ergy centre// Science and Technology 2014, 4(2): 9-16 DOI: 0.5923/j.scit.20140402.01

4. IAEA, 1994. Convention on Nuclear Safety,

Fidel Castro Parimala Rangan (India)

Tomsk Polytechnic University, Tomsk

Scientific adviser: Dr. Alexander G. Korotkikh,

CALCULATION AND COMPARISON OF HEAT TRANSFER COEFFICIENT & HEAT FLUX BY VARYING THE PARAMETERS FOR SCWR THROUGH CFD SIMULATIONS

Abstract

In the present study, CFD simulation was conducted for 2×2 rod bare bundle using water at supercritical pressures. Main objective of the simulation was to compare working of different turbulence models. K-epsilon, K-omega and Spalart-Allmaras turbulence models were chosen for our study. K-epsilon and K-omega turbulence models are two equation models and are widely used for industrial research. Whereas Spalart-Allmaras is one equation model which is least computationally expensive of all the models. All three turbulence models come under the Reynolds Average Navier Strokes model (RANS). CFD results were found to be sensitive with the appropriate turbulence model and this variation is documented through various plots.

Introduction

CFD simulation was performed to replicate the results from the experiment of heat transfer to supercritical water in 2×2 rod bundle conducted at Shanghai Jiao Tong University. This report presents the results to assess capability of the commercial CFD software Ansys fluent in simulating the convective heat transfer of water at supercritical pressures in nuclear fuel rod. Sensitivity studies were performed for three turbulence models, K-epsilon, K-omega and Spalart-Allmaras. Results from all the turbulence models will be closely monitored to compare them with the experimental data. Different mesh configuration will be decided for each turbulence model. K-omega turbulence model will require prism layers closer to the wall in order to fully resolve the fluid flow. Experiments used for the assessment of the current simulations are presented in next section.

A series of experiments were performed at Shanghai Jiao Tong University [1]. It consists of the main test loop, cooling water loop and I&C system Fig 1. shows water temperature in the two channels with different fluid inlet

temperatures. The working pressure is 25 MPa. The mass flux is 800 kg/m2s and the heat flux is 600 kW/m2. In our study we will compare our simulation results with the data in fig 3. and then will check which turbulence model gives results most accurately.
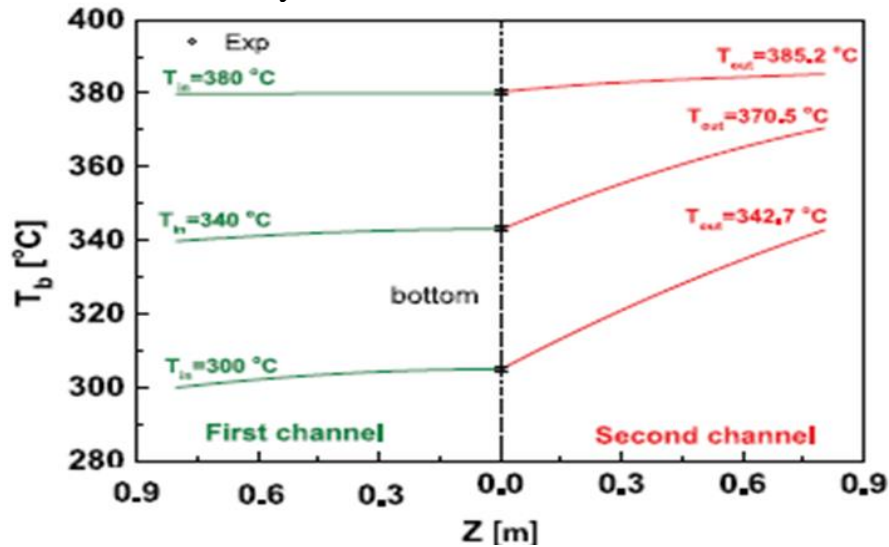

Figure 1. Water temperature distribution along the axial length in the two channels.

Background Research for turbulence models

- K-epsilon Turbulence model

Widely used despite the known limitations of the model. Performs poorly for complex flows involving severe pressure gradient, separation, strong streamline curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies

Using Wall Functions

Wall functions utilize the predictable dimensionless boundary layer profile to allow conditions at the wall (e.g. shear stress) to be determined by when the centroid of the wall adjacent mesh cell is located in the log-layer.To locate the first cell in the log-layer, it should typically have a y+ value such that $30 < y+ < 300$.

- K-omega Turbulence model

Superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows compared to models from the k-e family. Suitable for complex boundary layer flows under adverse pressure gradient. Separation can be predicted to be excessive and early

Resolving the Viscous Sublayer

First grid cell needs to be at about y+ ≈ 1 and a prism layer mesh with growth rate no higher than ≈ 1.2 should be used.

- **Spalart-AllmarasTurbulence model**

Spalart-Allmaras is one equation turbulence model which is economical for large meshes. Good for mildly complex (quasi-2D) external/internal flows

and boundary layer flows under pressure gradient (e.g. airfoils, wings, airplane fuselages, missiles, ship hulls). Performs poorly for 3D flows, free shear flows, flows with strong separation

CAD geometry and Mesh details

Geometry is created using Ansys design modular and meshing is performed in Ansys Mesher. Due to double symmetry, the geometry is reduced to only quarter portion, taking advantage of symmetry boundary condition thus reducing computational time.The simulation model can be classified as parallel flow heat exchanger device where energy is exchanged by providing heat flux. Water at 25MPa flows through the outer channel to the mixing chamber. From mixing chamber, it travels to the inner channel in counter parallel direction.
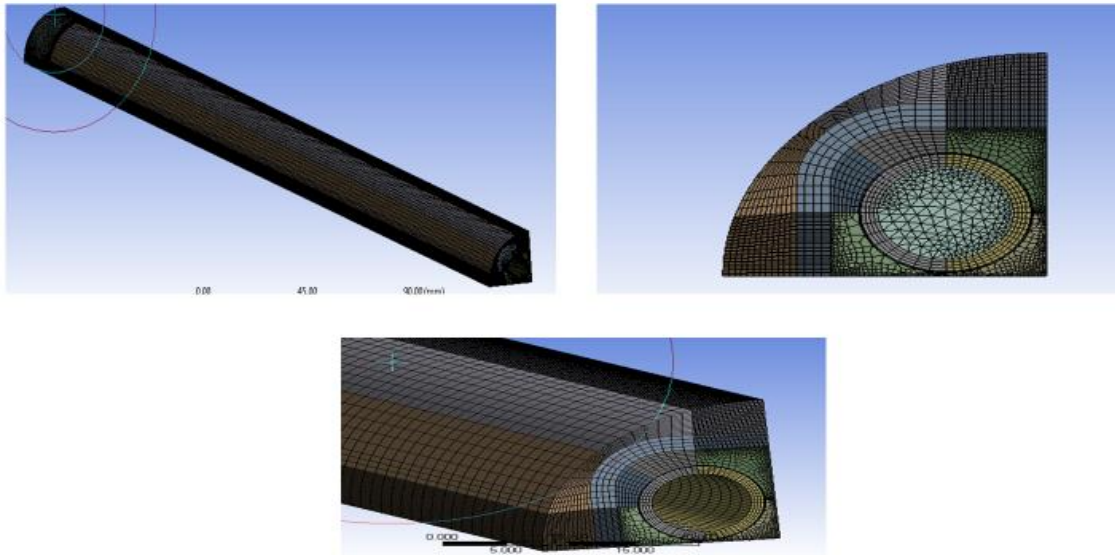
Mesh details

Figure 2. Mesh Type 1

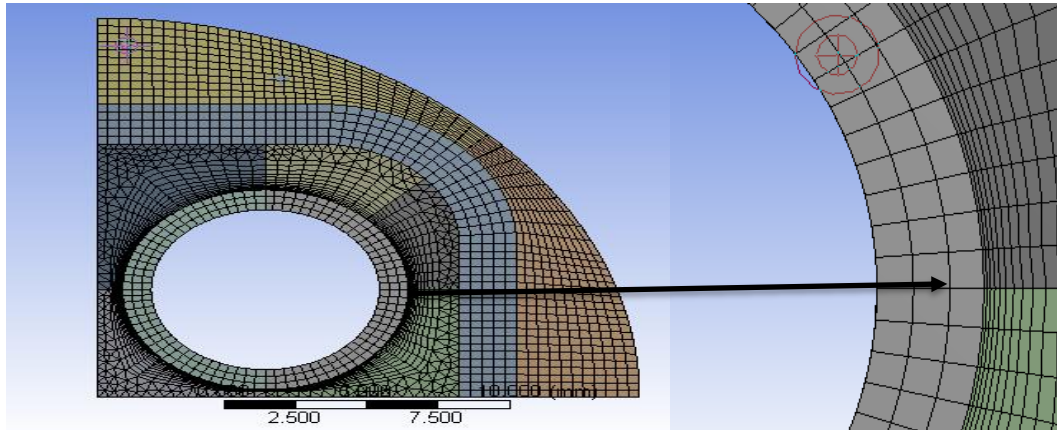| Mesh Type 1	Hexa+Tetra(Hybrid)	Mesh Type 2	Hexa+Tetra (Hybrid)
Mesh count	0.9 million	Mesh count	1.6 million
No. of prism layers	4	No. of prism layers	12
Prism Layer growth rate	1.2	Prism layer growth rate	1.2

Figure 3. Mesh Type 2

Two meshed geometries were generated with different configurations. Mesh type 1 was used for K-epsilon and Spalart-Allmaras Turbulence model where as Mesh type 2 was used for K-omega turbulence model. In Mesh type 2 the no of inflation layers were increased to 12 with the value of y+ =1. Mesh count was nearly increased to double from 0.9 million to 1.6 million. Hybrid meshing is applied for the model and prism layer is activated on account of turbulence flow. Prism layer is created on the surface of the channels to capture the physics of boundary layer creation. As the gradient are changing rapidly on the boundary layer therefore small element size is recommended inside the layer. First layer thickness is calculated for the value of Y+=11 as recommended for K-epsilon model for internal flow and Y+=1 is used for K-omega model. Growth rate for prism layer is taken as 1.2, so that the mesh element size is increased at a constant rate otherwise if the element size will increase rapidly then the solution may become diverged.

Simulation setup and Model description

In this present study to analyze the three-dimensional flow, Ansys fluent is used as CFD solver. The discretization of viscous and thermal diffusion terms has been achieved through the central differencing scheme. Second order upwind scheme is used to discretize the advection terms. SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm has been adapted to achieve the coupling of pressure and velocity fields, which implicitly takes care of the divergence-free nature of the incompressible fluid flow. The convergence criteria have been set to 10-6 for all the residuals (energy, momentum, continuity etc.). Computational fluid dynamics (CFD) is used for investigating the problem. In the present simulation all the governing equations continuity, momentum and energy are solved by finite volume method using academic version of Ansys 18.1. There are several schemes through which we can guide the CFD solver to set of equations used.Also, all the residuals can be given convergence criteria to get a converged solution. All the details

regarding the assumptions, governing equation, schemes, setup and solution methods are given below.

Assumptions

The assumptions taken in this study are as follows:

1. No slip boundary condition is assigned at the pipe surface.

2. Thermal conductivity of the pipe and water assumed to be constant

3. Homogeneous and isotropic material is presumed for pipe wall.

4. Heat loss from radiation are considered to negligible and hence neglected.

5. A three dimensional fully developed incompressible, turbulent and steady flow is considered.

Governing equations

Continuity equations:

$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$

$\frac{\partial \rho}{\partial x} + \nabla.(\rho V) = 0$

Momentum equations:

$\frac{\partial(\rho V)}{\partial x} + \nabla.(\rho V) = -\nabla p + \mu \nabla 2V + \rho g + S$

Energy equations:

$\frac{\partial H}{\partial x} + \nabla.(\rho V H) = k \nabla 2T + S$

Setup details

Description	Type
Solver	Pressure based
Energy	On
Solidifications and melting	Off
Gravity (y axis)	On (-9.81m/sec)
Time	Steady
Turbulence Model	K- epsilon, K-omega & Spalart-Allmaras
Velocity formulation	Absolute

For this simulation we will take solver as pressure based. Energy will be on because heat transfer is involved in this simulation. Gravity is taken as -9.81m/sec in y direction. The flow is taken as steady state flow.

Schemes

Different schemes can be used for solving the governing equations. First order schemes can be converged easily but are accurate only in first order. For second order upwind accuracy is quite high but the convergence time is

very large. Type of scheme used to be used can be decided on the basis of computational power available. If the system available had multi-core processor then high order schemes can be used. Also choosing schemes can be done on the basis of accuracy required. If high accuracy is needed then go for second order scheme otherwise prefer first order scheme. Solutions methods and scheme used for our investigation are provided in table below.

Schemes details

Solution method	Scheme
Pressure	SIMPLE
Pressure- velocity compounding	SIMPLE
Momentum equations	2ND order Upwind
Energy and continuity equations	2ND order Upwind
Gradient	Least square cell based
Turbulent and kinetic energy equations	2ND order Upwind

Convergence criteria

Convergence criteria are the tolerance limit between the current value and the previous value at a node. With each iteration the CFD solver checks for the convergence criteria. If tolerance limit is more than the convergence criteria then the solver will go for one more iteration until the solution gets converged.

Residuals	Convergence criteria
Continuity	1e-06
X- velocity	1e-06
Y-velocity	1e-06
Z-velocity	1e-06
Energy	1e-08
K	0.001
Epsilon	0.001

Results and Discussion

Results from the CFD simulations for different turbulence models were plotted with the experimental data. The results were obtained for three inlet temperature of 300 ℃, 340 ℃ and 380℃ for 25 MPa pressure.

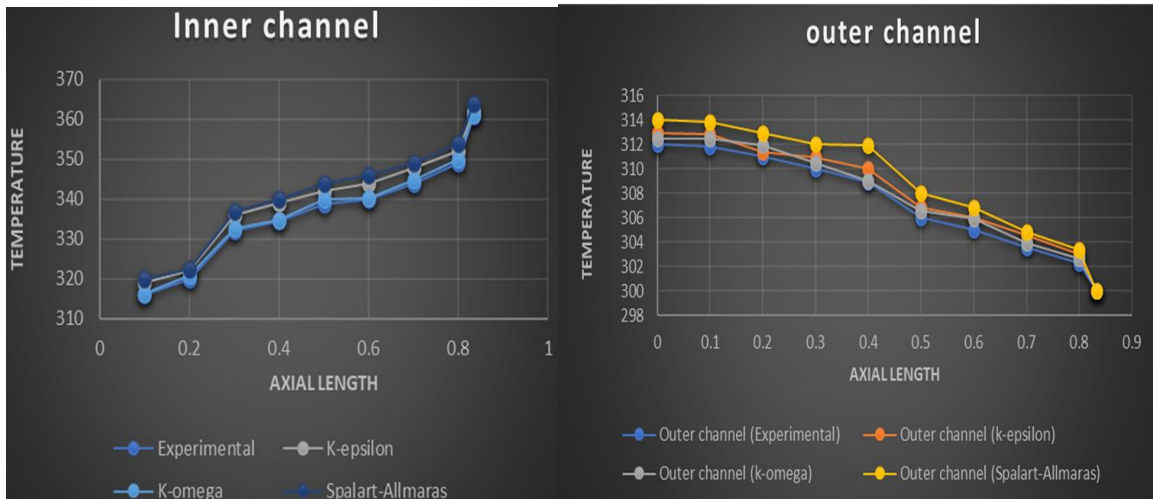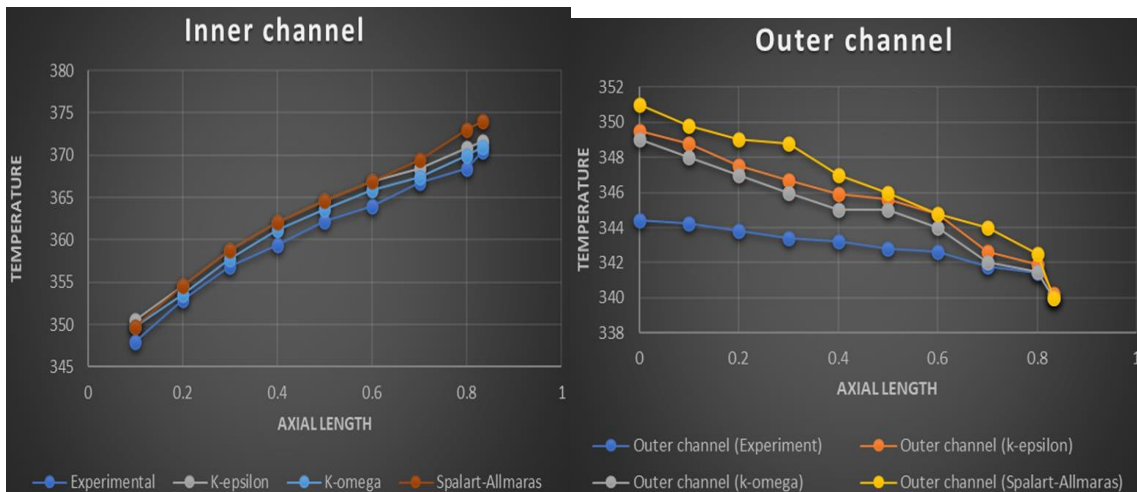*Figure 4. Inlet Temperature 300℃*
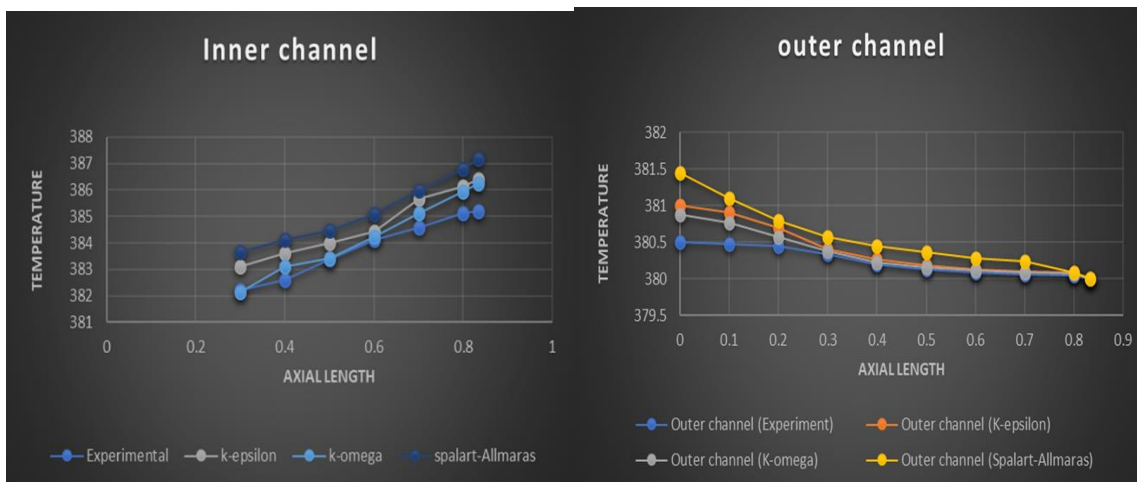

Figure 5. Inlet Temperature 340℃

Figure 6. Inlet Temperature 380℃

Conclusion

All the turbulence models give results in acceptable range closer to the experimental data. K-omega turbulence model predicts result most accurately

387

among all the turbulence models whereas Spalart-Allmaras Turbulence model deviates the most from experimental data. Due to ability of K-omega turbulence model to capture near wall physics, it is most suitable for conjugate heat transfer problems. K-epsilon model gives better results for mixing of fluid at the mid-section of pipe away from the wall and is computationally less expensive as compared to K-omega turbulence model. To summaries it all, when we require accurate results, we can go for either K-omega or K-epsilon turbulence model whereas when we only need a crude estimation then Spalart-Allmaras Turbulence model is more suitable as it is least expensive of all the turbulence models available.

REFERENCES

1. Experimental studies on heat transfer to supercritical water in 2 × 2 rod bundle with two channels /H.Y. Gua, Z.X. Hua, D. Liua, Y. Xiaoa, X. Cheng //Nuclear Engineering and DesignVolume 291, September 2015, Pages 212-223

2. forced-convection heat transfer to water at near-critical temperatures and supercritical pressures / Bishop, A A, Sandberg, R O, and Tong, L S.// United States: N. p., 1964. Web.

3. A simplified method for heat transfer prediction of supercritical fluids in circular tubes / ChengY.H.YangS.F.Huang // Annals of Nuclear EnergyVolume 36, Issue 8, August 2009, Pages 1120-1128

4. Experimental study on heat transfer of supercritical Freon flowing upward in a circular tube / HanyangGuaXuChengabZhenqinXionga //Nuclear Engineering and DesignVolume 280, December 2014, Pages 305-315

5. Experimental investigation of heat transfer from a 2 × 2 rod bundle to supercritical pressure water / HanWangaQinchengBiaLinchuanWangaHaicaiLvaLaurence K.H.Leungb //Nuclear Engineering and DesignVolume 275, August 2014, Pages 205-218