International Journal of Scientific Research and Engineering Development-– Volume 9 Issue 1, Jan-Feb 2026
Available at **www.ijsred.com**

RESEARCH ARTICLE OPEN ACCESS

Computational Fluid Dynamics Study of Flow Behavior in pipe Bends

Utharkar Nikhil Kumar*,A.Mahaboob Basha**

*(Mechanical Engineering, CRIT College, Anantapur Andhra pradesh India
** (Mechanical Engineering, CRIT College, Anantapur Andhra pradesh India)

--************************----------------------------------

Abstract:

Flow-accelerated corrosion and erosion in pipe elbows is one issue that the nuclear power sector is now dealing with. The FLUENT commercial software, created and distributed by Fluent, Inc., is being used to run the simulations. ANSYS, another program from Fluent, Inc., was used to construct the model's geometry and mesh. The results of the simulations conducted thus far are documented in this report; baseline findings using the RNG k-ε turbulence model are shown. The diametrical pressure coefficient's estimated value and documented correlations correspond rather well. The elbow section displays plots of the velocities, pressure field, wall shear stress, and turbulent kinetic energy next to the wall.

Keywords — **FLUENT,ANSYS,RNG-K,velocities,corrolations,turbulance.**

--************************----------------------------------

INTRODUCTION

Through the International Nuclear Energy study Initiative (INERI), the U.S. Department of Energy (DOE) and the Korean Ministry of Science and Technology (MOST) have partnered to conduct a cooperative program of study into issues affecting the nuclear power industry. Flow-accelerated corrosion/erosion in pipe elbows is one such issue. Centrifugal and viscous forces combine to produce a powerful secondary flow in the plane corresponding to the pipe axis when a fluid—in this case, water—passes through a pipe elbow. Two counter-rotating vortices, one in each half of the pipe cross section, make up this secondary flow. If successful, this kind of monitoring can alert plant staff to possible issues before they arise, preventing expensive and invasive unplanned shutdowns.

Sandia National Laboratories is conducting Computational Fluid Dynamics (CFD) simulations of the flow in one elbow of the FAC test loop concurrently with the KAERI tests. The FLUENT software, created and sold by Fluent, Inc., is being used to run the simulations. The GAMBIT software, which is also available from Fluent, Inc., was used to construct the model geometry and mesh. Gaining a better understanding of the factors impacting the corrosion and erosion that take place within the pipe elbow is the aim of the simulations. The outcomes of the simulations conducted thus far are compiled in this report. The mesh and model geometry are explained in the following section. The chemical sensor and the corresponding support bracket that were utilized in the studies are included in the geometry.

1.2 FLOW ACCELERATED CORROSION IN NUCLEAR POWER PLANTS

The general definition of corrosion is the deterioration of a material through chemical interactions with its surroundings. Nuclear power stations experience several forms of corrosion in a range of circumstances Additionally, utilities have often taken the following preliminary actions when disposing of severely vulnerable or damaged areas:

1. Swap out worn parts for other parts made of the same material.
2. Swap out worn parts for other parts made of FAC-resistant material.
3. Use different parts made of the same material to replace the entire worn line.
4. Substitute alternate FAC-resistant material components for the most vulnerable sections or the complete susceptible lines.

18" elbow wall thickness decreased from 12.7 to 1.5 mm on feed-water pump inlet at Surry, 1986

Wall Thickness reduced from 10 to 1.5 mm on Feed-water piping at Mihama unit 3, 2004

Failure in a high pressure extraction line at Fort Calhoun in 1997

Failure downstream of the LCV in the reheater drain line at Millstone unit 2, 1991.

. Certain types of corrosion, like rusting steel in a damp environment, are typical, while others, like flow-accelerated corrosion, need specific attention because they affect the dependability and safety of the plant. Large sections of piping and fittings become thinner due to the FAC degradation mechanism, which can cause abrupt and even catastrophic failures as well as significant financial losses**.**

1.3 CONDITIONS REQUIRED FOR FAC

The following circumstances have been found to cause FAC degradation in fossil or nuclear power plants:

• Flow circumstances: single-phase and two-phase flow conditions where the liquid being flowed is water or a mixture of water and steam, the temperature is greater than 95 degrees Celsius, and the flow velocity is larger than zero. • Chemistry condition: a potential difference between the liquid and the carbon steel pipe wall should be present in the flowing liquid. The protective oxide layer in the flowing stream will dissolve as a result of this discrepancy. High magnetite solubility and quick magnetite removal.

Since 1981, there have been reports of FAC degradation-related failures and accidents at a number of nuclear power stations worldwide. However, until the severe elbow rapture downstream of a tee at the Surry Unit 2 power plant (USA) in 1989, which resulted in four fatalities, major plant damage, and a plant shutdown, a thorough examination of the FAC-related failures had not begun. Yurmanov and Rakhmanov reported a large-scale steam leak in 1999 from the shell side of a feed-water heater at the Point Beach power station (USA) (In Figure). The Mihama nuclear power station Unit 3 (Japan) experienced a deadly pipe burst in 2004 downstream of a condensate system opening caused by FAC.

Failure of 14" heater drain extraction line to high pressure heater at Arkansas unit 2, 1986

Failure of the Feed-water Heater Point Beach Unit 1, 1999

LITERATURE REVIEW

Shih et al [2]: The work is devoted to numerical simulation of pulverized-coal combustion processes in the vortex furnace which is a prospective design of a boiler unit for thermal power plants. New modification of this design characterized by additional tangential-injection nozzle located at the bottom of combustion chamber has been studied. Numerical results for the case of Siberian brown coal combustion in this vortex furnace with dual-port loading are presented, including 3-D aerodynamic structure, the fields of temperatures, radiated heat fluxes, species and dispersed phase concentrations, and NOx emissions.

Xiao Yang Lu et al [12]: On the basis of the experiments, hot pushing pipe bending process on ox horn core bar is simulated by using MSC.Marc software and orthogonal testing method. Simulations and optimization of sixteen possible combinations of the three parameters T, v and f are carried out with orthogonal testing method L16(43). Aimed at the uniform wall-thickness of elbow pipes, the influence regularity and the influence degree of heating temperature T, pushing speed v and friction coefficient f on the pipe quality are analyzed. The best parameter of hot pushing pipe bending process are T=750 °C, v=4 mm/s and f=0.16.

Uwe Deikmann et al [13]: This paper presents the finite element model developed for the simulation of pipe elbow production by the so-called 'Hamburg process' in order to improve productivity and resource efficiency. To optimize the tooling design, a sensitivity analysis of the tool parameters that influence the quality of pipe elbows, such as mandrel height and length, is conducted. Different materials data sets including damage models were considered. Using numerical simulations, it is possible to determine an optimized tool geometry for the production of specific pipe elbow dimensions. **E Salas-Zamarripa et al [15]:** This work presents the implementation of the hot-forging process of seamless elbow fittings into a finite element model (FEM) developed with the aid of a commercial software code. The strain gradients predicted by the FEM were compared with the strain distributions computed by a visioplastic analysis. Measurements made on forged elbows are used to validate the predictions of the FEM. Once the validation was successful, the FEM is used to evaluate the effect that the tooling design exerts on the final dimensions and characteristics of formed elbows; this was done by changing the design of the tooling (mandrel).

INTRODUCTION TO SOFTWARES USED

INTRODUCTION OF ANSYS

A general-purpose program for finite element analysis (FEA) is called ANSYS. These days, the ANSYS package is crucial in every industry. Static and dynamic structural analysis, steady state and transient heat transfer problems, mode-frequency and buckling Eigen value problems, and a variety of field and coupled-field applications are among the analysis capabilities of ANSYS.

ANSYS Workbench is a computer-aided finite element modeling and analysis program that was created by ANSYS Inc., USA. ANSYS Workbench's Graphical User Interface (GUI) allows the user to create 3-dimensional and FEA models, conduct analysis, and produce analysis results. It can also be used to carry out a number of tasks, such as Design Assessment, Finite Element Analysis, and Product Optimization Analysis.

The following is the list of analyses that can be performed by ANSYS Workbench:

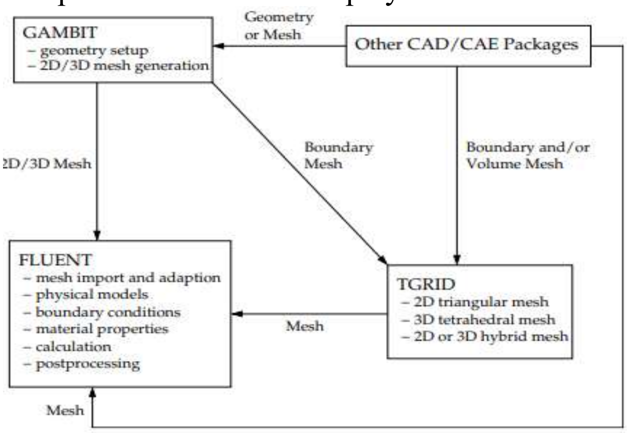
1. Design Assessment
2. Fluid Flow (CFX)
3. Harmonic Response
4. Linear Buckling
5. Modal
6. Rigid Dynamics
7. Static Structural
8. Steady-State Thermal
9. Transient Structural

10. Transient Thermal

INTRODUCTION TO FLUENT INC.

A cutting-edge computer program called FLUENT is used to simulate fluid flow and heat transfer in intricate shapes. Complete mesh freedom is offered by FLUENT, including the ability to address your flow issues with unstructured meshes that are rather easy to generate about complex geometries. 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes are among the mesh types that are supported. Additionally, FLUENT lets you adjust the grid's coarseness or refinement according to the flow solution. FLUENT fully utilizes the flexibility and power provided by the C programming language.

As a result, effective data structures, flexible solver control, and true dynamic memory allocation are all feasible. Furthermore, FLUENT's client/server architecture enables it to operate as independent concurrent processes on both powerful compute servers and client desktop computers. Effective execution, interactive control, and total adaptability across various machine types or operating systems are all made possible by this design. Through an interactive, menu-driven interface, FLUENT provides access to all functions needed to compute a solution and display the results.

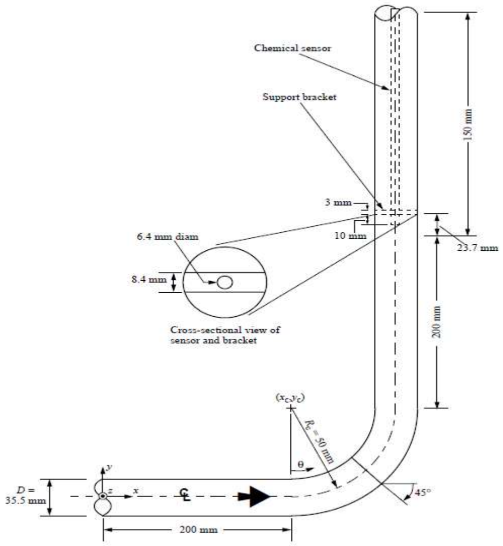


The FLUENT serial solver uses a single solver process on a single machine to handle data storage, file input and output, and flow field computations. Additionally, FLUENT makes use of a program named cortex, which controls its user interface and fundamental graphical features. With the help of FLUENT's parallel solver, you can compute a solution utilizing several processes running on separate computers in a network or on the same computer. FLUENT, a host process, and a group of compute-node processes interact during parallel processing. FLUENT uses the cortex user interface tool to communicate with both the host process and the group of compute nodes.

MESH AND MODEL GEOMETRY

There is a "tee" in the pipe where we believe our output to be in the real FAC test loop. In order to give the flow time to adapt to a condition where a uniform outlet boundary condition could be applied sensibly, we would have needed to include a considerable amount of pipe downstream of the tee in order to incorporate the tee into our model. As a result, the model would be significantly larger, the meshing process would be more difficult and time-consuming, and the number of mesh cells would increase significantly. Furthermore, the tee is far enough downstream that its existence shouldn't significantly affect the elbow's flow. It is reasonable to assume that it will have less of an impact than the chemical sensor.

A cross-sectional view of the mesh in the section of the model upstream of the chemical sensor is displayed in Fig. 4.2. Observe that the mesh along the pipe wall is much refined. The thickness of the cell next to the wall is required to be 0.15 mm; as the cell gets farther away from the wall, its thickness progressively rises. This was achieved by affixing to the pipe wall what GAMBIT calls a Boundary Layer Mesh. The remaining space was paved with quadrilateral elements with a nominal size of 0.5 mm to produce the mesh outside the border layer. A volume mesh of hexagonal cells was created by "sweeping" this 2D surface mesh along the pipe's axis.



CONCLUSIONS

The exact conclusions that have been reached are as follows:

1. All of the anticipated flow field's qualitative characteristics are consistent with the body of existing literature. A reported correlation based on experimental data is reasonably quantitatively consistent with the simulation's projected value for the diametrical pressure coefficient defined by Eq. (8). This offers us some assurance regarding the accuracy of the numerical findings.

2. Based on our intuition, we concluded that the largest corrosion/erosion would occur on the outside radius of the bend and that its location would correspond with the maximum wall shear stress, even though it was not modeled in these calculations. Nevertheless, the models show that the inside radius, directly downstream of the elbow entrance, experiences the most wall shear.

3. We thought about the idea that the corrosion might possibly be caused by another fluid mechanical event, like turbulence. However, Eq. (3), a plot of turbulent kinetic energy at the wall, shows that its maximum also happens on the inside radius, immediately downstream of the elbow entrance.

4. The baseline simulation, which used the RNG k-ε turbulence model, served as the foundation for the aforementioned conclusions. The standard k-ε and realizable k-ε models were also used in simulations to see how much the turbulence model selection may have affected the outcomes. The conventional and RNG models produced remarkably comparable simulation results.

5. The flow field close to the elbow was not significantly affected by the use of FLUENT's radial equilibrium pressure distribution option in the PRESSURE OUTLET boundary condition.

6. Simulations conducted both with and without the chemical sensor and the support bracket that goes with it show that they have very little effect on the flow around the elbow. Therefore, it is not a result of the sensor being added to the flow that the maxima in wall shear stress and turbulent kinetic energy occur on the inner radius.

7. As anticipated, the sensor and its support bracket have the primary effect of significantly increasing the pressure loss in the straight pipe segment **downstream of the elbow.**

References

[1] Yakhot, V. and Orszag, S. A., "Renormalization Group Analysis of Turbulence: I. Basic Theory," Journal of Scientific Computing, Vol. 1, No. 1, pp. 1-51, 1986.
[2] Shih, T.-H., Liou, W. W., Shabbir, A., and Zhu, J., "A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows—Model Development and Validation," Computer Fluids, Vol. 24, No. 3, pp. 227-238, 1995.
[3] L. Wang, D. Gao, and Y. Zhang, "Numerical simulation of turbulent flow of hydraulic oil through 90° circular sectional bend," Chinese Journal of Mechanical Engineering, p. 3901, 2012.
[4] Richard A "Simulation-Driven Design Optimization: A Case Study on a Double 900 Elbow Bend" International Journal of Modeling and Optimization, Vol. 4, No. 6, December 2014
5] W. Yang and B. T. Kuan, "Experimental investigation of dilute turbulent particulate flow

inside a curved 90° bend," Chemical Engineering Science, vol. 61, no. 11, 2006, pp. 3593-3601.
[6] Elok A Devi "Paleofacies of Eocene Lower Ngimbang Source Rocks in Cepu Area, East Java Basin Based on Biomarkers and Carbon-13 Isotopes" Recent Adv Petrochem Sci 5(1): RAPSCI.MS.ID.555654 (2018)
[7] YUSUF "Elbow Pipe Design Optimization Of Oil And Gas Pipeline System Owned By Joint Operating Body Pertamina-Petrochina East Java Job P-Pej Tuban Based On Reliability"
[8] N Crawford " A numerical investigation of the flow structures and losses for turbulent flow in 90° elbow bends" First Published December 23, 2008 Research Article.
[9] Pengfei Wang "Numerical Investigation on Fluid Flow in a 90-Degree Curved Pipe with Large Curvature Ratio" Mathematical Problems in Engineering, Volume 2015, Article ID 548262,.
[10] PrasunDutta "Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling" Engineering Science and Technology, an International Journal, Volume 19, Issue 2, June 2016, Pages 904-910

[11] Juan C "Computational Fluid Dynamic Modeling to Determine the Resistance Coefficient of a Saturated Steam Flow in 90 Degree Elbows for High Reynolds Number" Journal of Fluids Engineering | Volume 141 | Issue 11 | research-article

[12] Xiao Yang Lu " Numerical Simulation and Parameter Optimization of Hot Pushing Elbow Pipe Bending Process"

[13] Uwe Deikmann "Optimization of Tooling Design for Hot Mandrel Bending of Pipe Elbows" Materials Science Forum, Vol. 918, pp. 159-164, 2018

[14] D. Tabakajew "Increasing the efficiency of hot mandrel bending of pipe elbows" Procedia Engineering 207, 2310–2315 (2017)
[15] E Salas-Zamarripa "Finite element model simulation of hot forging of seamless elbow fittings" Volume: 219 issue: 2, page(s): 85-90, Issue published: April 1, 2005

