Research Article
Transient Simulation of Thermal Stratification in Horizontal Pipe with Different Turbulence Models

Athar Rasool, Sun Zhongning and Wang Jianjun
College of Nuclear Science and Technology, Harbin Engineering University, Harbin,
Heilongjiang 150001, China

Abstract: Thermal stratification is a well-known phenomenon in pressurizer surge line of pressurized water reactor. The pressurizer surge line has a complex geometry it runs down vertically and horizontally with varying slopes and curvatures connecting pressurizer with the hot leg of the primary loop. The significant thermal stratification is usually observed in horizontal sections of the pressurizer surge line during reactor starting up phase (heat up) from cold shut down condition. To study the thermal stratification phenomenon a simple horizontal pipe model is considered. The pressurizer surge line is typically a 250 to 350 mm diameter stainless steel pipe. The dimensions of horizontal pipe are selected on the basis of pressurizer surge line dimensions. The horizontal pipe model inner diameter is 305 mm with thickness of 33.5 mm and length of four diameters. The transient analysis is performed using commercially available software ANSYS CFX with three different turbulence models (K-E, RNG K-E and SST). The transient temperature distribution along the horizontal pipe with different turbulence models is represented in this study. The transient temperatures obtained vary with selection of different turbulence models. Among the three different turbulence models, KE turbulence model predicts the lowest top to bottom inner and outer wall surfaces temperature differences.

Keywords: Pressurizer surge line, thermal stratification, turbulence models

INTRODUCTION

Pressurized Water Reactor (PWR) type Nuclear Power Plant (NPP) has been widely used as commercial power source. Pressurizer is a key component of the primary loop of a PWR. It controls and compensates the pressure and volume surges in the primary loop. The pressurizer is connected with the hot leg of the primary loop through pressurizer surge line making it a vital part. Any damage to the pressurizer surge line influence the integrity of primary loop and hence hampers the operation of NPP. A schematic diagram of PWR is depicted in Fig. 1.

Thermal stratification in pressurizer surge line is well known phenomena. When during in-surge or out-surge hot fluid from hot leg or pressurizer flows in the pressurizer surge line already occupied with the cold fluid, distinct layers of fluid is formed due to density difference. Such flow is categorized as thermally stratified flow and the related phenomenon is known as thermal stratification (IAEA, 2003).

The PWR type NPP operates at high pressure and temperature (15.5 MPa and 345ºC). The temperature difference between the hot leg fluid of the primary loop and the pressurizer fluid can be as high as 150ºC during heat up (Grebner and Hofler, 1995). Such high temperature difference in existence of thermal
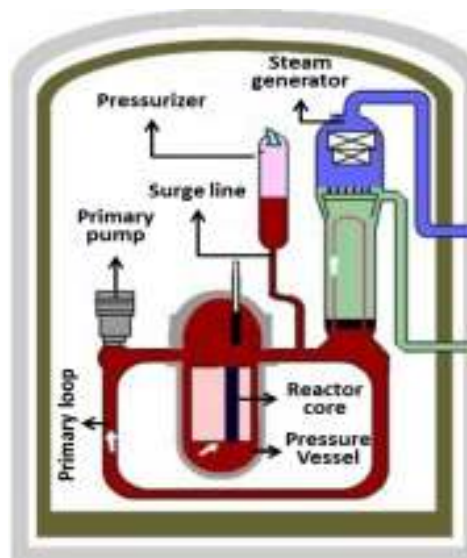

Fig. 1: Schematic diagram of Pressurized Water Reactor (PWR)

stratification can considerably affect the integrity of the pressurizer surge line (IAEA, 2003).

The damage to the pressurizer surge line due to thermal stratification has been reported since 1988

Corresponding Author: Athar Rasool, College of Nuclear Science and Technology, Harbin Engineering University, Harbin, Heilongjiang 150001, China

(NRC, 1988a, b, c). Initially thermal stratification effects have not been considered in the design of NPP. However, the new design of NPP incorporates the effects of thermal stratification phenomenon to the pressurizer surge line and subsequently is part of safety analysis report. Therefore it is necessary to analyze and evaluate the thermal stratification phenomenon in pressurizer surge line of PWR. Many researchers and scientists have used various methods and techniques to analyze this phenomenon (Kang and Jo, 2008). However with the breakthrough in computer technologies and advert of parallel processing it has been made possible to simulate such complex phenomena.

Pressurizer surge line has a complex geometry it runs down from the pressurizer till the hot leg vertically and horizontally with varying slopes and curvatures. A generic pressurizer surge line layout is depicted in Fig. 2. The thermal stratification is usually observed in horizontal sections (Grebner and Hofler, 1995), since vertical portion is near to the pressurizer and there is enough mixing in the sections with varying slopes and curvatures.

The flow in pressurizer surge line is considered as thermally stratified flow. Since, when two fluids at different temperatures flow through a space, the convection type can be determined based on the magnitude of the Richardson number (Ri), a non-dimensional parameter, which is directly proportional to gravity, thermal expansion, temperature difference, and characteristic length, and inversely proportional to the square of flow velocity. It is well known that, in the range of Ri above 10, the natural convection dominates the flow field so that a strongly stratified flow can be maintained until the temperature difference decreases enough to lower the value of Ri below 0.1 or less. The significant thermal stratification in the pressurizer surge line occurs in the two modes of reactor operation: starting up phase (heat up) at the reactor cold shut down condition and cooling-down at the normal operation condition. Both reactor operation modes are the bounding cases in terms of temperature gradients caused in the wall of the surge line, which may eventually result in undesirable mechanical damages on the pressurizer surge line and its supporting system (Kang and Jo, 2010).

In this study transient temperature distribution along the horizontal pipe with three different turbulence models for the reactor starting up (heat up) phase at reactor cold shutdown condition is analyzed. During heat up phase, hot fluid from the pressurizer is assumed to flow into the surge line, which initially is filled with cold water at the same temperature as that of
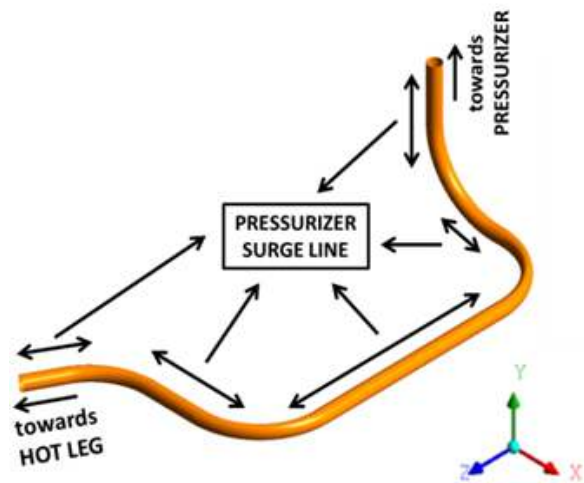

Fig. 2: Layout of pressurizer surge line

the surge line wall. The heat up phase simulation for horizontal pipe is performed using commercially available software ANSYS CFX.

MODEL DESCRIPTION

Geometric configuration and properties: The dimensions of horizontal pipe are selected on the basis of surge line dimensions. Generally pressurizer surge line is 250 to 350 mm diameter stainless steel pipe (Shah and Conley, 1989). The geometrical data and properties of pipe material and fluid are given in Table 1. IAPWS IF97, built in ANSYS CFX, is utilized for fluid properties in calculations (ANSYS, 2010a).

The horizontal pipe geometry configuration along with location of monitoring points (top to bottom inner and outer surface) and planes (at distance 1D to 3D) are depicted in Fig. 3.

Problem statement: In the present stratified flow problem, the Richardson number is very high, as shown in Table 1. Thus, the buoyancy force will strongly affect the flow field so that a typical thermally stratified flow will be formed inside the horizontal pipe.

Initially cold fluid at a specified temperature of 51.7ºC occupies the horizontal pipe maintaining a steady state condition. Then at a certain time hot fluid at a specified temperature of 218ºC begin to flow into the circular pipe at a fluid velocity of 0.05 m/sec. The fluid flow in the horizontal pipe is turbulent since the Reynolds number is large as shown in Table 1. The turbulent flow behavior is considered with three different turbulence models (K-E, RNG K-E and SST). For buoyancy calculations the density difference is

Table 1: Properties of pipe material and fluid

| Parameters	Values
Pipe material	ASME SA-312
Pipe Inner Diameter (ID)	305 mm
Pipe thickness (t)	33.5 mm
Density	8000 Kg/m^3
Thermal conductivity	16.3 W/m-K
Specific heat capacity	500 J/kg-K
Thermal expansivity	17e-6 /K
Fluid (water)	IAPWS IF97
Fluid initial temperature	51.7 C
Fluid inlet temperature	218 C
Fluid inlet velocity	0.05 m/sec
Reynolds number	$0.283×10^5$-$1.05×10^5$
Richardson number	187.86
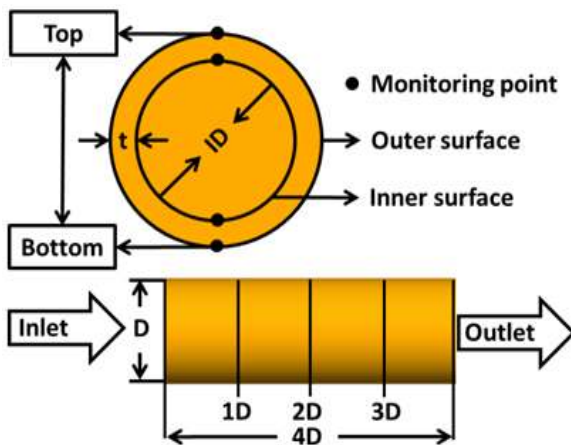
Fig. 3: Horizontal pipe layout and location of monitoring points and planes

evaluated directly using full buoyancy model. The reference pressure is equal to 2.2408 MPa.

Boundary conditions: The solution domain consists of horizontal pipe, which can be further subdivided into two sub domains, fluid and solid domains. The fluid domain consists of flow field inside horizontal pipe. The solid domain consists of the region bounded by the inner and outer wall surfaces of horizontal pipe. To take into account the Fluid Structure Interaction (FSI) Conjugate Heat Transfer (CHT) analysis is incorporated into the CFD analysis. In solid domain, where there is no flow and conduction is the only mode of heat transfer, the conservation of energy equation is simplified as the unsteady heat conduction equation. The Unsteady Reynolds Averaged Navier Stokes equations for conservation of mass, momentum, energy, and turbulent quantities used in this study can be expressed in a Cartesian coordinate system as in reference (ANSYS, 2010b; White, 1991).

CFD ANALYSIS

Model mesh: To simulate the flow as accurately as possible both flow and fluid solid interface boundary
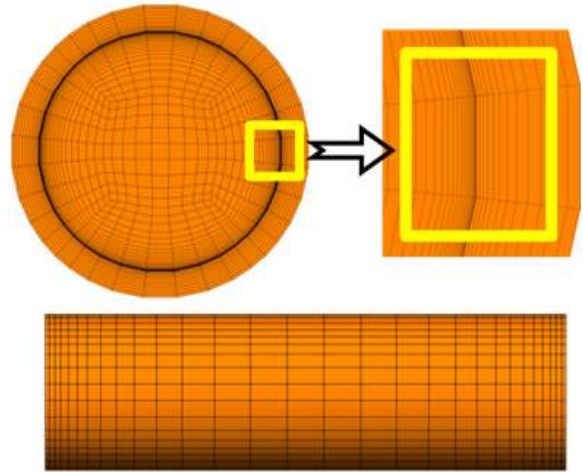

Fig. 4: Horizontal pipe grid scheme and grid density

regions are finely meshed using ANSYS ICEM. The grid density along the pipe length is less fine compared with radial or circumferential directions as shown in Fig. 4.

The fluid and solid domain is discretized into 43,065 hexahedral elements (30,537 elements in fluid domain and 12,528 elements in solid domain). The hexahedral elements type grid scheme is used due to its more realistic results. The overall mesh quality is greater than 0.65 on the scale from 0 to 1 based on ANSYS ICEM mesh quality criteria (ANSYS, 2010b). The aspect ratio, edge length ratio, element volume ratio and mesh expansion factor lies well within the specified criteria based on ANSYS CFX (ANSYS, 2010a). The same mesh is used for all simulations. The first node distance is approximated on the basis of boundary layer thickness for pipe ($\delta = 0.0324 D Re^{-0.875}$) and the boundary layer is resolved using over 10 nodes. The overall grid expansion ratio of 1.2 is used (Rasool *et al.*, 2011).

Problem setup: The governing equations are discretized using the finite volume approach. The CFD calculations are performed for total time of 1000 sec with time step size of 0.1. Flow near wall is treated using scalable wall functions for K-E and RNG K-E turbulence model and automatic wall functions for SST turbulence model, one of the built in user options in ANSYS CFX. Heat transfer in fluid and solid domains is simulated using total and thermal energy options respectively in ANSYSCFX, considering both conduction and convection in fluid domain and conduction in solid domain. The conservative interface flux option is selected for heat transfer between fluid and solid. High resolution advection scheme and second order backward Euler transient scheme is used. The convergence criterion for all major parameters is

determined at each time step when RMS residual is less than 10^{-3}. The convergence condition is met after three or four iterations at each time step, except for the initial time steps which requires up to ten or more iterations.

RESULTS AND DISCUSSION

The transient progressions of temperature difference between the top and bottom inner wall surfaces of horizontal pipe at a distance 2D for three different turbulence models are depicted in Fig. 5. The top to bottom temperature difference is large in the beginning and quickly reduces and later on progressively stabilizes almost to uniform temperature difference.

The large temperature difference in the beginning is due to the flow of hot fluid (218ºC) in circular pipe already occupied with cold fluid (51.7ºC). The hot fluid having lesser density settles in the upper region of the horizontal pipe thus elevating the top inner wall surface temperature and resulting in large top to bottom temperature difference. The primary mode of heat transfer between fluid molecules is convection which is much quicker thus sharply reduces the top to bottom inner wall surfaces temperature difference. The hot fluid inflow is constant which keeps on increasing the bulk fluid temperature and thus progressively increasing the bottom inner wall surface temperature which in result reduces the top to bottom inner wall surfaces temperature difference. The transient progressions of Top Inner wall Surface (TIS) and Bottom Inner wall Surface (BIS) temperatures for different turbulence models are depicted in (Fig. 6a, b and c).

The transient progressions of temperature difference between the top and bottom outer wall surfaces of horizontal pipe at a distance 2D for three different turbulence models are depicted Fig. 7. The top

Fig. 6: Transient progressions of top and bottom inner wall surfaces temperature for (a) K-E, (b) RNG K-E, and (c) SST turbulence models
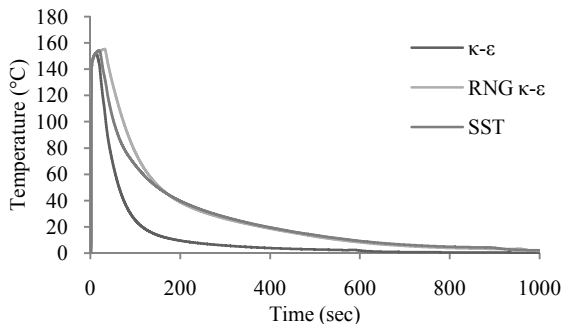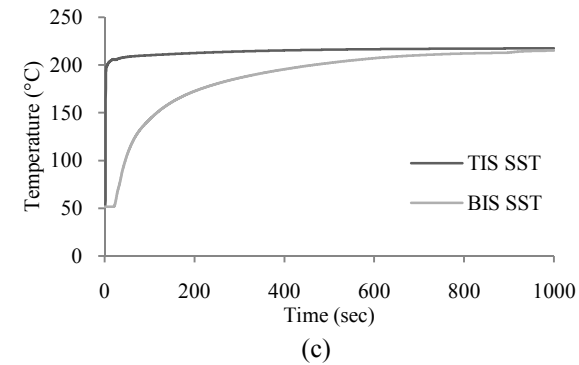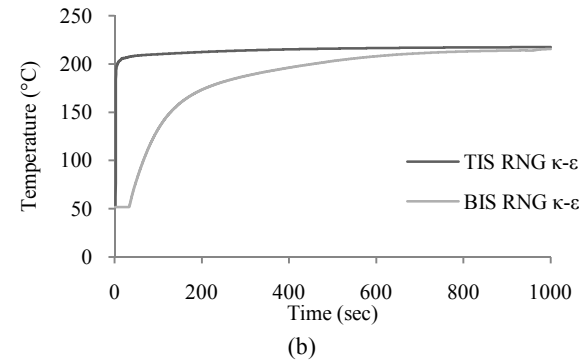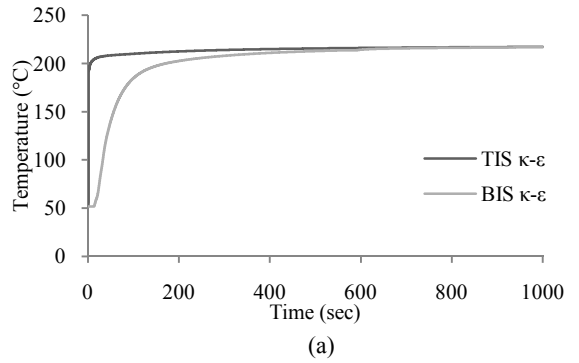

Fig. 5: Transient progressions of temperature difference between top and bottom inner wall surfaces of cross section at a distance 2D
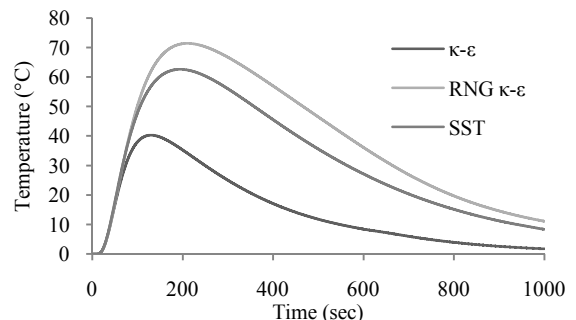

Fig. 7: Transient progressions of temperature difference between top and bottom outer wall surfaces of cross section at a distance 2D
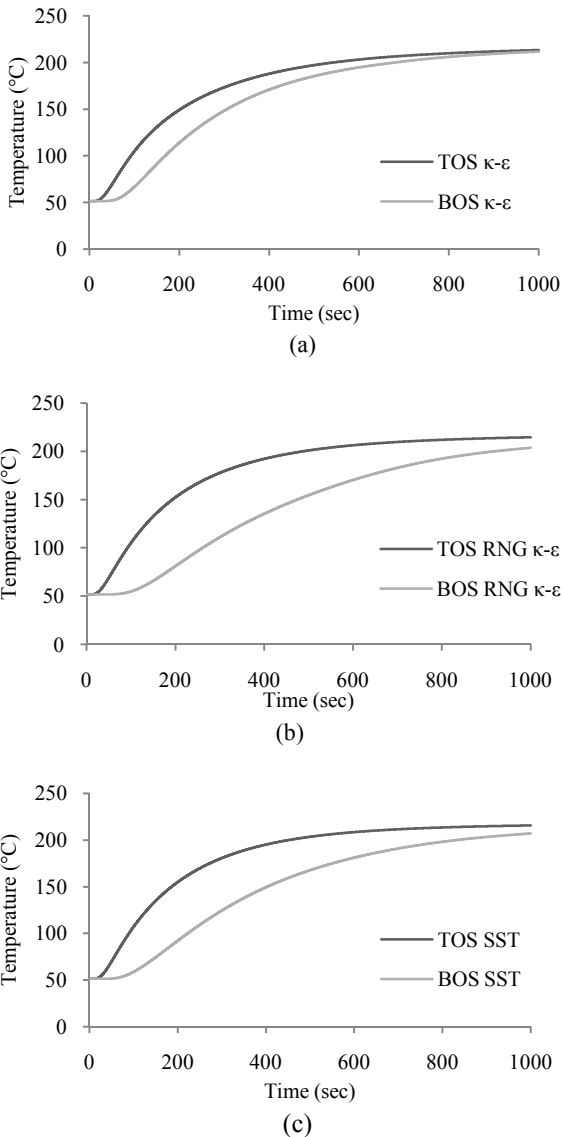
Fig. 8: Transient progressions of top and bottom inner wall surfaces temperature for (a) K-E, (b) RNG K-E and (c) SST turbulence models

Table 2: Transient top and bottom inner wall surfaces temperature and temperature differences at distance 2D for t = 200 sec

	Temperature (°C)		
	K-E	RNG K-E	SST
TIS	212.02	212.22	212.41
BIS	202.53	173.45	172.71
ΔTIS	9.49	38.77	39.70
TOS	149.35	152.47	154.96
BOS	114.13	81.12	92.36
ΔTOS	35.22	71.35	62.60

to bottom temperature difference increases initially and progressively decreases with the elapsed time.

The less quick progression of top to bottom outer wall surfaces temperature difference than top to bottom inner wall surfaces temperature difference is due to the mode of heat transfer. The heat transfer between the inner wall surfaces to the outer wall surfaces is only through conduction whereas convection takes place between fluid molecules and is much quicker than conduction. The bulk fluid temperature slowly progresses towards uniformity which results in comparatively large top to bottom outer wall surfaces temperature difference. The top to bottom outer wall surfaces temperature difference significantly decreases towards the end of simulation period. The transient progressions of Top Outer wall Surface (TOS) and Bottom Outer wall Surface (BOS) temperatures for different turbulence models are depicted in Fig. 8 (a, b and c).

The transient temperatures obtained vary with selection of different turbulence models. Among the three different turbulence models, the lowest top to bottom inner and outer wall surfaces temperature differences are obtained when K-E turbulence model is selected. SST turbulence model predicts almost similar temperature differences as of RNG K-E turbulence model. SST turbulence model predicts a slight more top to bottom inner wall surfaces temperature difference and slight less for top to bottom outer wall surfaces temperature difference than RNG K-E turbulence model. The transient top to bottom inner and outer wall surfaces temperatures and temperature differences at distance 2D for simulation time 200 sec are given in Table 2.

CONCLUSION

A simple horizontal pipe is considered to study the effects of thermal stratification on pressurizer surge line of PWR. The turbulent behavior fluid flow is analyzed with three different turbulence models. The results obtained by adopting K-E turbulence model predict the least top to bottom inner and outer wall surfaces temperature difference, whereas, RNG K-E and SST turbulence models predicts almost similar top to bottom inner and outer wall surfaces temperature difference. Both SST and RNG K-E turbulence models can be used to analyze the fluid flow behavior in the pressurizer surge line of PWR. However, the total time of simulation for identical conditions is considerably less for RNG K-E turbulence model.

ACKNOWLEDGMENT

REFERENCES

ANSYS, 2010a. CFX Manual. ANSYS Inc. ANSYS, 2010b. ICEM CFD Manual. ANSYS Inc.

Grebner, H. and A. Hofler, 1995. Investigation of stratification effects on the surge line of pressurized water reactor. Comput. Struct., 56(2): 425-437.

IAEA, 2003. Assessment and management of ageing of major nuclear power plants components important to safety, primary piping in PWRs. IAEA-TECDOC-1361.

Kang, D.G. and J.C. Jo, 2008. CFD analysis of thermally stratified and heat transfer in a PWR surge line. Proceedings of Pressure Vessel and Piping Division Conference, July 27-31, Chicago, Illinois, USA.

Kang, D.G. and J.C. Jo, 2010. CFD analysis of thermally stratified flow and conjugate heat transfer in a PWR surge line. J. Pressure Vessel Technol., 132: 021301-1-10.

NRC, 1988a. Thermal Stresses in Piping Connected to Reactor Systems. NRC, Bulletin, pp: 88-08.

NRC, 1988b. Pressurizer Surge Line Thermal Stratification. NRC, Bulletin, pp: 88-11.

NRC, 1988c. Unexpected Piping Movement Attributed to Thermal Stratification. NRC, Bulletin, pp: 88-80.

Rasool, A., S. Zongning and W. Jianjun, 2011. CFD analysis of PWR surge line subjected to thermal stratification. Adv. Mater. Res., 468-471: 78-82.

Shah, V.N. and D.A. Conley, 1989. Research needs for fatigue damage in PWR surge line and spray piping assessment. International Conference on Structural Mechanics in Reactor Technology, Anaheim, California, USA.

White, F.M., 1991. Viscous Fluid Flow. 2nd Edn., Mac-Graw Hill, New York.