Simulation of Fluid Structure Interaction of Heat Exchanger Tube Using ANSYS CFX

Abubakar Izhar[1], Arshad Hussain Qureshi[2] and Shahab Khushnood[3]

[1]University of Engineering & Technology, Lahore, Pakistan.
[2]University of Engineering & Technology, Lahore, Pakistan.
[3]University of Engineering & Technology, Taxila, Pakistan.
E-mail: bakarizr@gmail.com

Abstract: Fluid structural interaction analysis has become an important part in the design process of different industry components like heat exchangers, IC engines etc, and other equipment subjected to flow induced vibrations. ANSYS is a commercial simulation environment and is able to accurately predict fluid structural interaction behavior numerically employing CFX-MFX coupling. This paper deals with the application of these modules to predict flow induced vibration in the tube of a heat exchanger. A complete 3D and a symmetric unit cell simulation of tube bundle is carried and temperature and fluid force effects are observed.

[Abubakar Izhar, Arshad Hussain Qureshi and Shahab Khushnood. **Simulation of Fluid Structure Interaction of Heat Exchanger Tube Using ANSYS CFX.** *Life Sci J* 2013;10(2):2317-2328] (ISSN:1097-8135). http://www.lifesciencesite.com. 324

Keywords: Fluid structure interaction, Flow induced vibration, Multifield simulation.

1 Introduction

Fluid structural interaction analysis has become an important part in the design process of different industry components like heat exchangers, IC engines etc, and other equipment subjected to flow induced vibrations like the offshore risers. flow induced have also been exploited to enhance the heat transfer rates in heat exchangers as explained in Zohir,(2011). Theoretical prediction Pettigrew & Taylor (2003), as well as experimental analysis Connors (1970), Chen (1981), Chen (1987). Tanaka & Takahara (1981) is rich in literature as compared to numerical assessment of FI. some numerical treatment of the behavior is found in Piperno (1997) Gillen & Meskell (2009). this study is an attempt to assess the FI behavior numerically using CFX. There are two main approaches for solving a Fluid Structure case the monolithic approach and the partitioned approach. In the monolithic approach a single solver is used to solve the equations for solid and fluid domains whereas in a partitioned approach a coupling is required between the two solvers and data is interpolated between the solvers through a fluid solid interface which has an identical reference number on both sides. Coupling is done in two ways one way coupling and two way coupling. As the name indicates in the one way coupling the data is exchanged from one side to the second side and no data is transferred from the second side to first side i.e. data transfer is uni-directional. It represents weakly coupled physics such as a thermal stress problem, whereas in two way coupling the data is transferred both ways different variables are transferred through either side. Two way coupling is used where the variables on both sides influences the results at either side. Various commercial packages are used to solve different FSI cases. ANSYS and CFX can be coupled through MFX using standalone models in Mechanical APDL or ANSYS Mechanical and CFX. This coupling allows to transfer various combination of variables at either side at the Interface i.e. only surface mapping is allowed. Both codes are iteratively coupled. During the outer loop simulation proceeds.during the stagger loop the coupled codes iterate within the given step until a complete implicit solution is obtained. The implicit coupling procedure is critical for obtaining an accurate fluid structure interaction solution. In the stagger loop ANSYS structural and ANSYS CFX can run either simultaneously or sequentially. Variables are exchanged at each stagger iteration. There's another coupling method in ANSYS known as MFS code coupling. MFS code coupling allows volume mapping whereas MFX allows surface mapping. CFX cannot be coupled through MFS as it is single code coupling. ANSYS Flotran (CFD) can be coupled to ANSYS structural through MFS. Both surface and volume mapping of variables can be employed through field surface interface and field volume interface respectively together with a remeshing control. Various other packages are used for FSI coupling such as Fluent allows for steady state one way mapping of thermal and structural loads to the structural model using either surface or volume mapping. Various structural modeling softwares like Abaqus, ANSYS Mechanical APDL can be used for this data transfer. Fluent can also be coupled with MPCCI (Multiple Code Coupling Interfae) to other structural analysis software like Abaqus to setup a two way fluid structure interfacing.

several authors have used FSI 3-D simulation for cross flow induced vibration in tube bundles Eisinger, et al. (1995) developed a numerical model for fluid structure coupling analysis by connecting a numerical solver to unsteady flow model of Chen (1983) and provided a comparison between the theoretical and experimental deflection data. showed the flow pattern using LED (Large Eddy Simulation) in a 4x4 and 5x5 channel. Kim and Mohan,(2005) calculated forces on tube for high Reynolds no cross flow with LES.Karl, et al. (2009) have performed unit cell CFD simulations for tube bundles in ANSYS validating the results of (Simonin and Barcouda 1986) and employed fluid structure interaction through MPCCI(Multi Physics Code Coupling Interface) to find the deflections of tube. Also Karl, et al. (2009) carried out FSI using ANSYS CFX and MFX coupling of a Hydrofoil and also validated the experimental results of Hover and Techet (1998) for tubes through ANSYS FSI simulation. Oakley, Constsantinides and Holmes (2005) employed DES model to study fluid structure coupling effects. Hossin (2010) and Hossin and Hassan (2011) have used CFX to simulate Fluid elastic instabilities in normal square and normal triangular tube arrays. They have used CFX to find the force coefficients in unsteady flow schemes of Chen (1991) and Tanaka and Takhara (1981) and have predicted the force coefficients with much accuracy and have generated a series of Fluid elastic instability curves covering a wider range of reduced velocities and pitch to diameter ratios. A complete 3D model Thermal Fluid Structure Interaction (TFSI) is performed in first section for research type heat exchanger with 10 tubes of 12.9mm outer diameter and P/D ratio of 1.22. The tubes are in a normal triangular array. Flow induced deflections and stresses are found in tubes for a steady state case. In the following section a unit cell simulation for a tube bundle is employed and a coupled unsteady FSI analysis is carried out to find the flow induced deflections and stresses in a tube span. Damping is also incorporated based on empirical formulas from previous literature like Karl, et al.(2009) and Khusnood (2005).Figure 1 shows the view of flow of unsteady fluid structure interaction between the CFX and ANSYS mechanical sides as described in (ANSYS CFX Rel 12.0 Documentation 2012)

2 Interpolation Methods

MFX Multifield simulations can do run with a mismatched mesh between the mechanical and CFX applications on the boundary between them. an interpolation method is used in such cases to interpolate data from one mesh onto other. There are two types of interpolation schemes used in CFX: Profile Preserving, and Conservative.

Profile preserving simply take the profile of the variable like temperature on one mesh and matches or maps it to the other mesh as accurately as it can. Conservative interpolation ensures that the profile is interpolated in such a way as to ensure that a total quantity passing across the interface is conserved i.e. same total passes out of one mesh and into the other like a conservative interpolation for heat flow variable ensures that the total heat flowing out of one mesh is identical to the total heat flowing into the other mesh even if the mesh resolution is poor. In general, temperatures and displacements are sent using the Profile Preserving and heat flows and forces are sent using Conservative method. Mesh motion is specified as a part of the domain specification with mesh motion option set to regions of motion specified and if CFX is in Multifield mode then mesh motion setting on the boundary condition form for a wall can be set to ANSYS Multifield. This automatically sets up the CFX simulation to receive its mesh displacement from the mechanical application solver. If the model has a heat transfer model specified as part of the domain specification and CFX is in Multifield mode then the heat transfer setting on the boundary condition form for a wall can be set to Multifield. This automatically sets up the CFX simulation so that it can send the temperatures to ANSYS Mechanical. A fluid and solid domain can meet at a face known as fluid solid domain interface or different fluid domains can meet at a face known as fluid domain. Energy can flow through these domain interfaces. These two types of interfaces are automatically created when domains are formed. The interface forming method can be one per interface type or one per domain pair. In this case one per domain pair is used so that distinct interfaces are created for each tube so that tubes can be analyzed through FSI separately. Two types of mesh connection methods are available, direct and general grid interface. The direct connection is available if after any applicable rotational or translational transformation all of the nodes on one side of the interface correspond in location with all of the nodes on the other side of the interface to within a spatial tolerance governed by the Mesh Match Tolerance parameter which can be edited. GGI refers to the class of grid connections where the grid on either side of the two connected surfaces does not match. In general, these connections permit non-matching of node location, element type, surface shape and flow physics across. GGI connections are made in a conservative and implicit fashion even if the nodes on the two sides of the connection are not aligned. Likewise element types on both sides can be of different types. Even if the size of the connection region on one side is different than that on the other side the connection will be made automatically between the mutually overlapping surfaces.

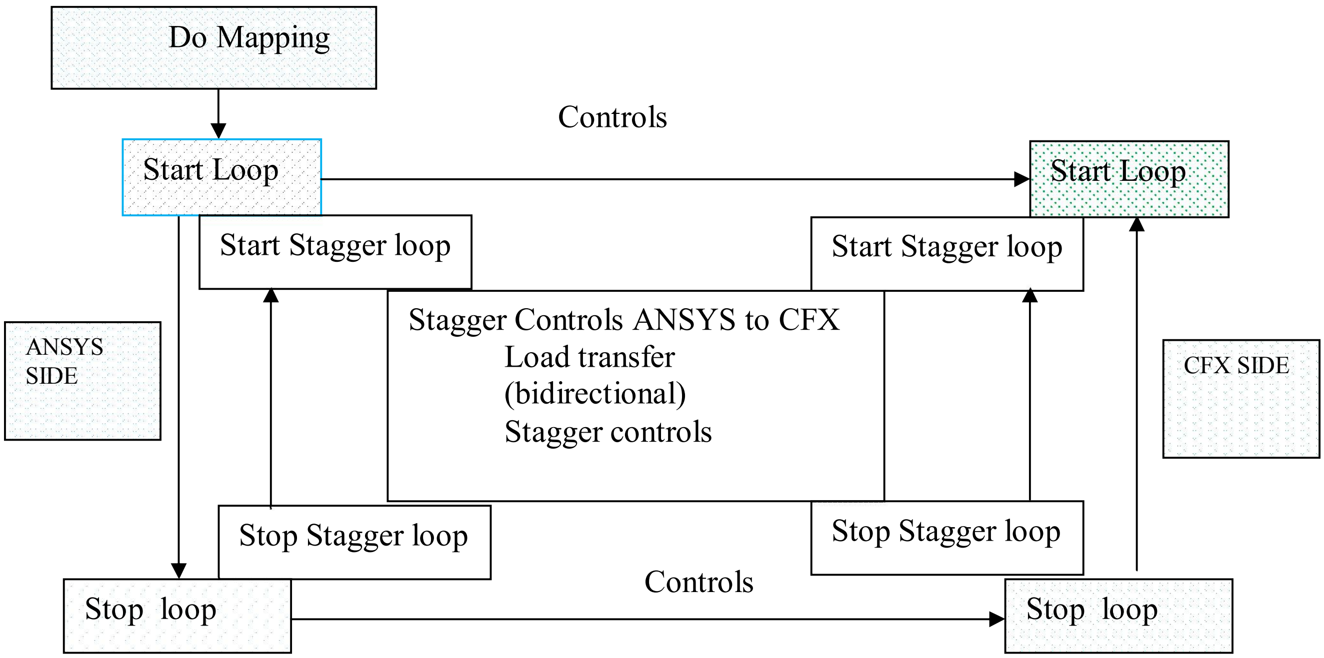

Figure 1 Flow diagram for unsteady fluid structure interaction.

3 Three Dimensional Simulation

3.1 Steady State Thermal

A steady state thermal analysis was performed in ANSYS CFX. Thermal energy was also enabled in the solid for a conjugate heat transfer simulation to get temperature distribution within the solid tube. The temperatures and fluid forces obtained through this analysis were transferred from CFX side to ANSYS Mechanical side. The meshes of two sides were mapped and data was transferred within ANSYS workbench. Temperature inlet of 3390K and velocity inlet of 6m/ was used. Pressure outlet of 0 Pa was imposed. K-epsilon turbulence with high resolution advection scheme was employed. RMS residual of.001 was used as convergence criteria.

The convergence behavior for the tube side, cylinder side and tube temperatures are shown in Figure 2An almost steady state is reached after about 392 iterations. Figure 3 shows the steady state temperature distribution on cylinder side and temperature is apparent from this contour plot, as the fluid moves through the cylinder across the baffles about 6^0 K decrease in temperature is observed due to heat transfer to the tube side. Figure 4 gives temperature distribution of the fluid for the tube side and about 3.9 K increase in temperature seen. Figure 5 shows convective heat transfer coefficient profile inside the cylinder. It can be seen from this profile that as the fluid passes over the baffles there's a variation in the fluid velocity which in turn effects the convective coefficient. Figure 6 shows the forces on tube side through CFX simulation. Heat transfer results are verified with empirical formulas of Kern (2011) in the following section.

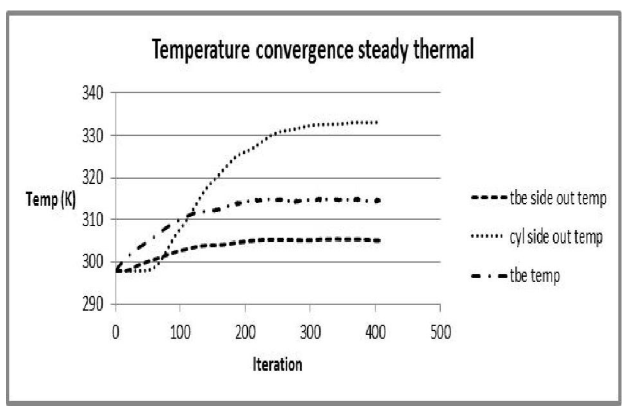

Figure 2. Temperatures' Convergence for Heat Exchager.

3.1.1 Validation

This method was based on experimental work on commercial exchangers with standard tolerances and gives a reasonably satisfactory prediction of heat transfer coefficient. The prediction of pressure decrease is less satisfactory as the pressure decrease is more affected by leakage and bypassing and this method doesn't take into account these factors. The friction factor sf and heat transfer factor sh used in calculations are obtained from the figures based on the data given by Kern and Ludwig.

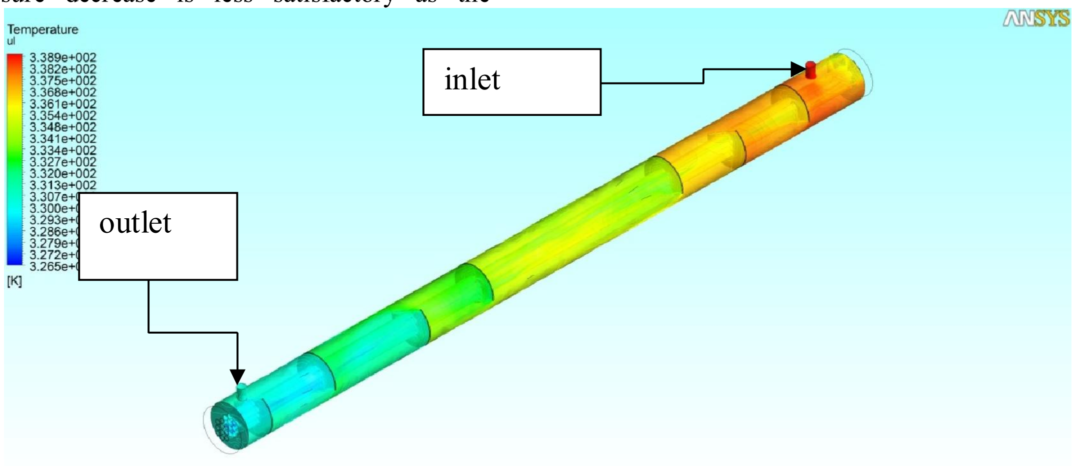

Figure 3 Steady state temperature profile in heat exchanger.

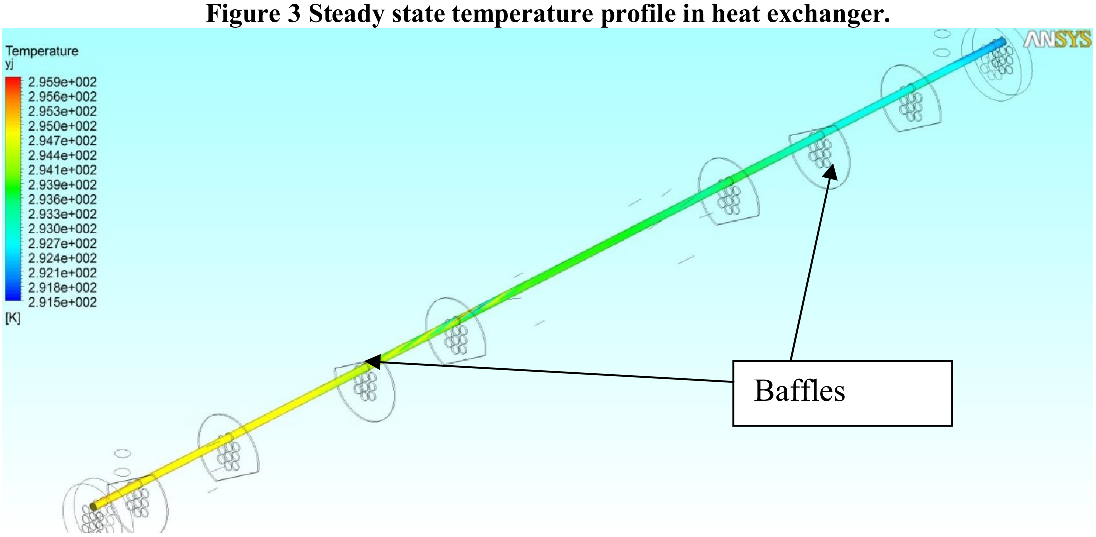

Figure 4. Steady state temperature profile in heat exchanger tube side.

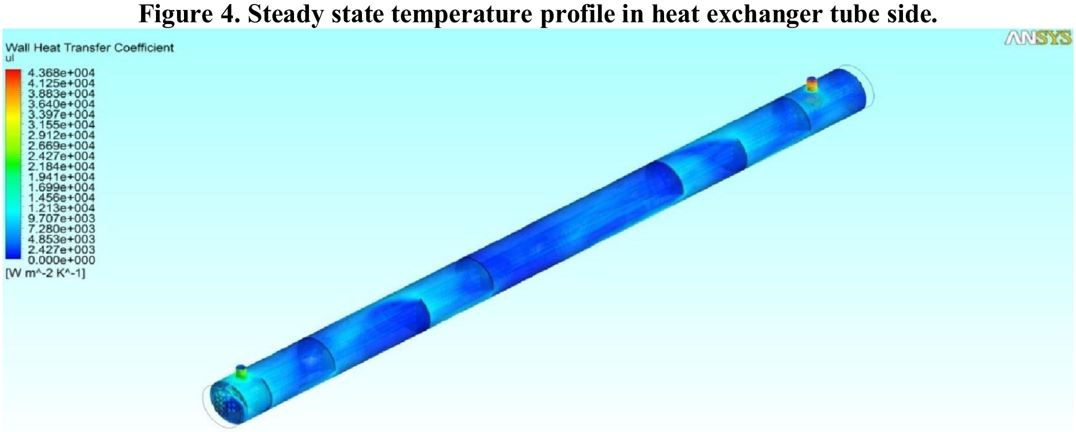

Figure 5. Steady state convective heat coefficient profile in heat exchanger

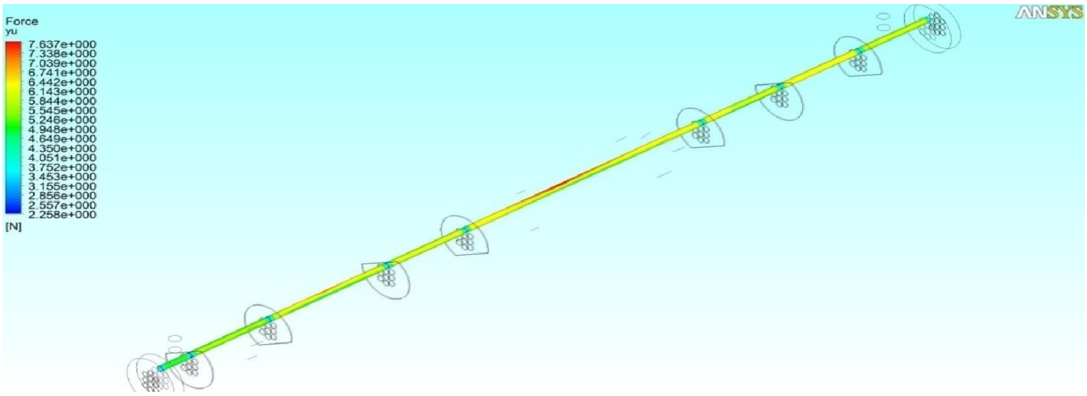

Figure 6 Force Profile on Fluid Solid Boundary.

These factors can be used for the calculated Reynolds number for various baffle cuts and pitch arrangements. The heat transfer coefficient outside the tubes is calculated by following the procedure of Kern (2011) as follows. Cross flow area as for a hypothetical row of tubes is calculated as

$$a_{s=}\frac{(p_t - d_o)d_s l_b}{p_t} \qquad (1)$$

Where p_t=tube pitch=15.875mm, d_o=outside tube dia=12.9mm, d_s=cylinder inside dia=122mm, L_b=baffle spacing=17.9. The term $(p_t\text{-}d_o)/p_t$ is the ratio of the clearance between tubes and the total distance between tubes center. With these values a_s comes out to be 14.4e-6m^2. Cylinder side linear velocity (Us) and mass velcity,(gs) are calculated as follows

$$g_s = \frac{W_s}{A_s} \quad , \quad U_s = \frac{g_s}{\rho} \qquad (2)$$

Where Ws=flow rate outside tubes (kg/s), ρ= density of fluid (kg/m^3), using a flow rate equal to.06kg/s the linear velcity comes out to be approx 4.2m/s. The cylinder equivalent dia (hydraulic dia), d_e is calculated using the flow area between the tubes taken in axial direction. The relation for equilateral triangular pitch arrangement of tubes is given by Eq (3)

$$d_e = \frac{1.10}{d_o}\left(p_t^2 - .917 d_o^2\right) \qquad (3)$$

Where d_e is equivalent diameter (m).using the above values d_e comes out to be 5.679e-3m^2. The Reynolds no is then calculated using the relation

$$\mathrm{Re} = \frac{\rho U_s d_e}{\mu} = 27499 \qquad (4)$$

For the calculated Reynolds number, value of heat transfer factor s_h is obtained from the graph for the selected baffle cut and tube arrangement, and cylinder side heat transfer coefficient h_s is calculated from Eq (5)

$$nu = \frac{h_s d_e}{k_f} = s_h Re Pr^{.33}\left(\frac{\mu}{\mu_w}\right)^{.14} \qquad (5)$$

Where k_f is thermal conductivity of the fluid, μ is viscosity at the mean cylinder side temperature and μ_w is viscosity at average wall temperature. The last term is applied for the viscosity correction. j_h is read against the Reynolds number from the figure to be 3.69e-3, Pr=5.59,k_f=.649, d_e=5.679e-3m^2. For these values without using the viscosity term h_s comes out to be approx 20622. The average value from the CFX simulation is about 12000 W/m^{2}K.

3.2 Static Structural

Static fluid structural analysis is performed to see the effect of fluid force and temperature on the tube structure by importing the loads from fluid side to structural side. The structural case is set up by using a fixed support condition at the two edges of the tube and therefore the complete span of tube is used in structural analysis.

Figure 7 shows results with temperature and pressure loads and it is seen that largest deflection of about 7mm occurs at the center, similarly Figure 8 shows deflection with fluid pressure loads with a deflection of about 10mm at the center. Figure 9 and Figure 10 shows the equivalent stresses with temperature and pressure loads respectively.

3.3 Unsteady Structural

Unsteady simulation was run to see the temporal flow induced effects. Structural case was setup with the fixed edges of the tube with 4 bar pressure with one way fluid-structural coupling setup in ANSYS CFx for the complete 3D case. Step size of.002 was calculated based on dividing the period of the vibration of tube into twenty steps to ensure stability of the unsteady computations.

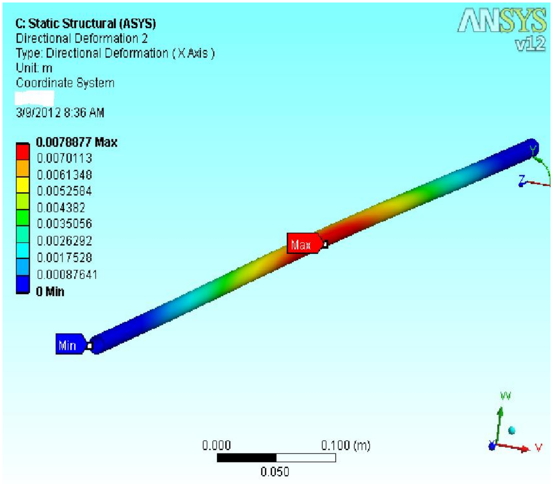

Figure 7 Deflection with Pressure and Temperature Loading.

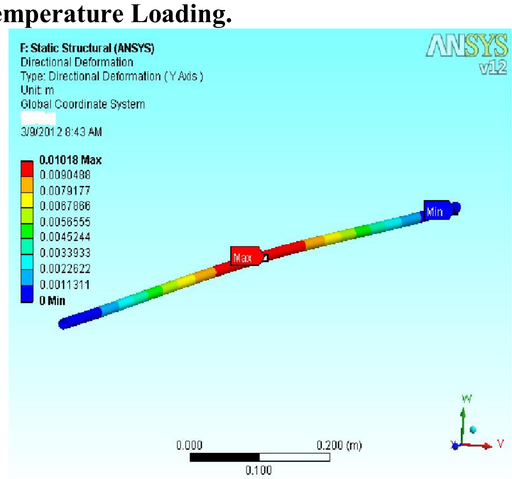

Figure 8 Deflection With Pressure Loading.

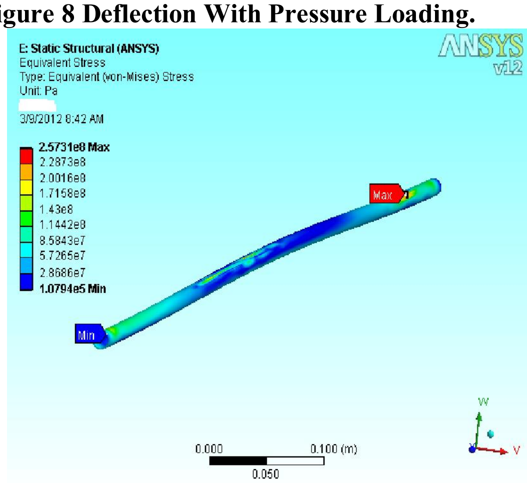

Figure 9 Equivalent Stress With Temperature Loading.

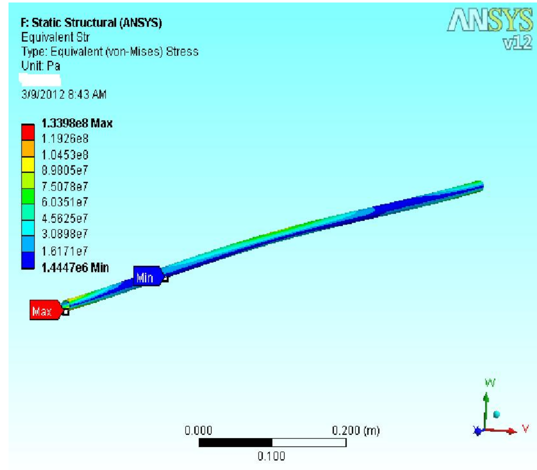

Figure 10 Equivalent Stress With Pressure Loading.

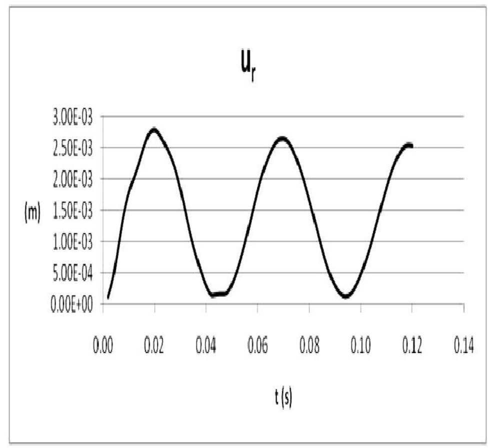

Figure 11 Dynamic Flow Induced Deflection (Complete Span) In Radial Direction (m).

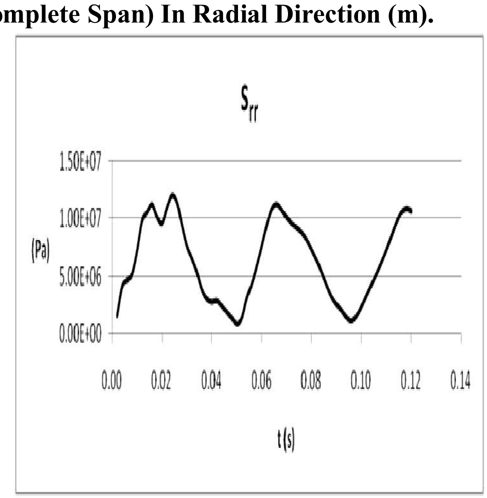

Figure 12 Dynamic Flow Induced Direct Stress Radial Direction (Pa).

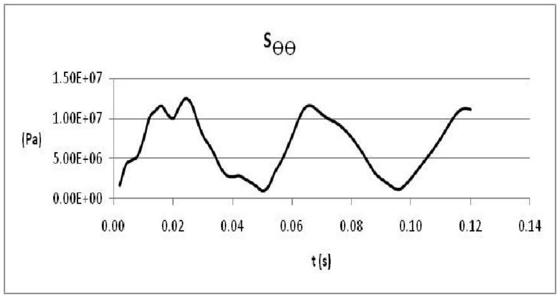

Figure 13 Dynamic Flow Induced Direct Stress Theta Direction (Pa).

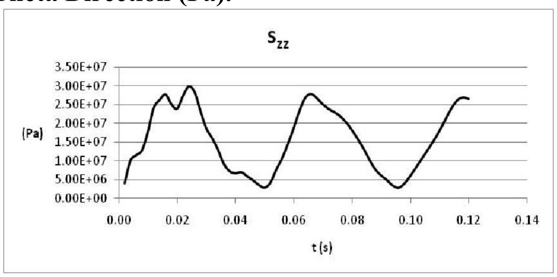

Figure 14 Dynamic Flow Induced Direct Stress In Z Direction (Pa).

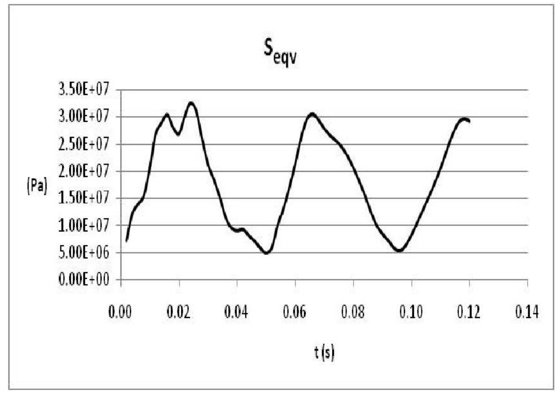

Figure 15 Resultant (equivalent) stress

4 Unit Cell Simulation.

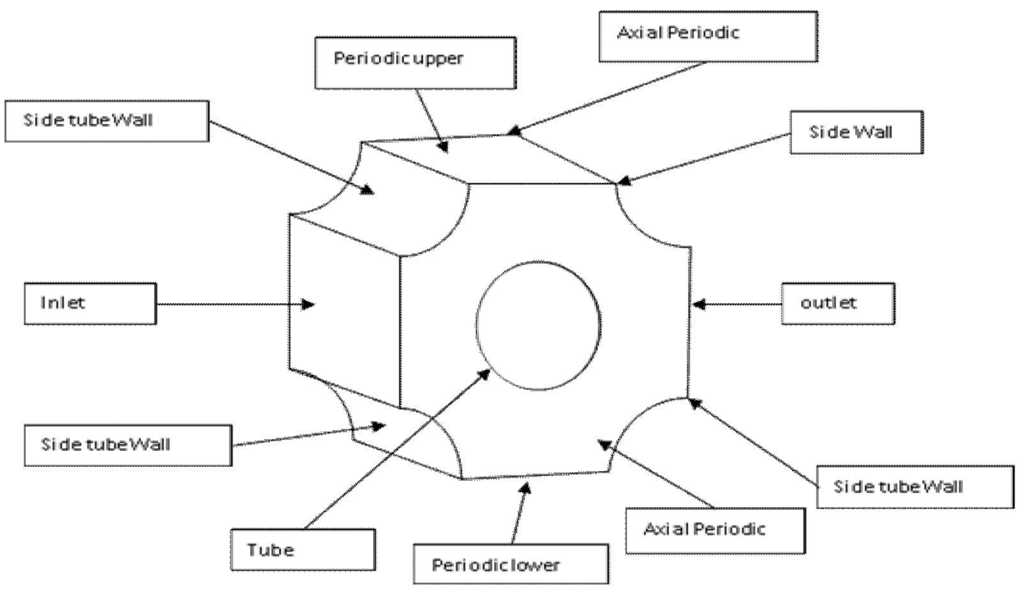

Figure 16 Steam Generator Tube Bundle Unit Cell Representation.

Simonin and Barcouda (1986) and Simonin and Barcouda (1988) employed a unit cell simulation to represent the flow within the bundle as represented by **Error! Reference source not found.**. Similar unit cell was taken for analysis by Karl, et al.(2009) and Kuehlert, Webb and Schowalter (2006). This section follows that technique and a center tube with periodic conditions around the cell is taken. Translational periodicity within CFX is used to represent the bundle. Pairs of domain interfaces are used for periodic conditions. Periodic conditions take advantage of symmetrical geometry and are identified in Figure 16. mass flow rate is specified across the periodic boundaries in x direction i.e the direction of free stream velocity. Pressure level information is supplied through solver options in CFX

4.1 Static Structural

Unit cell simulation representing the cross flow in a three-dimensional she tube type heat exchanger with a triangular pitch pattern with pitch to dia ratio 1.22 is performed. Fluid solid interface is defined at the surface of the tube for the structural side and pressure loads are mapped on to this interface for static structural analysis. A nonlinear static analysis was performed with large deflection on in ANSYS Mechanical. The tube represents an exterior span with fixed-damped conditions. Damped condition at the edge is employed to take into account the damping at the baffle support and as mentioned in (Khusnood 2005) almost 75% of system damping occurs at the supports. Support damping and viscous damping ratios as given in Khusnood (2005) are represented by Eqs (6) and (7) respectively.

$$\zeta_s = \left(\frac{n-1}{n}\right)\left(\frac{2200}{f}\right)\left(\frac{\rho d^2}{m}\right)\left(\frac{L}{L_m}\right)^{1/2} \qquad (6)$$

$$\zeta = \frac{100\pi}{\sqrt{8}}\left(\frac{\rho d^2}{m}\right)\left(\frac{2\upsilon}{(pi)fd^2}\right)^{1/2}\left(\frac{1+\left(\frac{d}{d_e}\right)^3}{\left(1-\left(\frac{d}{d_e}\right)^2\right)^2}\right) \qquad (7)$$

ρ is fluid density, m is mass per unit length of tube which includes hydrodynamic mass and interior liquid mass, de is the equivalent diameter to take into account the confinement due to the surrounding tubes, d is the tube diameter, f is the frequency of tube vibration and ν is liquid kinematic viscosity, L length of tube and Lm is the average of three representative spans. Hydrodynamic mass for a vibrating tube in liquid surrounded by a circular cylinder as mentioned in Khusnood (2005) is given in Eq (8)

$$m_h = C_m m_a \qquad (8)$$

Cm is the inertia coefficient and is the function of geometry and is given by Eq (9),m_a is the added mass and is given by Eq (10),where R is radius of the annulus and r is radius of tube

$$C_m = \frac{R^2 + r^2}{R^2 - r^2} \qquad (9)$$

$$m_a = \rho(pi)r^2 \qquad (10)$$

Added mass is the mass of the fluid displaced by the cylinder and according to Khusnood (2005) it is an effective additional inertia of an accelerating body in a liquid caused by the acceleration of liquid in addition to the structure and its effect is more obvious in liquids than in gasses. The spring stiffness (k_{sp}) and the natural frequency (f_n) of vibrating tube can be related according to (Karl, et al. 2009) by Eq (11)

$$f_n = \left(\frac{1}{2(pi)}\right)\sqrt{\frac{k_{sp}}{m+m_a}} \qquad (11)$$

The damping coefficient c is calculated using the relation of damping ratio ζ as given by Eq (12)

$$c = 2\zeta\sqrt{k_{sp}(m+m_a)} \qquad (12)$$

Khusnood (2005) have formulated the analytical case for fixed damped edges, the natural frequency can be calculated by Eqs (13)

$$kl = \frac{(4n+1)(pi)}{4} \quad (\textit{for fixed damped case})$$

$$k = \sqrt{\frac{\omega}{C}} \quad , \quad C = \sqrt{\frac{EI}{\rho A}} \qquad (13)$$

$$f_n(Hz) = \left(\frac{1}{2(pi)}\right)\left(\frac{(4n+1)(pi)}{4l}\right)\sqrt{\frac{EI}{\rho A}}$$

Khusnood (2005) performed an experiment to measure flow induced vibrations for research type heat exchanger with 15.9mm triangular pitch and 1.221 pitch to dia ratio. An interior and exterior span was taken to take the readings at 26^0C and 33^0C. Pressure was taken from 3.9 to 4.9 bars with Reynolds no between $7.15x10^3$ $4.16x10^5$. The readings of the experiment are shown in

Khusnood (2005) have taken data for an interior span representing a damped-damped case whereas the simulation in CFX is done using a.49m exterior span with fixed-damped conditions. It has been tried to simulate same pressure velocity conditions as given in

The data in Table 2 can be compared to that in Table 3 The trend of pressure, velocity and deflection are same, however deflections cannot be compared due to different boundary conditions employed. The drag direction is taken in the direction of stream.

4.2 Unsteady Structural

A unidirectional unsteady cooped simulation is done within ANSYS environment. Free stream velocity of 4m/s and a pressure of 422 kPa was used. Step size criteria of (1/20fn) corresponding to the fundamental frequency of the span is used for the stability of structural analysis, and a value of.0039 was used to ensure the stability of the numerical computations.

Table 1. Calculations for Mass And Damping Parameter.

CACULATIONS FOR MASS AND DAMPING PARAMTERS			
natural frequency [Hz]eq.(13)	226.0822311	**L1**	
volume of tube[m^3]	9.5211E-06	**L2**	0.23
Modulus of elasticity of tube [n/m^2]	2E+11	**L3**	0.26
mass of tube [kg]	0.076092617	**Lm (average of spans) [m]**	0.326666667
Density of liquid [kg/m3]	992	**Sectional area of tube [m^2]**	1.94308E-05
Density of tube [kg/m3]	7992	**Support damping ratio Eq.(6)**	0.326159473
Added mass [kg/m] eq.(10)	0.131467828	**mass of interior liquid per unit length [kg/m]**	0.112192462
Spring constt eq.(11)	7182.006954	**Damping coefficient from psi (Ns/m) Eq (12)**	25.18576805
Length of span [m]	0.49	**Parameter for hydrodynamic mass Eq (9)**	1.040683499
moment of inertia of tube	3.798E-10	**Hydrodynamic mass)Eq (8)**	0.1368164
outer dia of tube [m]	**0.01299**	**Dia of cylinder**	**0.092**

Table 2. Data from Static FSI Simulation Using CFX.

DATA FROM STATIC FSI SIMULATION (ANSYS CFX).									
No	**Volume Averaged Pressure [Pa]**	**Volume Averaged Velocity U-m/s**	**force on fluid-solid edge (using CFX force function) N**	**Volume Averaged Velocity Uy-m/s**	**Volume Averaged Reynolds no**	**Drag Deflection-mm**	**Lift Deflection-mm**	**Drag Micro Strain [x10^{-6}]**	**Lift Micro Strain [x10^{-6}]**
1	397380	4.10039	111	22.563	387814	1 2.9	-3.3	633	-526
2	426196	3.68154	92.9	20.26	348204	1 1.4	-2.9	559	-466
3	461966	1.8312	34	10.076	173200	6 .49	-2.49	226	-249
4	490010	0.074967	.264	0.412	7090.69	. 24	.366	-46	-36

Table 3 Data From Experiment Conducted by (Khusnood 2005).

Natural Frequency (f_n=120.7Hz)   Pump supply Head = 0.76 m (Positive Suction)   Tube Span = 0.49m (Interior)   Temperature (33^{0}C)

Test No.	P (bars)	Q(m^3/hr)	U(m/sec)	U_p (m/sec)	M	S	V_r	R_e	y(mm)	$\dot{y}$ (m/sec)	$\ddot{y}$ (m/sec^2)	Force (N)	Wear Work (watt)	Vol. Loss (mm^3/year)
48	3.9	16.6	4.98	27.5	0.0866	0.057	17.52	4.105x10^5	7.4825	0.4701	29.539	2015	946	42625
49	4.0	16.17	4.739	26.16	0.0176	0.060	16.67	3.8x10^5	7.3995	0.4649	29.2119	1856	863	38146
50	4.2	14.3	4.29	23.68	0.0160	0.066	15.1	3.44x10^5	5.4414	0.3419	21.4816	1521	520	22984
51	4.3	12.6	3.75	20.7	0.0140	0.0757	13.2	3.00x10^5	4.4506	0.2796	17.5703	1162	325	14365
52	4.4	11.06	3.22	17.77	0.0120	0.0883	11.32	2.58x10^5	3.5605	0.2237	14.0565	856.6	192	8470
53	4.5	9.38	2.76	15.2	0.0103	0.103	9.68	2.2x10^5	2.4489	0.1539	9.6679	627	97	4263
54	4.6	7.16	2.133	11.77	0.0080	0.1333	7.5	1.71x10^5	1.7893	0.1124	7.0638	376	42	1870
55	4.7	4.55	1.319	7.28	0.00494	0.2155	4.64	1.06x10^5	1.7606	0.1106	6.6505	144	16	703
56	4.8	0.7379	0.223	1.23	0.00083	1.27	0.784	1.79x10^4	0.7925	0.0498	3.1286	4.1	0.204	9
57	4.9	0.29	0.089	0.49	0.00043	3.20	0.312	7.15x10^3	0.8942	0.0562	3.5300	0.65	0.0366	1.62
													Average Vol. Loss	13344

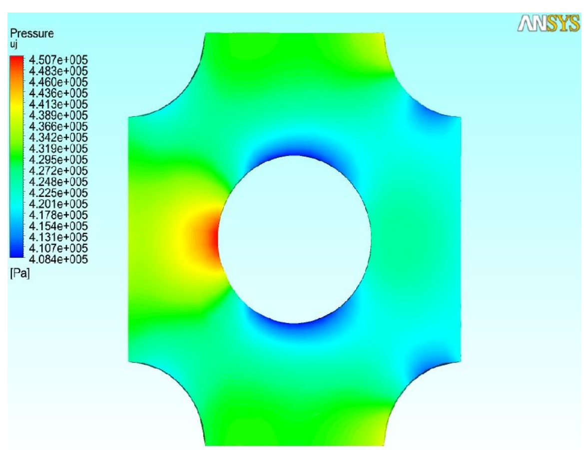

Figure 17 Pressure Profile In A Unit Cell.

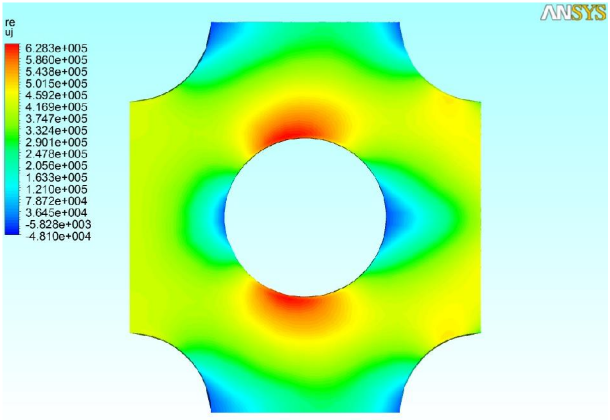

Figure 18 Reynold's Number Profile In Unit Cell.

Figure 17 and Figure 18 shows the pressure profile and Reynolds number profile within the cell.

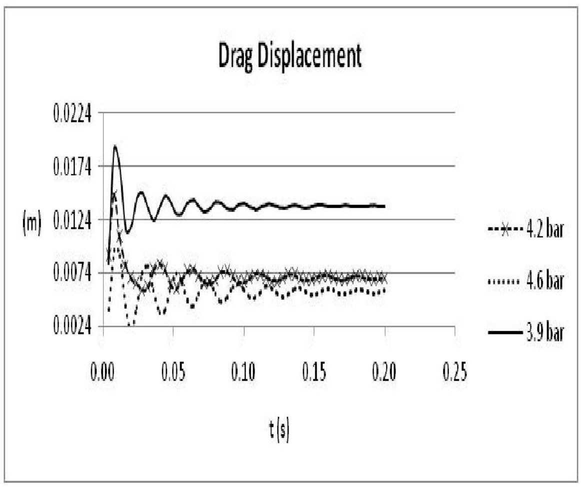

Figure 19 Drag Displacement For One Way Coupled FSI.

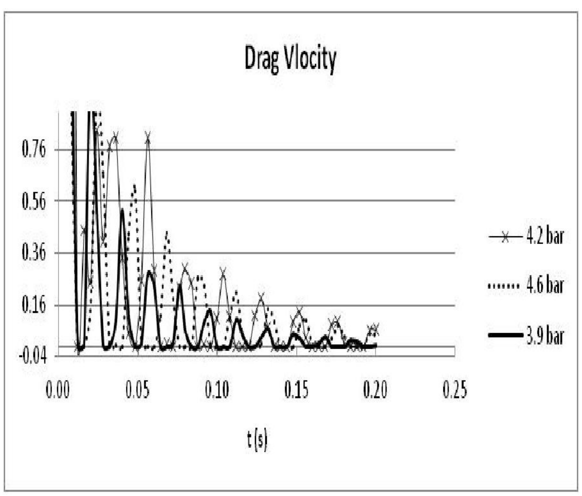

Figure 20 Velocity For One Way Coupled FSI.

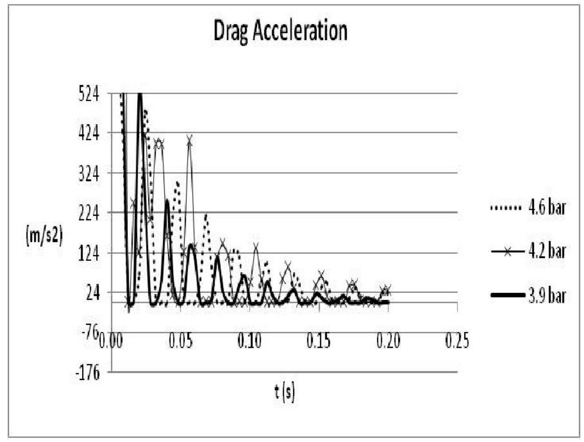

Figure 21 Drag Acceleration For One Way Coupled FSI.

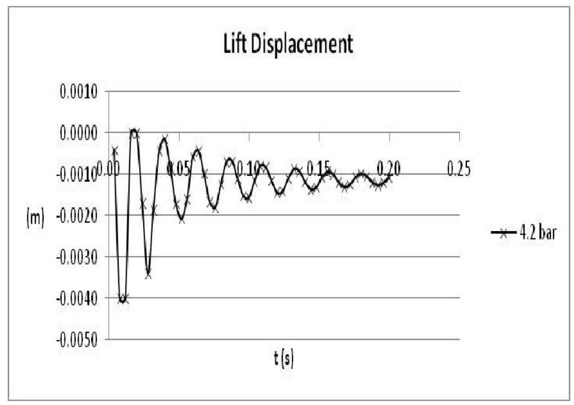

Figure 22 Lift Deflection For One Way Coupled FSI.

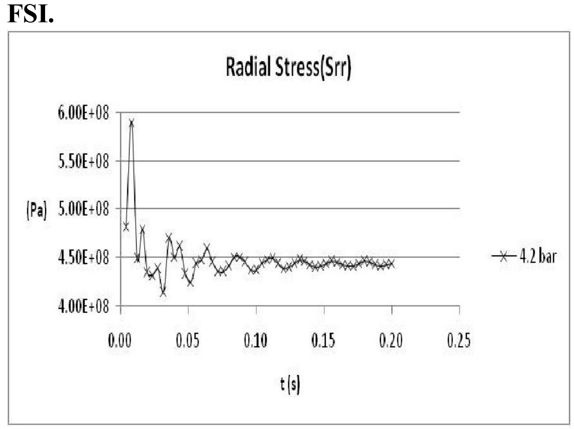

Figure 23 Radial Stress For One Way Coupled FSI.

Figure 19 shows the drag deflection at a point of the fixed damped span with data obtained for three different pressures. The damping effect is seen in the plot and a general trend of increased deflection with decrease in pressure is also visible. Figure 20 shows the plot of velocity for three pressure settings, while Figure 21 shows the corresponding acceleration at three pressures for the given location. Figure 22 shows the lft deflection at 4.2 bars. Figure 23, Figure 24 and Figure 25 gives radial, circumferential and longitudinal flow induced stresses at a point in the span respectively and Figure 26 shows the equivalent stress in the span which can be used as a criteria for design purposes and the corresponding strain as shown in Figure 27.

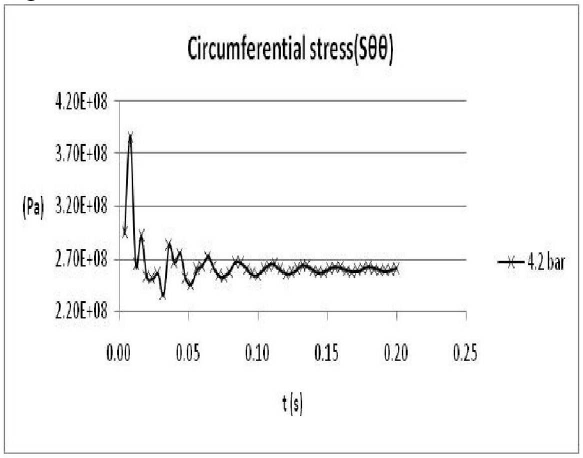

Figure 24 Circumferential Stress For One Way Coupled FSI.

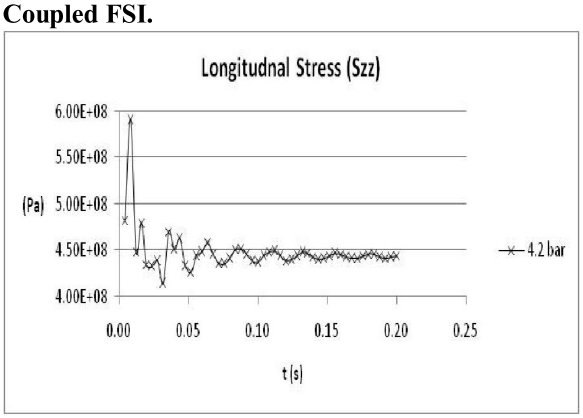

Figure 25 Longitudinal Stress For One Way Coupled FSI.

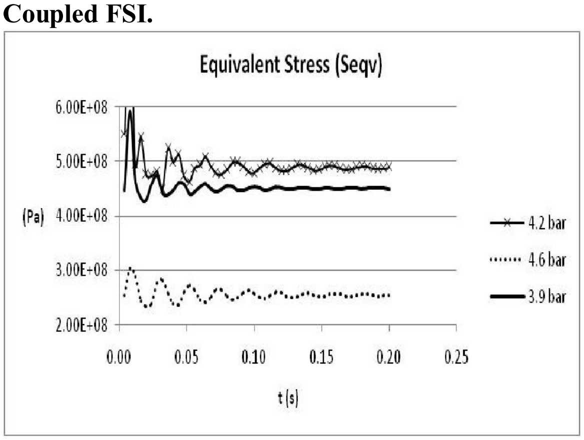

Figure 26 Equivalent Stress For One Way Coupled FSI.

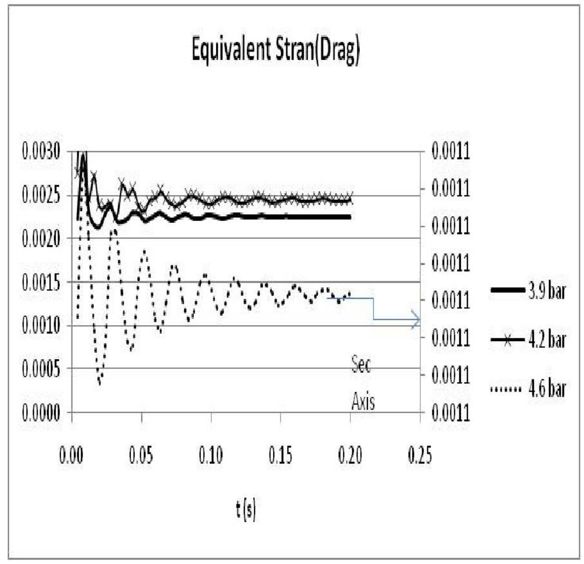

Figure 27 Equivalent Strain Drag Direction For One Way Coupled FSI.

5 Conclusion

This study is an attempt to simulate steady and unsteady FSI analysis using commercially available ANSYS code using the latest ANSYS workbench. The data obtained is satisfactory. Single way coupled unsteady analysis is done in this work as the displacements were not large enough to considerably affect the flow field. The strength analysis including cyclic stresses and fretting wear of tubes can be efficiently assessed using this approach. This can be extended to two way coupled analysis including mesh movement to see the effects of fluid forces as a consequence of moving mesh. Temperature can also be included to see the effects of temperature induced as well as flow induced stresses.

Nomenclature

Symbol	Description
Srr	Radial stress (direct)
Sθθ	Circumferential stress (direct).
Szz	Longitudinal stress (direct)
f_n	Natural Frequency of tbe.
c	Damping coefficient.
C	Parameter defined by (13)
k_{sp}	Stiffness of tube.
Pi	Greek letter

Corresponding Author:
Abu Bakar Izhar
Departement of mechanical engineering.
University of Engineering and Technology, Lahore.
E-mail: bakarizr@gmail.com

References

1. "ANSYS CFX Rel 12.0 Documentation." PA,US: ANSYS, Incorporated, 2012.
2. Chen, SS. "Fluidelastic instabilities in tube bundles exposed to no uniform cross-flow." journal of Fluids and structures. 1991:5: 299-322.
3. Chen, SS. "Instability mechanism and stability criteria of a group of circular cylinders subjected to cross flow." journal of viration and acoustics. 1983:105: 51-59.
4. Eisinger, Rao, Steininger, and Haslinger. "numerical simulation of cross flw induced fluidelastic vibration of tube arrays and comparison with experimetal results." 1995:117 : 31-39.
5. Hossin. "Numerical simulations of fluidelastic instability in tube bundles." PhD Thesis, Dept. of Mechanical engineering, University of New Brunswick, Canada, 2010.
6. Hossin, and Hassan. "Numerical Simulations of Unsteady Fluid Forces in Heat Exchager Tube Bundles." In Computational Simulations and Applications. intech.2011:307-430.
7. Hover, FS, and AH Techet. "Forces on oscillating uniform and tapered cylinders in crossflow." of Fluid Mechanics. 1998:363 : 97-114.
8. Karl, Keuhlert, Webb Stephen, Schowalter David, Holmsea William, and Reuss Steve. "Simulation Of The Fluid–Structure-Interaction Of Steam generator Tubes and Bluff Bodies." Engineering and Design. 2009:238: 2048-2054.
9. Kern, Donald Q. Process Heat Transfer. 23rd Reprint. 2011.
10. Khusnood, Shahab. Vbration Analysis of a Multi-Span Tube in a Bundle. PhD Thesis, Mechanical Engineering Department, College of Electrical and Mechanical Enigineering, Rawalpindi, Pakistan: National University of Sciences and Technology, 2005.
11. Kim, and Mohan. "Prediction of unsteady loading on a circular cylinder in high Reynolds number flows using Large Eddy Simulation." Proceedings of OMAE,24th International Conference on Offshore Mechanics and Arctic Engineering. Halkidiki, Greece, 2005.
12. Kuehlert, Karl, Stephen Webb, and David Schowalter. "Fluid structure interaction of a steam generator tube in a corss flow using Large eddy simulation." Florida,US: Proceedings of ICONE14, 2006.
13. Oakley, Constsantinides, and Holmes. "Modelling vortex induced motions of spars in uniform and stratified flws." Proceedings of OMAE, 24th International Conference on Offshore Mechanics and Arctic Engineering. Halkidiki, Greece, 2005.
14. Simonin, and Barcouda. "Measurement of fully developed turbulent flow across tube bundle." Proceedings of 3rd International Symposium of applications of anemometry to fluid mechanics, Lisbon, Portugal. Lisbon,Portugal, 1986.
15. Simonin, and Barcouda. "Measurements and prediction of turbulent flw entering a staggered tube bundle." Proceedings of 4th International Symposium of applications of anemometry to fluid mechanics. Lisbon,Portugal, 1988.
16. Tanaka, H, and H Takhara. "Fluid elastic vibration of tube array in cross flow." journal of soud and vbration. 1981: 77: 19-37.
17. Zohir, A.E."The Influence of Pulsation on Heat Transfer in a Heat Exchanger for Parallel and Counter Water Flows", New York Science Journal, 2011;4(6):61-71.
18. Connors, 1970. An experimental investigation of the flow-induced vibration of tube arrays in cross flow. University of Pittsburgh.
19. Chen, S.S. A general theory for dynamic instability of tube arrays in crossflow. Journal of Fluids and Structures. 1987:1: 35–53.
20. Chen, S.S, J.A,. Experiments on fluid elastic instability in tube banks subjected to liquid cross flow. Journal of and V bration. 1981: 78: 355–381.
21. Piperno, S. Explicit/implicit fluid/structure staggered procedures with a structural predictor and fluid subcycling for 2d inviscid aeroelastic simulations. International Journal for Numerical Methods in Fluids. 1997: 25: 1207–1226.
22. Pettigrew, M.J, Taylor, C.E. Viration analysis of shell-and-tube heat exchangers: an overview—Part 1: flow, damping, fluidelastic instability. Journal of Fluids and Structures. 2003 :18, 469–483.
23. Gillen, S, Meskell, C. Numerical Analysis of Fluidelastic Instability in a Normal Triangular Tube Array. {ASME} Conference Proceedings. 2009: 447–455.

2/28/2013