Numerical Analysis of Heat Transfer in sub channels of Nuclear Reactors using supercritical Freon R-12 as Coolant

Chandradeep Barman[1], **Deepak Sharma**[2], **K. M. Pandey**[3]
[1]*Research scholar, Department of Mechanical Engineering*
National Institute of Technology, Assam, Silchar, 788010, India
[2]*Research scholar, Department of Mechanical Engineering*
National Institute of Technology, Assam, Silchar, 788010, India
[3]*Department of Mechanical Engineering*
National Institute of Technology, Assam, Silchar, 788010, India

---

Abstract- This paper analyses the problem through ANSYS CFD technique to obtain steady state heat transfer in nuclear fuel rod assembly and the local heat transfer coefficients of supercritical Freon R-12 in sub channels of a CANDU reactor under cooling conditions. The objective of the present research work is to get reference dataset on heat transfer in Freon (R-12) and to improve fundamental knowledge of the heat-transfer processes and handling of supercritical fluids. The experimental data has been taken from" G. Richards et el, 2012" and this consists of tests performed in upward flow of supercritical Freon (R-12) inside 9.5mm diameter 7 fuel rod assembly with a 1000 mm heated length. Supercritical Freon (R-12) heat transfer data were obtained at reactor equivalent conditions - inlet pressureof 4.65 MPa and the inlet temperature of 119° C, mass fluxat 517 kg/m^2s, local wall heat flux at 33.5kW/m^2 . The effects of pressure, mass flux and heat flux on heat transfer are investigated of supercritical Freon (R-12). The simulated results show that the difference in predicted temperature and heat transfer coefficient is only ± 6% of the experimental results.

Keywords: CFD, Fuel rod assembly, Sub channels, Supercritical Freon R-12, Mass flux, Heat flux.

1. INTRODUCTION

In the present world, there is a great demand for environment friendly power generation for which nuclear power plant is a suitable alternative. Although it has a controversial reputation but its importance cannot be ignored. Nuclear power plants are efficient and reliable. Nuclear power is economically feasible and is responsible for meeting 20% of world's energy demand .The extraordinary high energy density of nuclear fuel as compared to fossil fuels is a very worthwhile physical characteristic [**2**].

In the experimental work done so far on the heat transfer of fluids at supercritical conditions, majority of the flow passages arevertical, some are horizontal and just a few in other flow geometries[**3**]. A wide range of fluids have been used as coolants like water, carbon dioxide, Freon, helium, hydrogen etc. Although the data available has been limited mostly to water and carbon dioxide.While water is currently being used as the primary fluid for reactor coolant channels under supercritical conditions, the supercritical CO2 cooled reactor is expected to gain more significance in the future due to its lesser operational cost and lower critical parameters. In this paper we would be analyzing the use of Freon R 12 as a coolant in the sub channels of a nuclear reactor.

The aim of this study is to better understand the behavior of the 3-D steady state heat transfer in 7 fuel rod assembly hexagonal outer section and also understand the behavior of the coolant Freon R-12 under supercritical condition by means of CFD.Numerical simulation is an important tool to compute the hydraulic and thermal flow distributions in a complex geometry. The CFD results obtained need to be accompanied with the experimental data in order to validate the different models in obtaining the solution. Due to the complexities involved in the full fuel bundles and high computational demands, many researchers have conducted simulations with one or few sub channels or a section of the fuel bundle rather than the full fuel bundle[**4,5,6**]

The flow of the coolant inside the fuel bundles is highly turbulent and causes vibrations inside a channel. This present paper is concerned with CFD analysis of heat transfer to Freon R-12 surrounding a 7 bundle fuel rod bundle. It is based on the experiments carried out by G. Richards et el.[**1**]. These were conducted at a critical pressure of 4.167 MPa and critical temperature of 111.97 ° C. Freon-12 was used as working fluid as it has low latent heat, low critical pressure, well known properties and well researched fluid-to-fluid modeling for water and Freon- R-12[**7**]. Freon R-12 had widely been used in industry as a refrigerant for air-conditioning systems. Therefore, its thermo physical properties are well known across wide range of conditions. The properties of coolant and heat transfer were studied by incorporation of transport models into commercial CFD codes like CFD Fluent.

2. MATHEMATICAL AND PHYSICAL MODELS

The physical model used here is of a Supercritical water cooled nuclear reactor (SCWR). The model is of a 7 rod fuel assembly in a hexagonal flow channel placed vertically oriented.The CFD simulation has been performed on a 60° sector of the fuel bundle. Fig.1 shows the cross section and different sub-channels of the fuel rod bundle. The model has been prepared in Pro.E 2.0 software. Temperature distribution and flow pattern were simulated using ANSYS Fluent. ICEM CFD was used for generating mesh. The standard *k–ε* turbulence model was applied with enhanced wall functions. The second order algorithm was used for pressure discretization. The SIMPLE algorithm was applied for pressure–velocity coupling. The properties of stainless steel were used as provided in the FLUENT properties database.The dimensions of the fuel rod assembly has been provided in table.1.

Table 1: Parameters for the upward flow fuel rod assembly

Parameter	Value
Inlet pressure	4.65 Mpa
Inlet temperature	119°C
Heat flux	33.5 kW/m^2
Mass flux	517 kg/m^2s

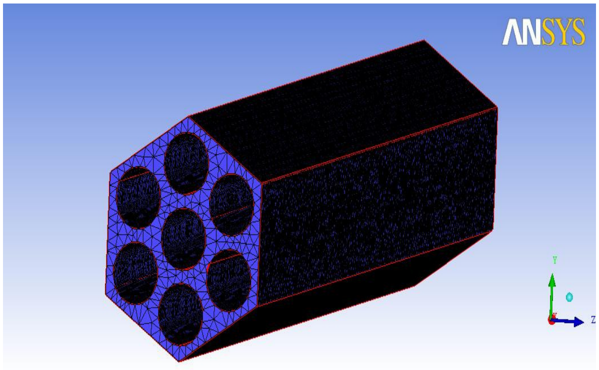


Fig. 1 Meshing of the 7 rod fuel assembly

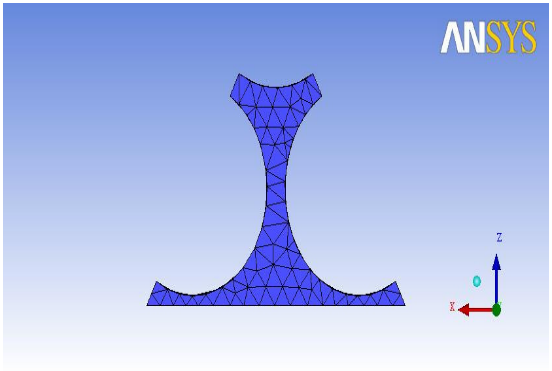


Fig. 2 Meshing of 60° sector of the fuel bundle

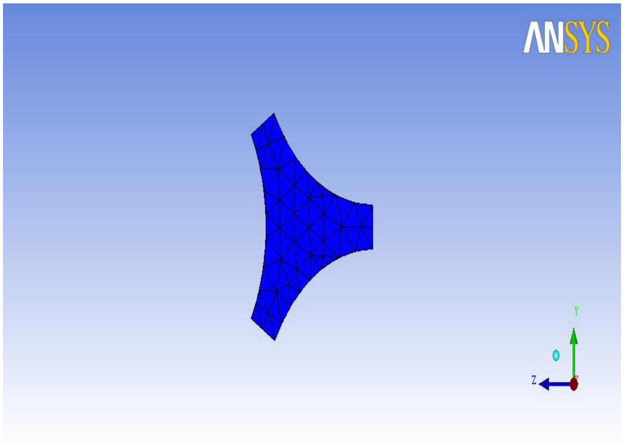


Fig.3 Meshing of the triangular sub channel

Table 2:Geometrical data fuel rod assembly[all dimensions are given in mm and mm^2]

Parameter	Notation	Geometry [360° sector]	Geometry [60° sector]
Hexagon side length	S	18.3	18.3
Fuel rod diameter	D	9.5	9.5
Heated length	L	1000	1000
Total perimeter	P_T	318.715	53.11
Free flow area	A	373.89	62.315
Hydraulic diameter	D_e	4.69	4.69

The hydraulic diameter of the present geometry is defined as

$$D_e = \frac{4A}{P_T} \qquad [1]$$

where A is the free flow area and $P_T$is the totalwetted perimeter (Table 2)
The free flow area for a 60°sector is calculated as:

$$A = \beta\ (Area_{Hexa} - 7\frac{\Omega}{4}D^{2}{}_{R}\ ) \qquad [2]$$

and the total wetted perimeter is given by

$$P_T = 6\,L + 7\Omega D_R \qquad [3]$$

3. GOVERNING EQUATIONS

The physical aspects of any fluid flow and heat transfer are governed by three fundamental principles:

i)Continuity equation
ii) Momentum equation
iii) Energy equation.

These are given as follows:

Continuity Equation $$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} + \frac{\partial W}{\partial Z} = 0 \qquad [4]$$

Momentum Equation

x Momentum: $$\rho\left[u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right] = F_x - \frac{\partial p}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right] \qquad [5]$$

y Momentum: $$\rho\left[u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right] = F_y - \frac{\partial p}{\partial y} + \mu\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right] \qquad [6]$$

z Momentum: $$\rho\left[u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right]=F_z-\frac{\partial p}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right] \qquad [7]$$

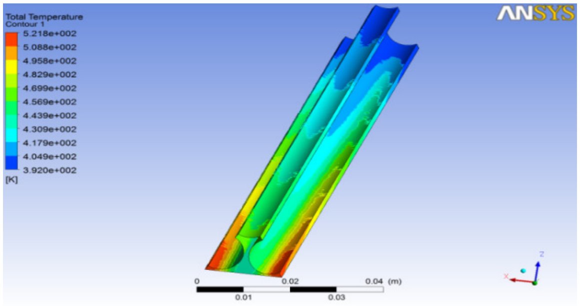
Energy Equation $$\rho c_p\left[u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right]=\lambda\left[\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right]+\mu\phi \qquad [8]$$

where, the viscosity-energy-dissipation function ϕ is defined as:

$$\phi=2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]+\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial z}{\partial y}\right)^2 \qquad [9]$$

4. RESULTS AND DISCUSSIONS

The contours of wall temperature, Wallheat transfer coefficient for the 60° section and the triangular sub channel are shown. Along with it, the plots of wall temperature and Heat transfer coefficients are given below.We have also analyzed different fluid flow models for the present study.



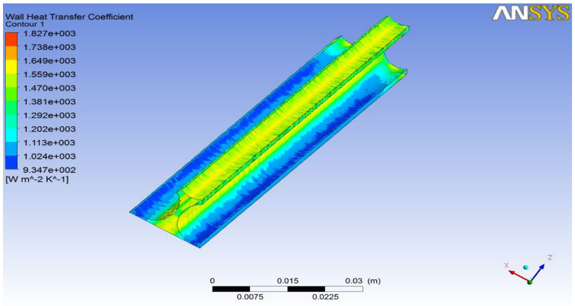
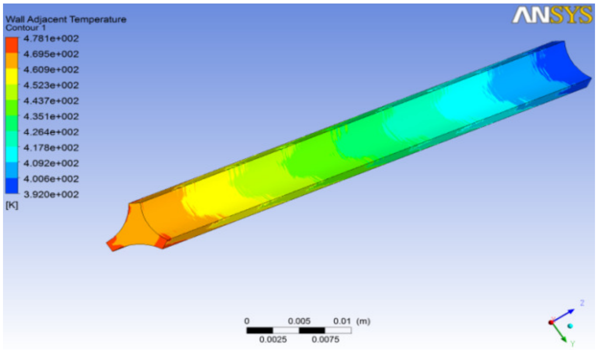


Fig.4 Wall Temperature contour

Fig. 5 Wall Heat transfer coefficient contour



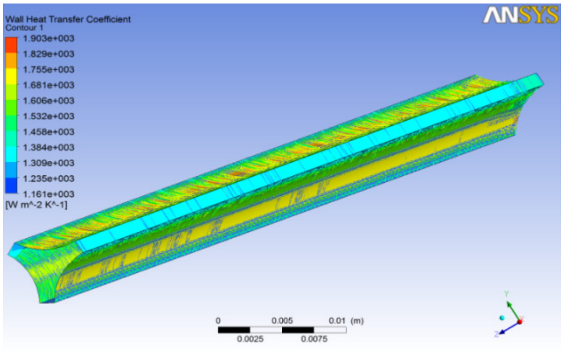


Fig. 6Wall Temperature contour of the triangular of the triangular sub channel

Fig. 7 Wall Heat transfer coefficient contour of the triangular sub channel

The accuracy of the computational results is not confirmed as long as it is not validated with the experimental data. The present simulation is validated with the experimental data by Richards et el.[**1**].

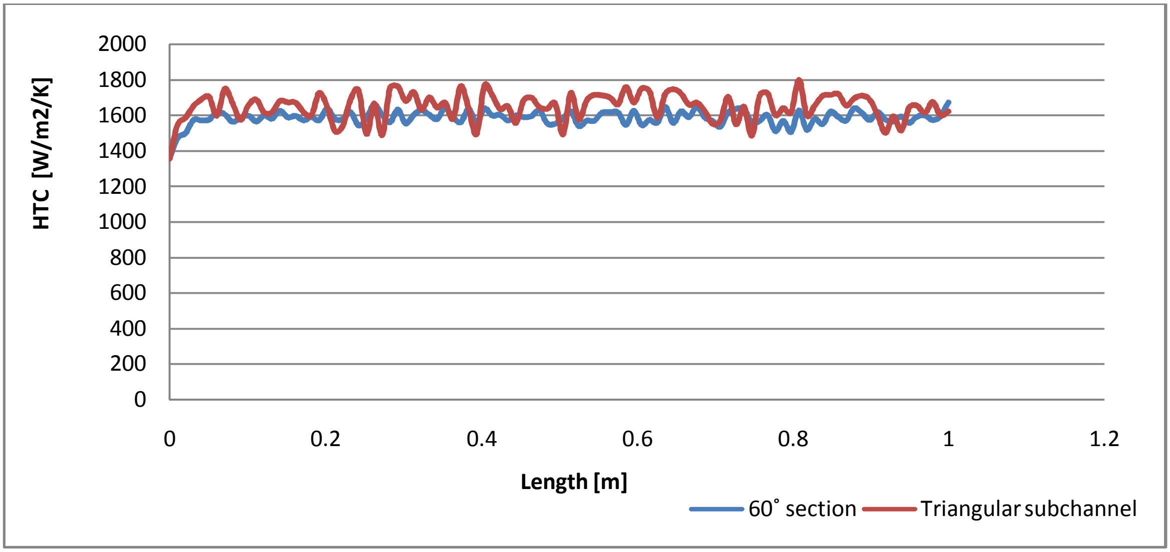


Fig. 8 Heat transfer variation with the 60˚section and the triangular sub channel

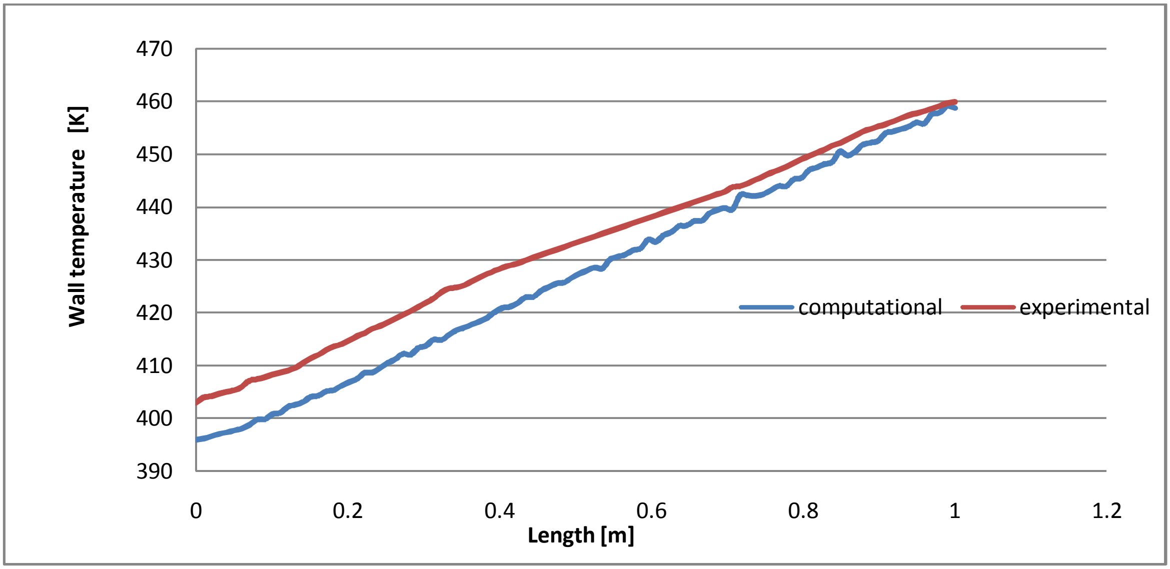


Fig 9. Wall temperature vs length plot

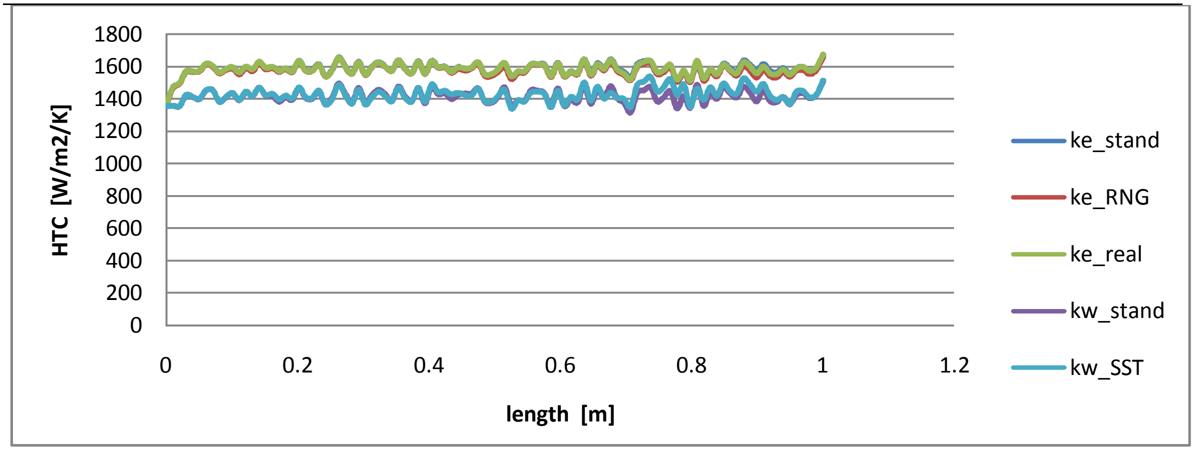


Fig. 10 Comparison of Heat transfer coefficient for various fluid flow models.

5. CONCLUSION

The computational results show that this is in good agreement with the experimental results and the predicted temperature and heat transfer coefficient is only ± 6 % of the experimental results. The heat transfer coefficient is analyzed for five fluid flow models like k-ε standard, k-ε RNG, k-ε realizable, k-ω standard and k-ω SST. Out of which, k-ε standard gives the highest heat transfer coefficient and hence is the preferable model for this study.From the above analysis of the 60° sub channels, the maximum temperature was 521 K and the maximum heat transfer rate was 1827 W/m^2K.

REFERENCES

[1] G. Richards, G.D. Harvel, I.L. Pioro, A.S. Shelegov, Heat transfer profiles of a vertical, bare, 7-element bundle cooled with supercritical Freon R-12, Nuclear engineering and design, 264, 246– 256,2013.

[2] International Atomic Energy Agency, Sustainable Development, and NuclearPower. Vienna. 1997.

[3] Romney B. Duffey, Igor L. Pioro, Experimental heat transfer of supercritical carbon dioxide flowing inside channels(survey), Nuclear Engineering and Design 235, 913–924, 2005.

[4] Krishna Pandey, Deepak Sharma, 3D CFD analysis of heat transfer of sub channels of nuclear reactors, Lap Lambert Academic Publishing, Germany, 2013.

[5] Amir Keshmiri, Three-dimensional simulation of a simplified advanced gas-cooled reactor fuel element, Nuclear Engineering and Design 241, 4122– 4135, 2011.

[6] Ajay Debbarma, K.M. Pandey, Numerical analysis of flow and heat transfer in sub-channel of supercritical water nuclear reactor, Procedia Engineering 64, 507 – 517, 2013

[7] William. M. Kopko, Beyond CFCs: Extending the search for new refrigerants, 13, 79-85, 1990.

[8] SahilGuptaa, Eugene Saltanov, Sarah J. Mokry, Igor Pioro, Liliana Trevani, Donald McGillivray, Developing empirical heat-transfer correlations for supercritical CO2 flowing in vertical bare tubes, Nuclear Engineering and Design 261, 116– 131, 2013)