 6 mm long in triangular configuration with fuel rod diameter equal to 9,1 mm and pitch equal to 12,3 mm (Fig.1). Resolving this geometry was made with four different types of meshes to study influence of mesh resolution (Fig.2) in results. Main characteristics of the mesh are in Table.1.

All computations were made with ANSYS CFX 12.0. ANSYS CFX code applies method of finite volumes to solve Navier-Stokes equations with RANS method. Boundary conditions for the surface of rods was defined as no slip smooth wall, symmetry boundary condition was applied on the symmetry planes and to solve fully developed flow, periodic interface was applied on remaining planes (Fig.1). BSL Reynolds stress was used as turbulent model with 10% intensity. As fluid was used water with properties from the database IAPWS-IF97 with absolute pressure 12,3 MPa and temperature 265ºC. Momentum source in axial direction was 8100 kg m^{-2} s^{-2} to gain Reynolds number in range 220-227*10^3 as it’s in fuel assembly.

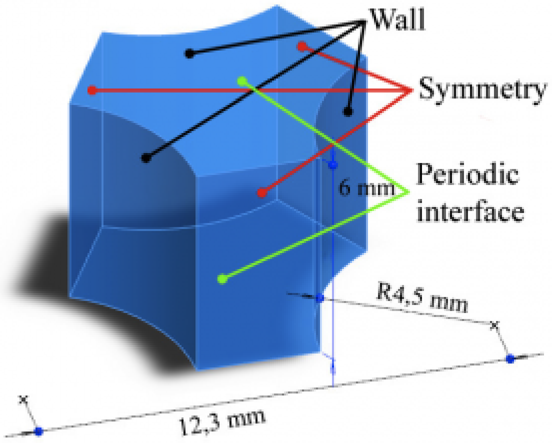


Fig.1. Subchannel with boundary conditions

Table. 1. Main mesh characterictics

MESH	A	B	C	D
number of nodes	10759	6979	4207	3493
number of elements	8700	5544	3264	2682

3.2. Mesh sensitivity study

In order to study influence of mesh quality on the results four different mesh with same boundary conditions were investigated. Sweep meshing method with inflation on walls to better capture effect of laminar boundary layer was chosen for this calculations. Tested mesh resolutions and results can be seen in Fig.2 and Fig.3. For further calculations mesh B as most appropriate mesh was chosen. Mesh A was also acceptable but has higher mesh resolution than mesh B and it would have higher requirements in next calculations. Results were compared with work of S.Tóth and A. Asyódi [3].

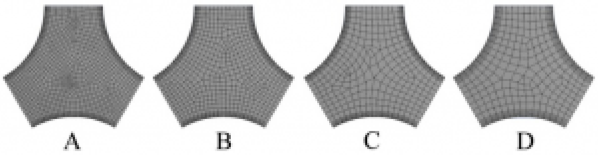


Fig.2. mesh resolutions

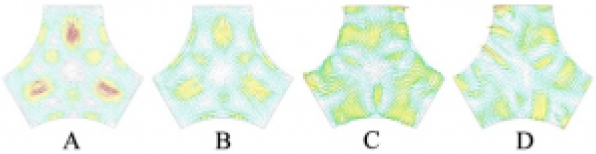


Fig.3. Tangential velocity projections

Correctness of results were confirmed with Reynolds number which value was equal 222601 and fits to range of values from fuel assembly. Average flow velocity was 3,47 m.s^{-1} with zero flow velocity close to walls what proofs theory of laminar boundary layer.(Fig.4)

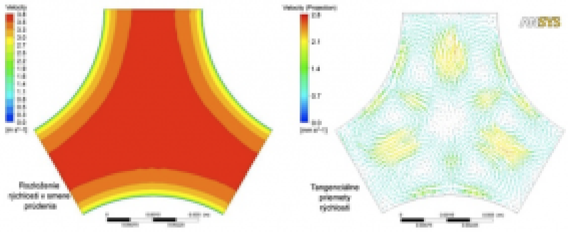


Fig.4. Distribution of flow velocity in subchannel

4. Fuel rod bundle simmulations

Two models were built to study influence of spacer grid on mixing processes in rod bundle. Both models were 250 mm long in triangular configuration with fuel rod diameter equal to 9,1 mm and pitch equal to 12,3 mm with six fuel rods. Model B had extra 10 mm long spacer grid in the middle (Fig.5). Before to simulate rod bundle flow, there had to be created subchannel 6mm long with identical base plane as model A and B with the same mesh properties and boundary conditions. Velocity from this calculation was used as velocity inlet profile for rod bundle calculations with spacer grid and without it.

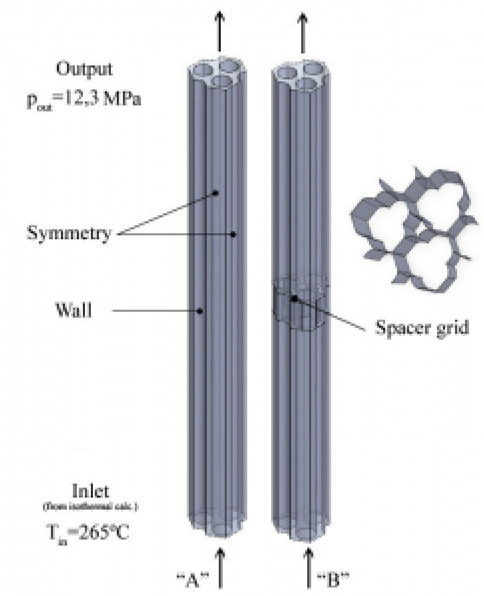


Fig.5. Model A and Model B with boundary conditions

Meshing properties of both models were the same as it was in Subchannel (Fig.5). In case of model B with spacer grid inflation method was used on grid walls to capture effect of laminar boundary layer (Fig.6).

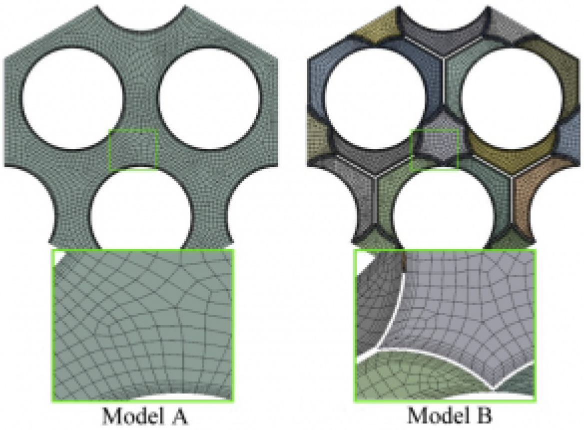


Fig.6. Mesh details of model A and model B

4.1. Results of calculations

Reynolds number in both results were approximately the same so it's plausible to compare calculations results. On Fig.7 and Fig.8 there are shown velocity distributions on outlet and tangential velocity profiles of model A and B. Main results values are shown in Table. 2.

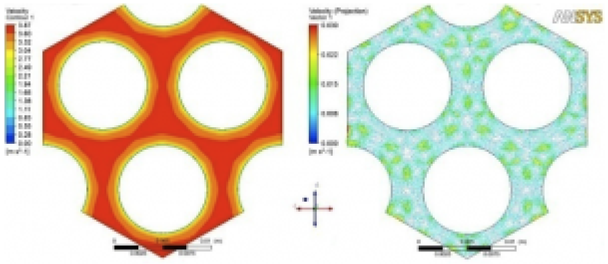


Fig.7. Velocity distribution and tangential velocity profiles on the outlet plane of model A

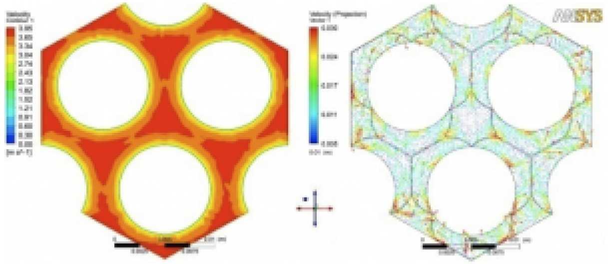


Fig.8. Velocity distribution and tangential velocity profiles on the outlet plane of model B

Table.2. Main results values

	Model A	**Model B**
Average outlet velocity	3,48 ms^{-1}	3,48 ms^{-1}
Maximum outlet velocity	3,87 ms^{-1}	3,95 ms^{-1}
Reynolds number	223243	223169

There are some differences in results but most of them are insignificant small. The most important differences between model A and B are tangential velocity profiles (Fig.9 and Fig.10). Maximum tangential velocity is 0,03 ms^{-1} what is really small value to have some influence on axial flow.

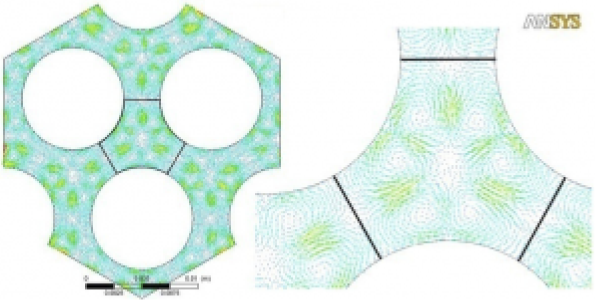


Fig.9. Tangential velocity profiles on the outlet plane of model A (zoom)

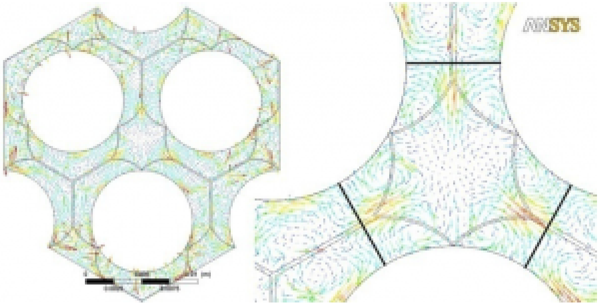


Fig.10. Tangential velocity profiles on the outlet plane of model B (zoom)

5. Conclusions

Spacer grid in fuel assembly has important task to hold fuel rods in exact distance to avoid asymmetric power distribution. From hydraulic point of view spacer grid has no significant influence on flow in fuel assembly so it doesn't represent any hazard on safe operation of nuclear reactors.

References

1. Frank M. White: Fluid Mechanics: Fourth Edition
2. WIKIPEDIA, http://www.wikipedia.org/
3. S.Tóth and A. Asyódi: CFD analysis of flow field in a triangular rod bundle

Coauthor of this paper is doc. Ing. Vladimír Kutiš, PhD., Department of Mechanics, Faculty of Electrical Engineering and Information Technology, Slovak University of Technology in Bratislava, Ilkovičova 3, Bratislava 812 19

Práca bola prezentovaná na Študentskej vedeckej a odbornej činnosti (ŠVOČ 2011) v sekcii Aplikovaná mechanika I, ISBN 978-80-227-3508-7