Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 4, 2021, Pages. 12009 - 12014
Received 05 March 2021; Accepted 01 April 2021.

CFD Simulation and Analysis of Fluid Induced Vibration in Pipe System

Sandeep Bandarwadkar[1*], Sandesh Mysore Sathyaraj[2], Mrunal Naidu Uppu[3]& Swaroop Kumar Muthyala[4]

[1*]Department of Civil Engineering and Architecture, Kaunas University of Technology, Lithuania.
[2]Department of Electronics and Electrical Engineering, Kaunas University of Technology, Lithuania.
[3]Department of Vehicle Engineering, Kaunas University of Technology, Lithuania.
[4]Department of Materials Physics, Kaunas University of Technology, Lithuania.

Abstract
The current research deals with vibration induced due to flow of fluid in different sections such as circular and square. The vibration is due to turbulence in the flow process of fluid. This occurs as the major flow follows discontinuities such as irregular bending, tees, incomplete close of valves and small-bore connections. The variation of vibration significantly depends on designing pipe sections, stiffness of supporting configuration and way of operating valves. Designing of pipe sections using ANSYS SPACECLAIM in which three types of sections are designed such as straight rectangular section, rectangular section bent with sharp outer corner and rectangular section bent with round corners. All these sections induce different levels of vibration depending on downstream kinetic energy at low frequency. Then computational fluid dynamics simulation, FLUENT launcher is used for simulation with specific dimensions of geometry which consist of both solid and fluid regions. Boundary conditions are updated with low frequency as 20Hz.The analysis and post-processing of pipe system, the inlet valve velocity is given as 5 m/s and outlet valve velocity is retained with default velocity. The SST k-omega turbulence model is selected and analysed with boundary condition. Finally, post processing two contours are created representing static pressure and velocity magnitude.
Keywords: CFD; Ansys software; Fluid induced vibration; Pipe system.

1. Introduction

Today, in fact, the complex behaviour of time-dependent flows in pipe systems is used to boost the energy efficiency of internal combustion engines [1]. It is therefore important to consider the dynamic behaviour of fluid flow inside the pipe under unsteady boundary conditions. Many problems are found in conjunction with the ICE exhaust system. Example, obtained analytical results on the problem by developing a one-dimensional approach for modelling the steady flow of fluid in shock pipes [2][3]. Several other attempts at computational simulation of unsteady pipe flow have been presented in earlier studies [3]. The findings of a comprehensive experimental analysis of the complex activity of pulsating currents in the pipe were discussed by a group of researchers who conducted noise measurements along a straight pipe [4]. Analogous to mechanical vibrations, dynamic parameters for the hydraulic system under consideration, such as self-frequency, damping, etc., may also be calculated [5]. From a theoretical point of view, these criteria are not well

Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 4, 2021, Pages. 12009 - 12014
Received 05 March 2021; Accepted 01 April 2021.

researched owing to the complicated definition of the time-dependent, turbulent flow of the viscous, compressible fluid, as evidenced by an analysis of the related literature [6][7].

2. Materials and Methods

2.1 System Circuit Duct

The flow induced vibration is considered as multi-physical problem. This problem is solved in a virtual environment using Ansys. This problem has an ability to interconnect individual engineering areas like flow and structural engineering. The whole geometry is divided into three parts [8]. The length of first part is 7m. It begins with rectangular section and ends with sharp bent rectangular section. The length of second part is 4m. It begins with sharp bent rectangular section and ends with round corner rectangular section. The length of third part is 3m [9]. It begins with round corner rectangular section and ends with plane rectangular section. The figure below shows the flow of fluid through different section of complete system with inlet velocity as 5m/s.

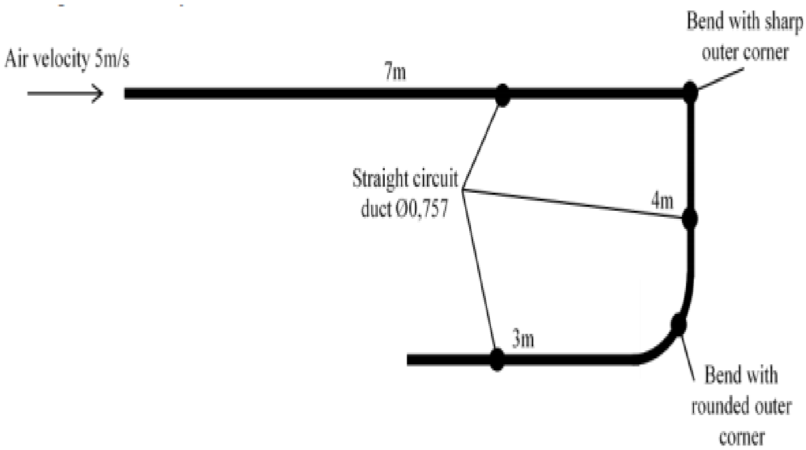

Figure 1 System Circuit Duct

Part	Width (mm)	Height (mm)	Thickness (mm)	Collar thickness (mm)	Angle (degree)
1	300	150	1	1	0
2	300	150	1	1	90
3	300	150	1	1	90

Table 1dimensions of steel chimney including reduced thickness

As said above, the complete circuit duct has three parts. All the parts are made up of galvanized sheet. The corner rectangular section is provided with collar thickness as 1mm.

2.2 CFD Simulation

The CFD simulation is explained as Activate Watertight Geometry Workflow as shown below.The Local Sizing input as "no" and Create Surface Mesh. In surface meshing, the Minimumand Maximum Size should be mentioned as 20 and 200 mm respectively. The curvature and proximity settings for sizing functions left as default [10]. As, Geometry consists of both solid and fluid regions, the cap openings and extract fluid regions are

Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 4, 2021, Pages. 12009 - 12014
Received 05 March 2021; Accepted 01 April 2021.

nil.Boundary change "Select By" option to "label", Select “velocity” for inlet and “pressure” for outlet. Now, the regions “estimated number of fluid region as one”. The fluid region type is “air volume” and solid region type to part 3.2 solid. The volume mesh simulated as follows:

a. Select offset method type as “smooth transition”.
b. Give number of layers as 3.
c. Select transition as 0.272.
d. Select growth rate as 1.2.
e. Fill it with “tetrahedral meshing”.

After CFD, the circuit duct is processed with volume meshing and it is shown below.

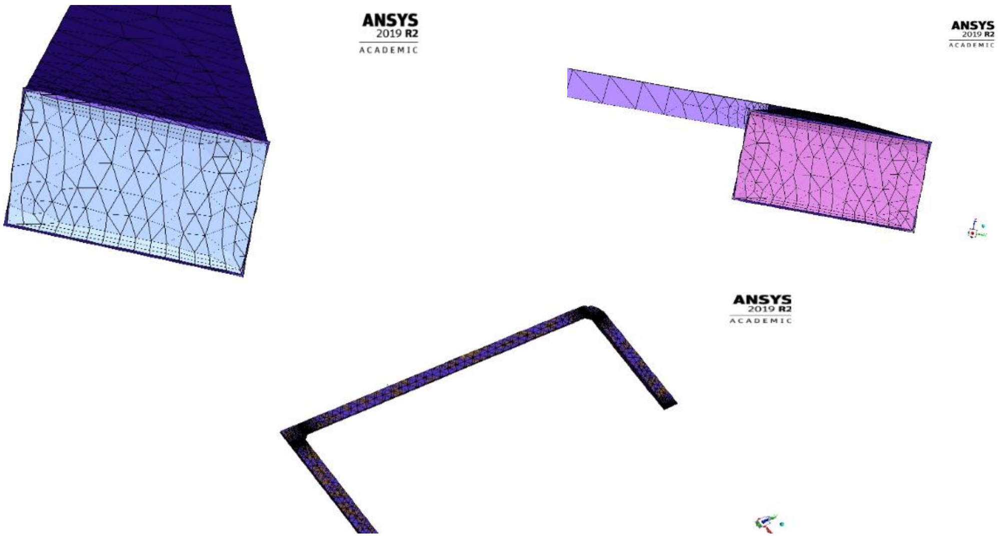

Figure 1Inflow, outflow and complete circuit duct

3. Analysis and Results

The analysis and post-processing are carried out as follows: Setting up physics, Viscous Model: In the Setting Up Physics tab, click Viscous and activate the SST k-omega turbulence model and Boundary conditions: Set inlet velocity to 7.5 m/s and retain default values for outlet. Then solving involves Pressure-Velocity Coupling scheme to Coupled, Pseudo Transient near the bottom of panel [12][13]. For most incompressible flows, these settings offer superior convergence to default settings [11].

To complete the final analysis, define surface report to plot the area weighted average of static pressure on the inlet. The inlet pressure is the quantity of interest for this simulation and the report plot will help to ensure that its converged value has been established. To generate results, set the number of iterations to 200 and Residuals decrease monotonically until convergence criteria are achieved. Plot of inlet pressure shows that final value has been attained.

Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 4, 2021, Pages. 12009 - 12014
Received 05 March 2021; Accepted 01 April 2021.

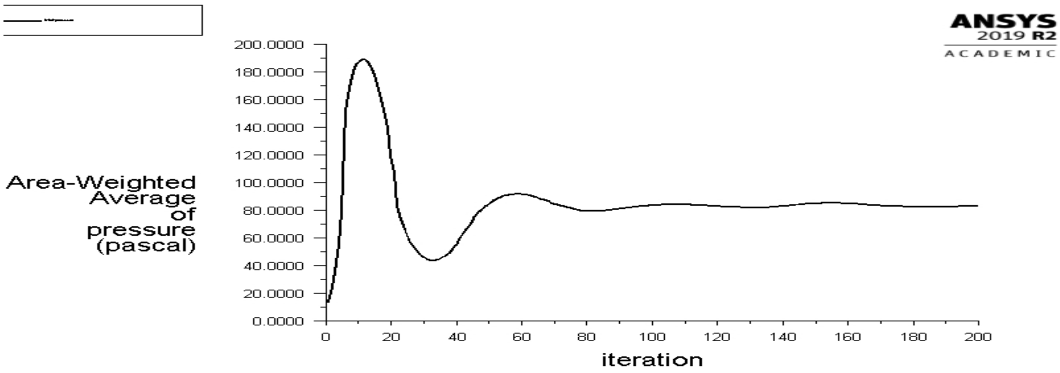

Figure 2Distribution of Pressure

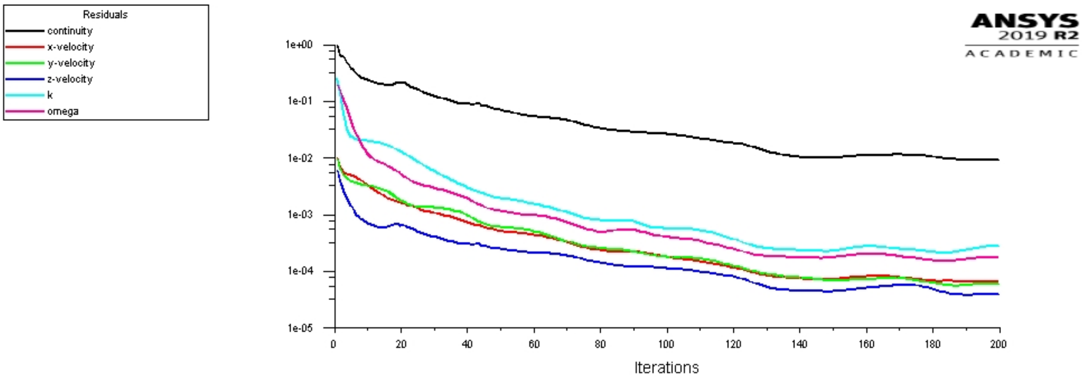

Figure 3Scaled Residuals

The above graph shows the variation of residual velocity throughout the length of circuit duct, calculated for different iterations. Once the variation of pressure and velocity is observed, the problem is carried with post-processing. The fluxes, in the Post processing tab and perform a mass balance [14]. The net imbalance is several orders of magnitude less than the inlet flow rate, indicating excellent mass conservation has been achieved in the numerical solution.

a. First combination.
   i. Mass Flow Rate (kg/s)
      Inlet: 0.26958688
      Net: 0.26958688
b. Second combination.
   ii. Mass Flow Rate (kg/s)
      Inlet: 0.26958688
      Outlet: 0.26944361
      Net: 0.00014327551
c. Third combination.
   iii. Mass Flow Rate (kg/s)
      Inlet: 0.26958688
      Interior--air-volume: 2.6410467
      Outlet: 0.26944361
      Net: 0.00014327551
d. Total heat transfer.
   iv. inlet: 0

Annals of R.S.C.B., ISSN:1583-6258, Vol. 25, Issue 4, 2021, Pages. 12009 - 12014
Received 05 March 2021; Accepted 01 April 2021.

Outlet: 0
Net: 0

e. Radiation heat transfer.

v. inlet: 0
Outlet: 0
Net: 0

Create an iso-surface of Z-coordinate = 0 to display the solution on a plane through the centre of the flow passage. Create contours of static pressure and velocity magnitude on the z=0 iso-surface. Contour representing static pressure.

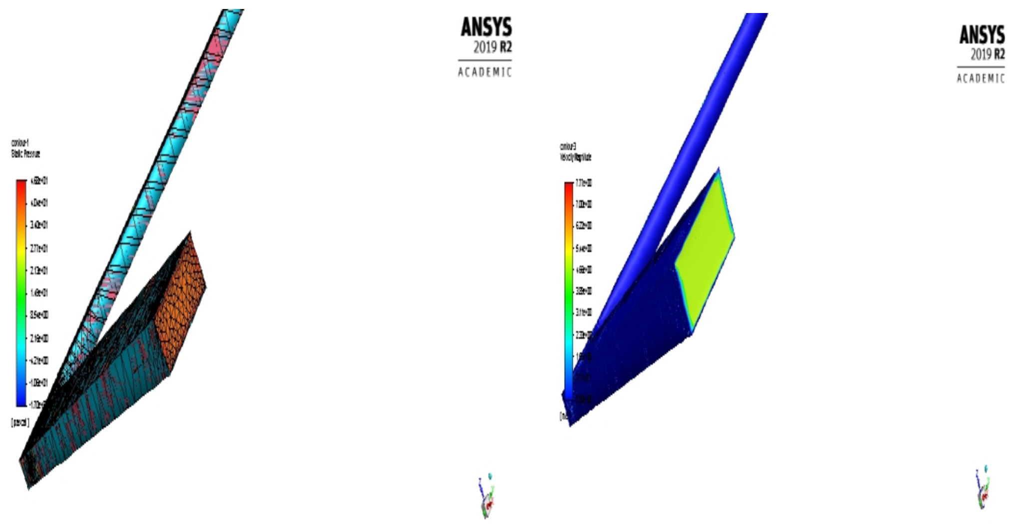

Figure 4 Static Pressure and Velocity Magnitude at Z=0

Finally, the path line was created as shown below.

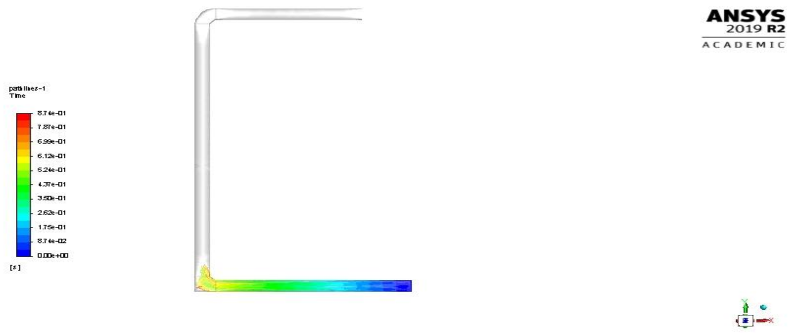

Figure 5Path line fluid flowing through circuit duct

4. Conclusions

A high quality CFD volume mesh was created with tetrahedral cells and boundary layers. Persistent graphics objects and scene displays were created in post-processing which shows variation of static pressure and velocity magnitude. The flow induced vibration Geometry workflow in Meshing mode enabled CAD import, volume extraction and meshing through a sequence of pre-defined tasks with simple and intuitive user inputs. From the above calculation it is clear that the net imbalance is several orders of magnitude less than the inlet flow rate, indicating excellent mass conservation has been achieved in the numerical solution. From the simulation it is evident that Total heat transfer and Radiation heat transfer from the inflow and outflow is zero. Hence, the possibility of inducing vibration is very less. To conclude, the pipe flow can be adopted in practice after considering various factors.

References

1. S.S. Chen 1987, A General Theory for Dynamic Instability of Tube Arrays in Crossflow, Journal of Fluids and Structures 1, 35-53.
2. H. Tanaka, S. Takahara 1981, Fluid Elastic Virbration of Tube Array in Cross Flow, Journal of Sound and Vibration 77, 19-37.
3. C.W. Hirt, A.A Amsden, J.L. Cook 1974, An Arbitrary Lagrangian-Eulerian Computing Method for all Flow Speeds, Journal of Computing Physics 14, 227 - 253.
4. "CFD-TASCflow Theory, V2.12.1", ANSYS Canada, 2002.
5. C.H.K. Williamson 1991, 2-D and 3-D Aspects of the Wake of a Cylinder, and their Relation to Wake Computations, Lectures of Applied Mathematics 28, 719 - 751. Providence, RI: American Mathematical Society.
6. P.K. Stansby, A. Slaouti 1993, Simulation of Vortex Shedding Including Blockage by the Random-Vortex and Other Methods, International Journal for Numerical Methods in Fluids 17, 1003-1013.
7. C. Norberg 2001, Flow Around a Circular Cylinder: Aspects of Fluctuating Lift, Journal of Fluids and Structures 15, 459-469.
8. Corcos, G. M. Resolution of Pressure in Turbulence, The Journal of the Acoustical Society of America, 35 (2), 192–199, (1963).
9. Hambric, S.A., Hwang, Y.F. and Bonness, W.K. Vibrations of plates with clamped and free edges excited by low-speed turbulent boundary layer flow, Journal of Fluids and Structures, 19 (1), 93–110, (2004).
10. Azimi, S., Hamilton, J.F. and Soedel,W. The receptance method applied to the free vibration of continuous rectangular plates, Journal of Sound and Vibration, 93 (1), 9–29, (1984).
11. David, A., Hugues, F., Perrey-Debain, E. and Dauchez, N. Vibrational response of a rectangular duct of finite length excited by a turbulent internal flow, submitted to Journal of Sound and Vibration.
12. Lecoq, D., Pézerat, C., Thomas, J.-H. and Bi,W.P. Extraction of the acoustic component of a turbulent flow exciting a plate by inverting the vibration problem, Journal of Sound and Vibration, 333 (12), 2505–2519, (2014).
13. Hwang, S.T and Moon, Y.J. Turbulent boundary layer noise in pipe flow, Internoise, Hamburg, (2016).
14. Trabelsi, H., Zerbib, N., Ville, J.-M. and Foucart, F. Passive and active acoustic properties of a diaphragmat low Mach number, European Journal of Computational Mechanics, 20, 49–71, (2011).