COMPUTATIONAL FLUID DYNAMIC APPROACH FOR CANDU6 AND ACR1000 FUEL CHANNEL COOLANT FLOW

Alexandru CATANA[1], Ilie PRISECARU[2], Daniel DUPLEAC[3], Danila NICOLAE[4]

In lucrare se prezintă analiza numerică a curgerii agentului de răcire în reactorii nucleari cu tuburi de presiune folosind abordarea CFD. Se analizează segmente semnificative din canalul combustibil cu sau fără considerarea unor elemente constructive caracteristice (apendaje). S-a utilizat o versiune academică a codului CFD, FLUENT. Curgerea agentului de răcire pe tubul de presiune este puternic turbulenta şi în general multi-fazică. Parametrii termohidraulici de-a lungul tubului de presiune sunt influenţaţi de apendaje. Condiţiile de curgere au fost simulate prin rezolvarea ecuaţiilor de conservare a masei şi impulsului. Modelul de turbulentă utilizat este modelul standard k-ε. S-au neglijat: generarea şi transferul de căldură. Condiţiile la margine au fost furnizate de analizele de subcanal. Sunt analizate fascicole combustibile standard si avansate pentru ACR1000. Lucrarea prezintă metodologia abordării CFD si probează fezabilitatea aplicării metodei la reactorii cu tuburi de presiune.

This paper presents the numerical investigation of coolant flow in pressure tube nuclear reactors using CFD approach. We analyze significant segments of fuel channel with appendages: bundle junctions, spacer planes, bare fuel bundles. The computer code used is an academic version of FLUENT CFD code. The complex flow in PT is highly turbulent, multi-phase. Turbulence intensity along pressure tube is influenced by appendages. The flow conditions have been simulated by solving mass and momentum conservation equations. Turbulence model is standard k-ε.

Heat generation/transfer is not simulated. Boundary conditions are provided by sub-channel analysis. Standard and advanced Fuel Bundles are analyzed.

Keywords: CFD, CANDU6, ACR1000, ADVANCED FUEL

1. Introduction

This paper presents the numerical investigation of coolant flow in pressure tube nuclear reactors fuel channels using CFD (Computational Fluid Dynamics) approach [1, 5]. Limited computer power available at Bucharest University POLITEHNICA enforced us to analyze only segments of fuel channel namely the significant ones: junctions with adjacent segments, spacer planes with adjacent

[1] Researcher, Nuclear Research Institute, RAAN, Pitesti, Romania
[2] Prof., Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania
[3] Prof., Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania
[4] Prof., Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania

segments, turbulence enhancement button planes with adjacent segments and regular segments of fuel bundles without appendages. The computer code used is an academic version of FLUENT CFD code at UPB [6, 7]. Fuel bundles contained in pressure tubes forms a complex flow domain. The flow is characterized by high turbulence and in some parts of fuel channel also by multi-phase flow. The turbulence intensity along pressure tube is strongly influenced by so called appendages. The flow in the fuel channel has been simulated by solving the equations for conservation of mass and momentum. For turbulence modeling the standard k-ε model is employed although other turbulence models can be used. In this paper we do not consider heat generation and heat transfer capabilities of CFD methods. Since we consider only some relatively short segments of a CANDU fuel channel we can assume, for this starting stage, that heat transfer is not very important for these short segments of fuel channel. The boundary conditions for CFD analysis are provided by system and sub-channel analysis. In this paper the discussion is focused on some flow parameters behavior at the bundle junction, spacer's plane configuration, etc. In this paper we present CFD simulation results for SEU43-ACR1000 fuel bundle flow characteristics and some comparisons with standard CANDU6 Fuel Bundles STD37 are made.

ACR-1000 is a light-water-cooled, heavy-water-moderated pressure-tube reactor evolved from CANDU6 from which it retains about 80% of plant features, equipment and specifications: fuel in pressure horizontal pressure tubes; simple fuel bundle design; on-power fuelling; separate, low-pressure, low-temperature, heavy-water moderator system providing an inherent emergency heat sink . It incorporates a specific set of innovative features and state of-the-art technologies to ensure its safety, operation, performance and economics: reactivity control and shutdown systems which includes rods for maintaining the reactor in a guaranteed shutdown state; light water (H2O) as cooler and low-enriched uranium fuel leading to a small negative reactivity coefficient; compact core; higher temperature and pressure increasing thermodynamic efficiency.

This paper focuses on some thermal-hydraulic characteristics for a hypothetical fuel bundle SEU43-ACR1000 using CFD approach. The analysis is based on available open source data as well as on our own evaluation about the detailed structure of this fuel bundle. In Table 1 some geometric data about standard (STD37) and SEU43-ACR1000 fuel bundles are presented.

Table 1

Parameter	CANDU6	ACR1000
Heat to steam generators (MW)	2064	3187
Gross/net electric output (MWe)	728/667	1165/1085
Number of channels	380	520
Core Diameter (m)	7.6	7.44
Lattice pitch (cm)	28.6	24.0
Moderator D2O volume (m^3)	265	235/250
Heat Transport System D_2O volume (m^3)	192	0
Total D_2O volume (m^3)	466	240
Fuel (wt% U^{235}/U)	0.71	up to 2.5
Number of rods per bundle	37	43
Total bundle weight (kg)	24.1	21.5
Reference discharge burn-up (MWd/Mg(U))	7500	20000
Outer header operating pressure (MPa)	9.9	11.1
Outer header operating temperature (^oC)	310	319
Inlet header operating pressure (MPa)	11.2	12.6
Inlet header operating temperature (^oC)	266	275
Bundles per channel	12	12
Coolant	D2O	H2O
Steam Temperature	260	276
PT Inlet temperature (^oC)	266	275
PT Inlet Pressure (MPa)	11.05	12.25
PT Outlet temperature (^oC)	312	321
PT Outlet Pressure (MPa)	10.285	11.45
Cp-Specific Heat at constant Pressure (J/kg-K)	4736.7	5073.4
Coolant Density at PT Inlet (kg/m^3)	862.66	767.20
Dinamic viscosity at PT Inlet (Pa.s or kg/m.s)	0.000111	0.00009735
Fuel Rods Diameter (cm)	1.308	1.150 2.110 Central
Fuel Bundle Ring Structure	1, 6, 12, 18	1, 7, 14, 21
Fluid flow area inside PT (m^2)	0.0035068	0.003766
Wetted Perimeter (m)	1.8468	1.9101
Mass Flow (kg/sec)	23.9	23.9
Volume Flow (m3/sec)	0.027705	0.031095
Flow velocity	7.9	8.256
Reynolds number	466350.7	514134.4
Turbulence Intensity (%)	3.13	3.092

2. CFD approach for thermal-hydraulic analysis ACR1000 pressure tube nuclear reactors

As a pressure tube ACR1000 is particularly suitable for CFD thermal-hydraulic analysis. The reactor core is made from pressure tubes, a repetitive component with a well defined geometry. The inherently small scale comparing with a pressure vessel makes this component more suitable for detailed and accurate analysis, like CFD analysis. In this paper we focused our attention on thermal-hydraulics analysis of flow inside ACR10000 pressure tube filled with twelve fuel bundles called SEU43-ACR1000, Fig. 1.

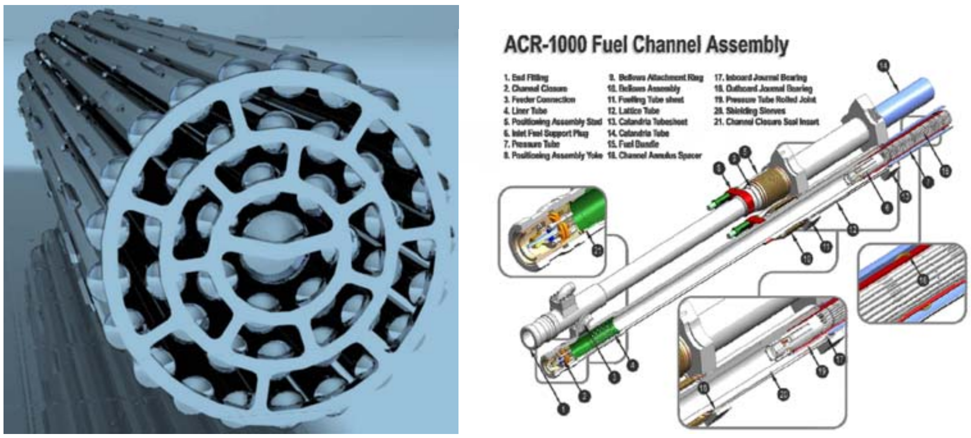

Fig. 1. SEU43-ACR1000 Fuel bundle and Fuel Channel Assembly

As in case of standard fuel bundles there are some characteristic segments of ACR1000 pressure tube filled with fuel bundles which exhibit a distinct flow field behavior. The flow domain has been modeled using GAMBIT [7], pre-processor. Fig. 2 shows the geometry of *two fuel bundle junction* as well as the geometry of a *spacer and mid bearing pads plane*. In this paper only the simulation of two aligned fuel bundles are presented.

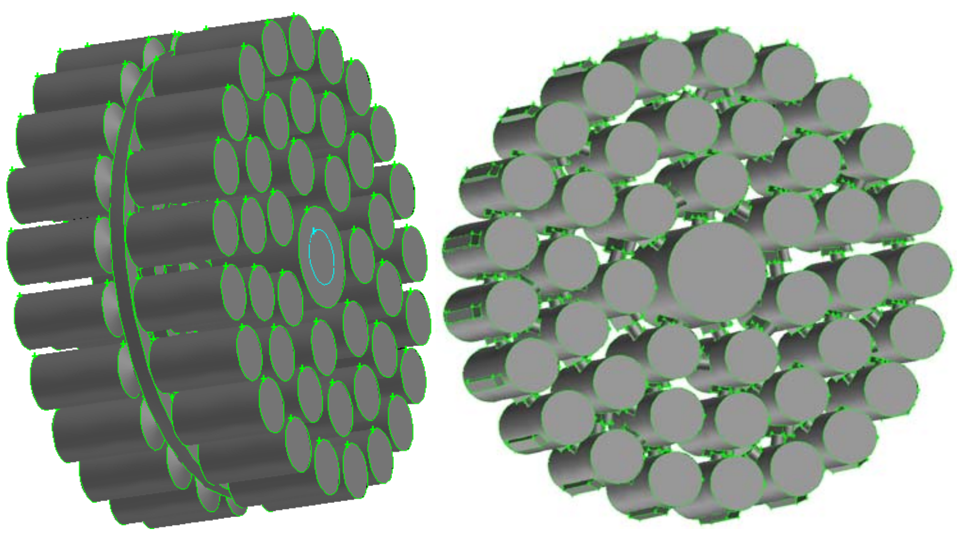

Fig. 2. The flow geometry of bundle junction and spacer plane

This configurations of fuel channel is repeated (11 times junction and 12 time spacers) along the fuel channel. Turbulence *enhancement buttons plane* and segments of *regular pressure tube segments* were analyzed, Fig. 3.

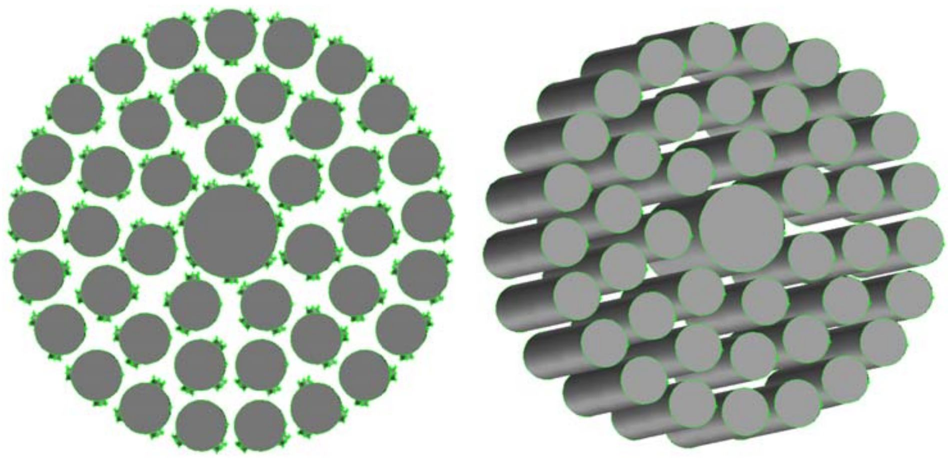

Fig. 3. The flow geometry of enhancement buttons regular bundle segment

For these modeled segments we succeed to accomplish a complete CFD cycle which means the definition of problem, geometric description of fluid flow domain, grid mesh generation, setting up boundary conditions, use of solver and post-processing the results. Operating conditions changes along fuel channel and these operating conditions are provided by using a sub-channel analysis code, a modified COBRA.

3. CFD Analysis results for ACR1000 with SEU43-ACR1000

ACR1000 is a III+ generation pressure tube reactor struggling in a competitive market. In order to succeed, the design must be near perfect. A good and accurate understanding of al phenomena both nuclear, thermal hydraulic and structural must be accomplished. A new generation of analysis tools are emerging based on basic mathematical model, advanced numerical solvers and powerful computer systems. In thermal-hydraulics a bunch of complex computer codes, namely CFD codes, both in-house and commercial are used for flow field numerical simulation. Almost all have in common some characteristics they are: complex codes, user friendly, best-estimates, need a lot of computer and human resource. Usually they involve three main stages: **pre-processing ~ solving** ~ **post-processing**. The complexity of flow field is far beyond the possibility of numerical or textual description. As a consequence graphical and some times tabular presentation is used for simulation result presentation. The climax of graphical presentation is animation in which transient results can be presented.

Some of graphical possibilities are illustrated in this paper. In *Fig. 4, Velocity Magnitude* is presented in particular geometries: fuel bundle junction and spacer plane for a SEU43-ACR1000 fuel bundle.

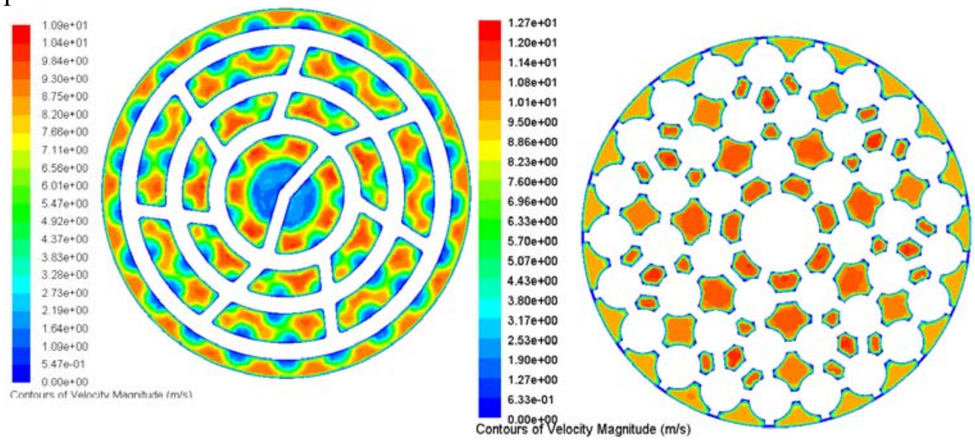

Fig. 4. Velocity Magnitude in SEU43-ACR1000: Left-bundle junction ; Right-spacer plane

Along the simulated segments the variation of parameters are also presented in graphical manner. *In Fig. 5 Turbulence kinetic energy (k) and Velocity Magnitude* contours are presented for SEU43-ACR1000. We can see that the speed is increased as flow area is narrowing. More interesting is the information which came from Turbulence kinetic (k) in Fig.5. We can see that some conclusions can be drawn from observing the value of turbulence in some specific points. The usefulness, size, shape and axial position of turbulence enhancement buttons is a matter of dispute. From picture we can say that these buttons must appear in the "blue" spots while in the red ones, don't.

From pictures depicted in Fig. 6-8 we see that some thermal hydraulic parameters: absolute pressure, velocity magnitude, turbulence kinetic energy, etc., manifest a strong variation in flow domain where appendages are present.

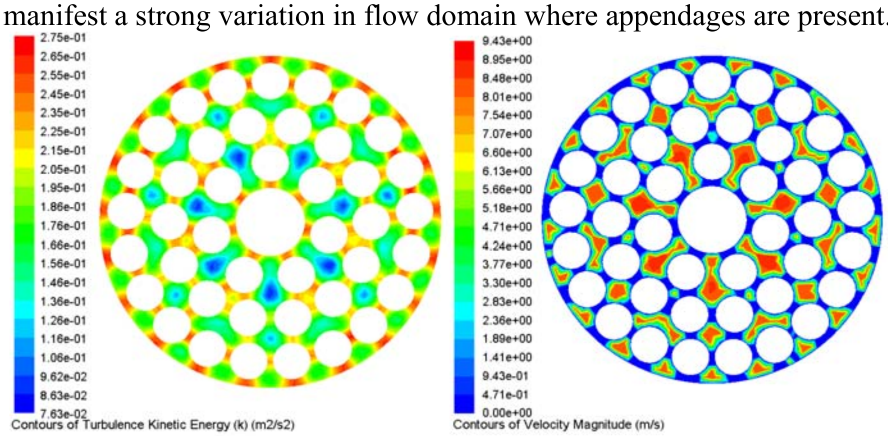

Fig. 5. Turb. kinetic energy (k) and Vel. Magnitude in regular segments in SEU43-ACR1000.

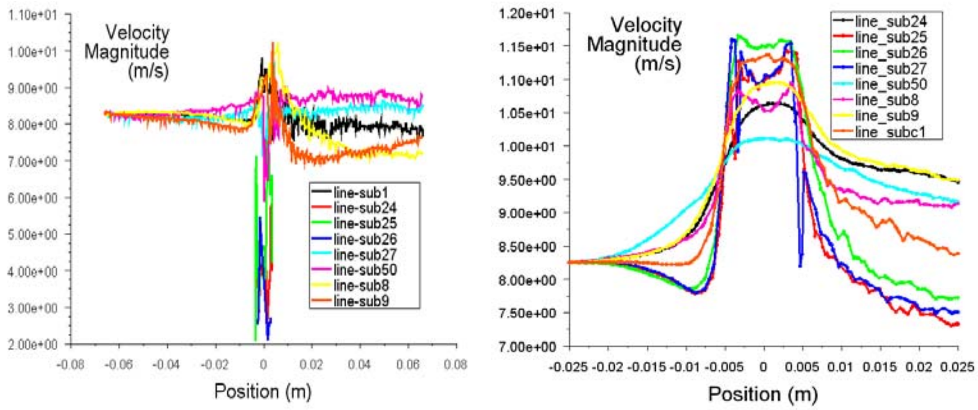

Fig. 6. Velocity Magnitude profiles in SEU43-ACR100. Left-bundle junction; Right-spacer plane

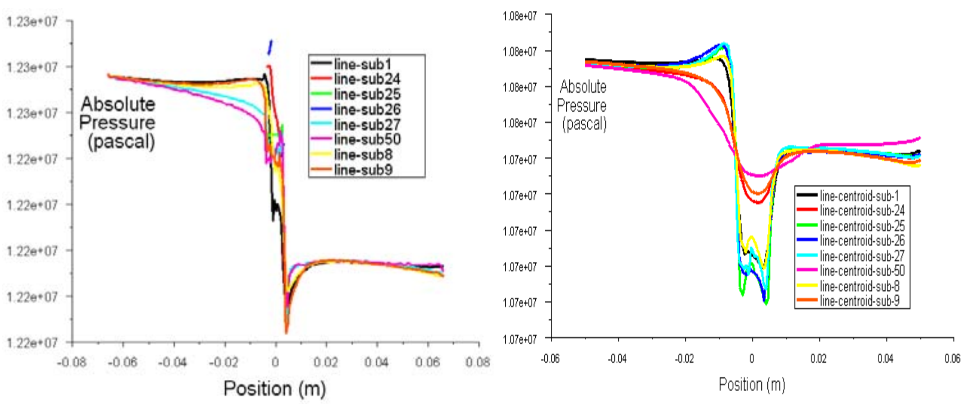

Fig. 7. Absolute Pressure profiles in SEU43-ACR1000: Left-bundle junction; Right-spacer plane

Although much more field variables can be displayed, only few of them we do present in this paper: pressure and pressure drop along simulated portions of pressure tube, velocity profiles along the same segments. Turbulence kinetic energy (k) as a measure of flow turbulence is also presented. Besides graphical information, some numerical results are even more important. Pressure variation along the pressure tube is one of most important thermal-hydraulic parameter. Area weighted averaged absolute pressure drop on appendages are due to geometrical sudden variation of flow area. In this particular complex geometries, as we can see in Fig. 2, we have highly turbulent flow. Adding all pressure drops we do not get exactly the pressure drop of 630000 or 690000 Pa. The difference is in inlet channel contraction (~35000 Pa) and in expansion on at channel exit.

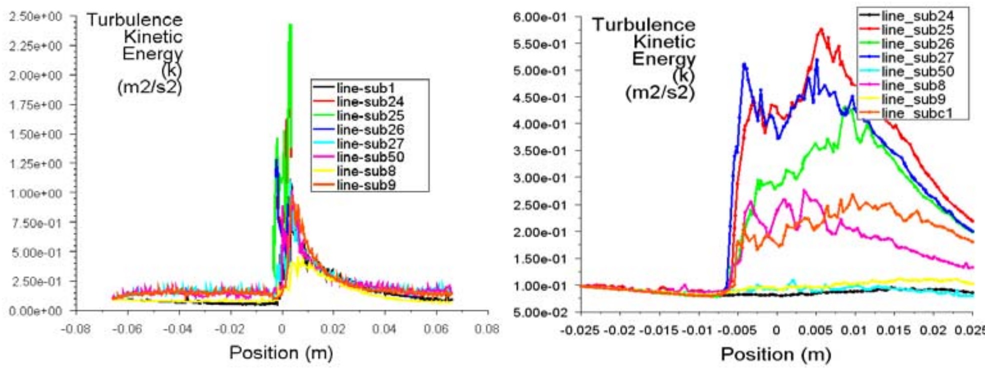

Fig. 8. Turbulence Kinetic Energy profiles in SEU43: Left-bundle junction; Right-spacer plane

Table 2

Bundle Type	Pressure drop [Pa]: SPACERS	Pressure drop [Pa]: ALIGNED JUNCTION	Pressure drop [Pa]: 1 M LONG FUEL RODS
STD37	8584 on one plane 103008 on all (12)	16458 on one junction 181038 on all (11)	40015 on 1 M length 240090 on 6 m
SEU43-ACR1000	10064 on one plane 120768 on all (12)	18302 on one junction 201322 on all (11)	39400 on 1 M length 236400 on 6 m

4. Conclusions

The CFD analysis for coolant flow on an ACR1000 pressure tube with specific SEU43-ACR1000 fuel bundles is first of a kind in Romania. A new pressure tube nuclear reactor, ACR1000, with a new fuel bundle type must be thoroughly assessed. CFD analysis provides an accurate tool for thermal-hydraulic analysis. This paper proves the feasibility of this new approach. Even if only a steady-state without heat transfer analysis was actually performed, a CFD methodology was setup enabling the accomplishment of new and more significant analysis in near future on the exiting basis. On this basis heat transfer, two-phase flow and transient analysis for normal and accident conditions could be performed.

Generation IV, CANDU-SCWR pressure tube nuclear reactor can also be analysed in similar manner using CFD. Thermo-physical properties beyond critical point of light and heavy water must be available. Not only coolant flow inside pressure tube but virtually any flow field in nuclear reactors can be analysed with CFD methods as long as the geometry, operating conditions and material properties are known.

R E F E R E N C E S

[1]. *Alexandru Catana, Ilie Prisecaru, Daniel Dupleac, Nicolae Danila,* CFD Thermal-Hydraulic analysis of a CANDU fuel channel, Proceedings of Nuclear 2009, May 27-29, Pitesi, Romania. ISSN 2066-2955

[2]. *Alexandru Catana,* Utilizarea metodelor CFD in analize termohidraulice de securitate pentru reactorii nucleari actuali si avansati (The use of CFD methods in thermal-hydraulic safety analysis for present and advanced nuclear reactors), RI-8054, SCN Pitesti, 30.03.2008.
[3]. *Alexandru Catana, Nicolae Danila, Ilie Prisecaru, Daniel Dupleac*, Thermal-hydraulics analysis for advanced fuel to be used in CANDU 600 nuclear reactors, PHYSOR'08, Proceedings of International Conference on the Physiscs of Reactors, "Nuclear Power A Sustainable Resource", 14-19 Sep.2008, Interlaken, Switzerland.
[4]. *O. Kirschner, A. Ruprecht,* Vortex Rope Measurement in a Simplified Draft Tube, in Proceedings of the 2nd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Scientific Bulletin of Politehnica University of Timisoara, Transaction on Mechanics, **vol. 52(66)**, no. 6., 2007, pp. 173-184.
[5]. *Tom Keheley,* AREVA use of CFD in Fuel Assembly Design and Licensing, Thermal Hydraulics, Salt Lake City, UT February,23/24,2006.
[6]. *Xie Zhonsheng, P.G. Boczar*,"CANDU Fuel – Cycle Vision", AECL.
[7]. Fluent Inc., (2004). Gambit 2.2 user's guide. Fluent Inc.
[8]. Fluent Inc., (2005). Fluent 6.2 user's guide. Fluent Inc.
[9]. *Alexandru Catana, Ilie Prisecaru, Daniel Dupleac, Nicolae Danila,* CFD Thermal-Hydraulic analysis of a CANDU fuel channel with SEU43 fuel bundle type, International Symposium on Nuclear Energy, SIEN'09, *"The Nuclear Power – A New Challenge"* Ramada Parc Hotel, Bucharest, ROMANIA, 12-16 October, 2009