Iran University of Science and Technology

International Journal of Automotive Engineering

Journal Homepage: ijae.iust.ac.ir

ISSN: 2008-9899

Determining an efficient numerical solution method for pressure loss problem in bends

Amirhasan Kakaee[1*], Milad Mahjoorghani[2]

[1]Associate Professor, Faculty of Automotive Engineering Department, Iran University of Science and Technology

[2]MSc Graduate, Automotive Engineering Department, Iran University of Science and Technology

ARTICLE INFO

Article history:

Received : 8 March 2019

Accepted:30 May 2019

Published:1 June 2020

Keywords:

Intake Manifold

Pressure Loss Coefficient

Miller's Test

3D Simulation

Turbulence Models

A B S T R A C T

Intake and exhaust manifolds are among the most important parts in engine in which pressure loss phenomena has direct impact on with changing volumetric efficiency. In typical 1D simulation codes, the quantity of pressure loss is proportional to the fluid's mean velocity by Pressure Loss Coefficient (K_p) value. This important coefficient which has substantial rule in engine simulation is usually determined using constant available values, extracted from complicated experiments (like Miller's tests) in a specified situation. But these values are credible only in situations according to those tests. Coupling 3D simulations with 1D codes is a common method to gain accurate values of these coefficients but this deals with drastic high simulation costs. To address this problem, a more efficient way is replacing an algebraic relation, extracted from 3D calculations, instead of a constant value in 1D code. It's obvious that in order to reach accurate coefficients in arbitrary conditions (geometric and flow specifications) determining the best numerical method is mandatory. In present research, after investigating all 3D simulation aspects, six different selected numerical solutions have been implemented on four different bends in ANSYS Fluent. Results have been validated by comparing loss coefficient values of incompressible fluid (water) with Miller loss coefficient values and method with the most accurate and stable results has been discovered. It was found that all these methods are suitable in general (with less than 5% error in coefficient values) but solutions with structured grid and SST k-ω turbulence modeling represented better stability and accuracy. Changing discretization or velocity-pressure coupling method was not that effective but the problem showed impressive sensitivity to grid structure type and turbulence modeling methods.

[DOI: 10.22068/ase.2019.494]

1. Introduction

Intake manifold system is in fact an assembly of pipes with different dimensions that are conjunct together with various connections. As a basic principal in design of these systems, geometric and layout limits have led to utilizations of bend pipes. These bends create more complexity in different aspects of fluid's behavior like pressure losses and velocity profile.

One of the most important characteristics related with these aspects in 1D modeling of engine is Pressure Loss Coefficient (K_p). In governing equations of these 1D models, bends pressure losses are proportional to these values which are considered constant experimental

https://doi.org/10.22068/ase.2019.494

values typically between 0.3 to 0.9 depending on geometric characteristics in lots of these pipes and connections by default[1]. On the other hand in an approach wider than 1D modeling, determining exact value of this coefficient requires a clear explanation of fluid's complicated behavior in intake system and it's a very solid problem even in labs. Two main reasons of this complexity are interaction between expansion and contraction waves and complicated geometry[2].

Nonetheless it's obvious that this coefficient's value depends on flow regime and fluid compressibility too and developing a general useful correlation for calculating it in terms of bend geometry and flow characteristics is a brilliant idea to give more flexibility and accuracy to our 1D modeling. The main purpose of present work is to determine the best numerical solution method for such problems by conducting several 3D simulations on various bends with geometries analogous to the main bend in TU5 engine's intake system and comparing results with experimental constant values.

Dean with a large number of experiments about laminar and turbulent flows inside bends[3] and Womersley with restructuring these tests by taking into account the fluctuation of fluid[4] are most notable pioneers in investigating this problem. Some decades after them, Miller conducted several tests about flow inside different geometries like elbows and T-junctions to calculate loss coefficient values in a specific flow condition (Re = 10^6) for water and presented them in a format known as Miller Charts[5]. Nowadays several 1D engine simulation codes like GT-Power use these charts to state geometry effects on pressure losses, however constraint of utilizing these values against alteration of flow characteristics (like Reynold number or compressibility of fluid) remains an interesting research gap.

Abou-Haidar and Dixon targeted this gap and after investigating the effect of fluid compressibility in T-junctions, suggested the angle between branches as a new variable and reformed Miller charts for this specific element[6]. Maharudrayya et al. changed flow's Reynolds number during their tests and extracted a useful correlation in terms of Reynolds number and bend geometric characteristics to calculate pressure loss coefficient in fuel cell stack[7]. Kumar implemented the same innovation to determine best numerical solution method and grid type and calculated discharge coefficient in different joints with different flows[8].

With a similar approach, Desantes et al. conducted several experiments with different diesel fuel injectors and calculated loss coefficient in new manner so they proved that their new correlations are accurate and reliable[9]. Nimadge et al. also simulated the flow inside T-junctions and after comparing results with experiments, they asserted that a flexible approach to present pressure loss coefficient values in general is the key to gain access to a comprehensive statement about fluid's behavior[10].

2. Approach

Contrary to widespread usage of classic experimental results in concepts like pressure loss and their appreciable leading impact on subsequent researches, since these results have been attained in a specific lab conditions their validity in unstable transitory conditions like engine and intake manifold flow problems is unreliable. Running 3D simulations to study specifications effects on available results is a common approach that first of all needs a validated practical numerical solution method in considered problem.

2.1. Pressure loss coefficient in 1D simulations

In famous one-dimensional engine simulation codes like GT-POWER the flow model for flow in pipes and connections involves the solution of Navier-Stokes equations, namely the conservation of continuity, momentum and energy equations. The momentum equation is our point of interest here:

$$\frac{d(\dot{m})}{dt} = \frac{dpA + \sum_{boundaries}(\dot{m}u) - 4C_f\frac{\rho u|u|}{2}\frac{dxA}{D} - K_p(\frac{1}{2}\rho u|u|)A}{dx} \quad (1)$$

where $\dot{m}$ is boundary mass flux into volumes, p is pressure, A is cross-sectional flow area, u is velocity at the boundary, C_f is Fanning friction factor, D is equivalent diameter, K_p is pressure loss coefficient (commonly due to bend, taper or restriction) and ρ is density.

In bends, the total pressure loss coefficient K_{total} is the sum of both the loss due to the bend, K_p and the loss due to friction, K_f defined as:

$$K_{total} = \frac{\Delta P}{\frac{1}{2}\rho u^2} = K_p + K_f \quad (2)$$

where ΔP is the total pressure drop across the bent pipe and u represents the velocity of the fluid at the inlet to the bend. This coefficient is approximated with a curve-fit to the Miller chart as shown in next page. The chart shows the total loss coefficient for a circular, smooth pipe at Re = 10^6 as a function of R/D and bend angle[1].

[DOI: 10.22068/ase.2019.494]

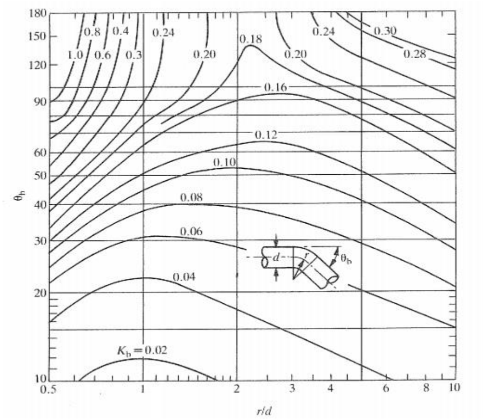

Figure 1: Miller chart – Loss coefficient for circular cross-section bends (Re = 10^6) [5]

2.2. Governing Equations

2.2.1. Basic equation of fluid flow

Theoretically, in steady and turbulent flow, the continuity and momentum equations for incompressible, time-dependent and viscous fluid are represented as follows:

$$\frac{\partial \rho_f}{\partial t}+\frac{\partial(\rho_f v_x)}{\partial x}+\frac{\partial(\rho_f v_y)}{\partial y}+\frac{\partial(\rho_f v_z)}{\partial z}=0 \quad (3)$$

where v_x, v_y, v_z are velocity vectors in the x, y, z directions, respectively and ρ_f the fluid density.

$$\frac{\partial(\rho_f v_i)}{\partial t}+v_i\left(\frac{\partial(\rho_f v_x)}{\partial x}+\frac{\partial(\rho_f v_y)}{\partial y}+\frac{\partial(\rho_f v_z)}{\partial z}\right)= \rho_f g_i-\frac{\partial P}{\partial i}+R_i+\frac{\partial}{\partial x}\left(\mu_c\frac{\partial v_i}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_c\frac{\partial v_i}{\partial y}\right)+ \frac{\partial}{\partial z}\left(\mu_c\frac{\partial v_i}{\partial z}\right)+T_i \quad (4)$$

where i present x, y, z directions, respectively, g_i the accelerations due to gravity, P the fluid pressure, μ_c the effective viscosity, R_i the distributed resistance, and T_i the viscous loss terms.

2.2.2. Turbulence modeling

One of the main considerations in a reliable valid numerical solution method for fluid flow is to find the best turbulence model. As a result, in present work 3 different turbulence models have been examined in terms of stability, accuracy and convergence. All these 3 models are RANS (Reynolds Average Navier-Stokes) models that offer the most economic approach for computing complex turbulent industrial flows with decomposing governing equations into mean and fluctuating components. Typical examples of such models that have been used in this work are the $k-\varepsilon$ and the $k-\omega$ models. These models simplify the problem to the solution of two additional transport equations and introduce an eddy-viscosity (turbulent viscosity) to compute the Reynolds Stresses. First one is Standard $k-\varepsilon$ model which is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). These variables are obtained from following transport equations:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+ G_b-\rho\varepsilon-Y_M+S_k \quad (5)$$

and

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+ C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \quad (6)$$

In these equations, G_k represent the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms. The turbulent (or eddy) viscosity, μ_t , is computed by combining k and ε as follows:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \quad (7)$$

Realizable $k-\varepsilon$ contains an alternative formulation for the turbulent viscosity and a modified transport equation for the dissipation rate ε, has been derived as follows:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+ \rho C_1 S\varepsilon-\rho C_2\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}}+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b+S_\varepsilon \quad (8)$$

where

$$C_1=max\left[0.43,\frac{\eta}{\eta+5}\right], \eta=S\frac{k}{\varepsilon}, S=\sqrt{2S_{ij}S_{ij}} \quad (9)$$

The eddy viscosity is computed from same equation as (7) but with different constants.

SST $k-\omega$ model is another choice which has been tested in this research. It includes all the refinements of the BSL $k-\omega$ model and in addition accounts for the transport of the turbulence shear stress in the definition of the turbulent viscosity. Generally $k-\omega$ models define the specific dissipation rate ω (as the ration of ε to k and obtain it from the following transport equation:

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial}{\partial x_i}(\rho\omega u_i)=\frac{\partial}{\partial x_j}\left[\Gamma_\omega\frac{\partial \omega}{\partial x_j}\right]+G_\omega-Y_\omega+ S_\omega \quad (10)$$

[DOI: 10.22068/ase.2019.494]

In this equation Γ_ω represent the effective diffusivity of ω which is given by:

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_k} \tag{11}$$

To obtain eddy-viscosity in SST $k-\omega$ model we have this equation:

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{max\left[\frac{1}{\alpha^*}, \frac{SF_2}{\alpha_1 \omega}\right]} \tag{12}$$

where F_2, α^* and α_1 are functions of model constants[11].

3. Method

As mentioned before, in order to investigate the credibility of Miller pressure loss coefficients in different conditions and assess possible differences, doing 3D simulations of fluid flow is a proper method but first of all determining the best numerical solution method that is reliable in this kind of problem is vital. 6 different methods have been tested in present work by performing 3D simulations in ANSYS Fluent. These methods are classified in separate aspects that have been explained in following sections.

3.1. Geometry details

Four different geometries (in terms of the curvature radius (r_c) to pipe diameter (D) ratio) with lengths equal to 120 times pipes diameter (120D), consisted of two equal 60D parts (before and after the bend) have been selected and created in ANSYS. Fully developed flow in bend inlet and developed flow reformation after the bend are main terms of Miller tests and selecting this length is suggested by Cengel et al. in this kind of problems to fulfill these terms, albeit this leads to a large flow domain and increment of simulation costs[12]. Flow domain is illustrated here:

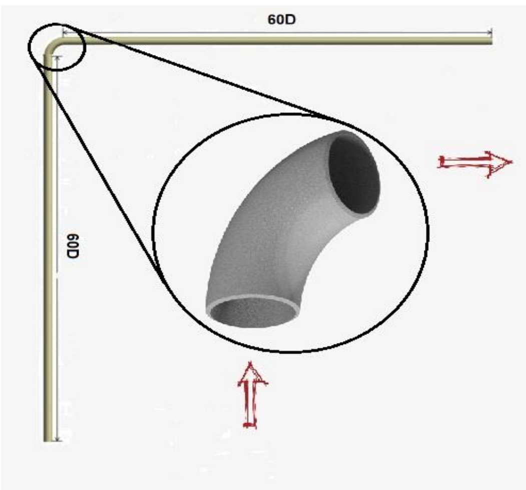

Figure 2: Flow domain in 3D simulations

Geometric characteristics and desired Miller loss coefficients are mentioned in the following table:

Table 1: Geometric characteristics and loss coefficient of simulation cases

$({r_c}/{D})$ ***Ratio***	***Miller pressure loss coefficint*** $(Re = 10^6)$
2	0.16
3.5	0.16
5	0.184
6	0.2

3.2. Boundary conditions

In order to conform simulations to Miller tests, boundary conditions in simulations determined alike tests. For inlet flow, as long as Reynolds number should have been identical to Miller tests (10^6), inlet velocity has been set equal to 25.12 m/s. Since in Miller tests the flow directly discharged into atmosphere, outlet gauge pressure has been set to zero.

3.3. Loss coefficient calculation method

In terms of forming turbulent and velocity vectors, there is a huge difference between the flow passed through a bend and the flow inside a straight pipe due to additional losses. In next figure, dimensionless pressure values have been demonstrated in upstream and downstream of the bend. So the value equal to the difference between two lines after shaping linear form is pressure loss coefficient due to bend's presence. This is the mathematical method that has been utilized to calculate the bend pressure loss coefficient in each case. Note that in each case, part of the total loss is due to significant separation exactly after the bend and the other part is due to the reformation of velocity profile which needs longer pipe length to occur.

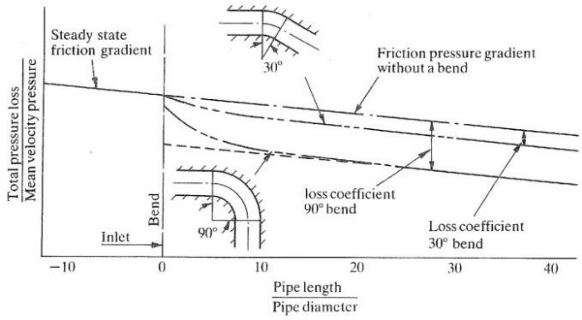

Figure 3: Flow pressure gradient after bend [5]

[DOI: 10.22068/ase.2019.494]

3.4. Mesh settings

In present research, solution grids have been structured from the flow domain with 2 separate form concepts: structured and unstructured. The former grid type has been assigned by utilizing ANSYS ICEM CFD package while the latter has been formed in ANSYS Meshing tool. These two classifications have been illustrated in the following figure for inlet part of the domain:

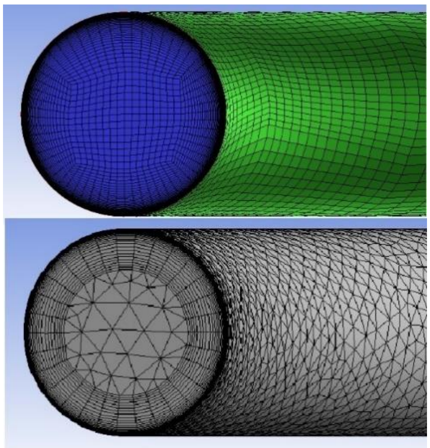

Figure 4: structured and unstructured grids from inlet point of view

It should be noted that all of these grids have been assessed with several mesh quality techniques like element skewness, orthogonality and aspect ratio.

3.5. Turbulence models and near wall consideration

Just like other problems in fluid mechanics that deal with turbulence phenomena, in this problem also there is no unique model that overcome others and choosing the right model depends on problem physics and flow characteristics. As mentioned before 3 famous RANS models have been tested in this research. Kindly note that each of these models modify the grid by changing the first layer thickness so that its y^+ value fits in suitable range.

Table 2: Selected turbulence models from similar researches

Model Name	Model Type	Flow modeling near pipe wall	y^+ range
Standard $k-\varepsilon$	2 Equation (RANS)	Scalable Wall Function	$30 < y^+ < 300$
SST $k-\omega$	2 Equation (RANS)	-------------------	$y^+ \approx 1$
Realizable $k-\varepsilon$	2 Equation (RANS)	Enhanced Wall Treatment	$y^+ < 5$

3.6. Selected numerical methods specifications

As previously stated, in order to determine the best numerical solution method, all possible choices in all aspects that affect final results have to be asserted. In present work, combinations of possible choices in different key aspects led to 6 final methods that have been utilized in 3D fluid flow simulations to calculate 4 different bend's pressure loss coefficients and validate these methods against Miller tests. These methods specifications are mentioned with details in Table (3) in next page.

Performing 3D simulations according to these 6 solution methods contain remarkable hints as follows:

- Beside two main boundary conditions that have mentioned earlier (inlet velocity and outlet pressure), other settings like temperatures, water characteristics as fluid in pipes and turbulence models constants remained unchanged and set by default.
- In terms of velocity-pressure coupling, all simulations have been performed with SIMPLE algorithm and in terms of equations spatial discretization, second-order upwind scheme have been utilized in all cases. Other possible options have been asserted too but it seemed that this problem is somehow rigid against these terms.
- Mentioned number of cells in second column of Table (3) is for a grid that has been formed after passing mesh independency test. This allegation has been shown in next figure in which calculated loss coefficient error changes with number of cells. It highlights that after some points, shrinking the grid cells does not make any sense.

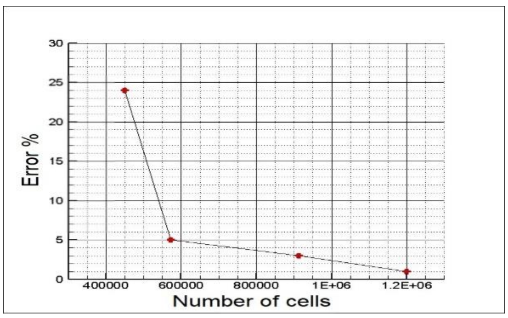

Figure 5: Mesh independency in performing 3rd method on 2nd geometry

Table 3: Selected solution methods specifications

Method	Grid type	Approximate final number of cells	Convergence criteria	Turbulence model	Near wall flow modeling method	Estimated required time for each simulations
1st	Unstructured	1,100,000	10^{-3}	Standard k-ε	Scalable Wall Function	1 hour
2nd	Unstructured	5,000,000 (with special surface dimensions)	10^{-6}	Standard k-ε	Scalable Wall Function	5 hours
3rd	Unstructured	1,200,000	10^{-6}	Standard k-ε	Scalable Wall Function	3,5 hours
4th	Unstructured	4,400,000	10^{-6}	SST k-ω	----------------	5,5 hours
5th	Structured	4,000,000	10^{-6}	SST k-ω	----------------	6 hours
6th	Structured	4,000,000	10^{-6}	Realizable k-ε	Enhanced Wall Treatment	6 hours

4. Results and discussion

After performing 3D simulations utilizing these 6 methods on 4 selected geometries, all methods have been assessed in terms of calculating bend loss coefficient and comparing them versus Miller tests results to determine best method. This coefficients have been attained after drawing pressure gradient diagrams that look alike figure (3) and dividing ΔP to average dynamic pressure of flow. An example of pressure gradient diagrams illustrated here:

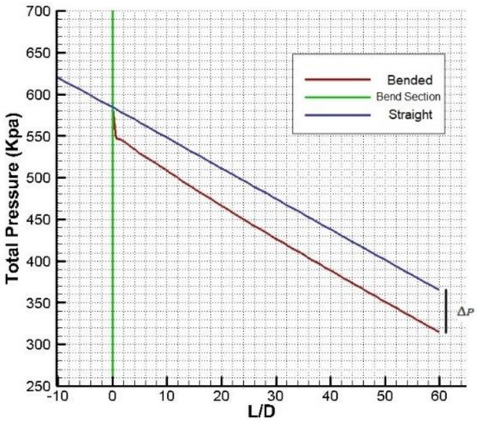

Figure 6: Pressure gradient after utilizing 5th solution method on 3rd geometry

Comparing between these 6 methods is done after calculating errors against Miller's loss coefficients. These errors are calculated using simple percentage error equation:

$$\text{error} = \left|\frac{K_{simulation} - K_{Miller}}{K_{Miller}}\right| \times 100 \qquad (12)$$

With calculating these errors for different geometries in each case, it has been revealed that these methods show different behavior and at first it meant that there is no solution method that guarantees minimum error in calculating pressure loss coefficient. But after completing error calculations for each case and each method, as it has been demonstrated in Figure (7) fifth solution method is the most successful one. Third solution method is also acceptable with totally less than 3% in average error and particularly lower computational costs.

It's quite interesting to note that although second method has relatively high error in calculating loss coefficients (in the region of 4%) it has the lowest standard deviation and errors in all 4 cases are almost equal together. This means that if we utilize these 6 methods in a new case with new geometric characteristics, second method is the most predictable one in terms of error. 1st solution method has lower average error than sixth method even though it has rough grid and higher convergence criteria.

[DOI: 10.22068/ase.2019.494]

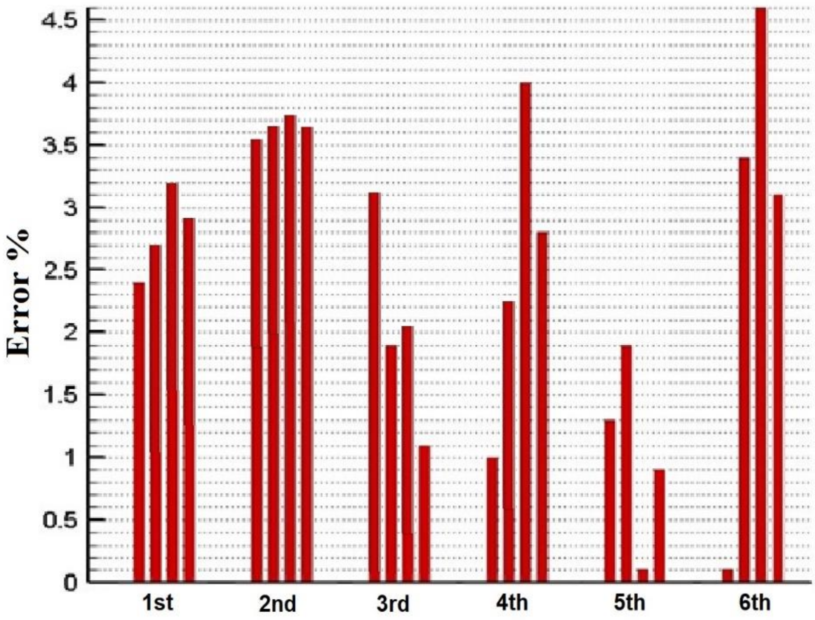

Figure 7: Final errors for each numerical solution method in each case

Calculated errors from equation (12) have been shown in figure 7 for each case in a bar chart. In every 4 bar set for each solution method, error for each geometry from first to fourth is the bar from left to right. All errors are below the limit of 5% and at first glance this looks like that in general all of these 6 methods are successful but with the high value of dynamic pressure in this kind of problem even 0.1 % of error in loss coefficient value can lead to a large deflection in calculating pressure drop.

5. Conclusions

In present research, 6 different numerical solution method have been assessed with 4 separate geometries to find and determine the best one in simulating fluid flow in bended pipes and calculating pressure loss coefficients. These values are available after Miller's several valuable efforts in conducting experiments about water flow in long pipes with bends. The main problem with these available data is that they had been attained in a specific experimental condition and some phenomena that are present in practical problems like engine intake system bends, have inconsistency with those test. For instance, Miller used an incompressible fluid for his tests but intake manifold contains air as a compressible fluid and this distinction is a research gap that needs to be investigated.

But it's obvious that in order to perform 3D simulations to pursue these corrective researches, at the first step determining a validated numerical solution method is essential. These methods are in fact a combination of all possible choices in separate aspects of a complete numerical solution method.

After performing 3D simulations of Miller tests in ANSYS Fluent and calculating errors in resultant loss coefficients, several interesting assertions have been attained as follows:

- The error in calculating loss coefficient varies from case to case in each method and it depends on geometric characteristics. So literally there is no method that surely guarantee minimum error.
- Nonetheless in average error of all 4 cases 3rd method with structured grid and SST k-ω turbulence modeling is the most successful one. This assertion is in accordance with literature that in problems with flow separation this model is more suitable.
- Difference between final errors are about 0.5% in each method but even this apparently negligible values can lead to a large amount of pressure drop.
- Against expectations, 6th method with a refined structured grid and a famous turbulence model, has the highest average

[DOI: 10.22068/ase.2019.494]

error (even higher than first method with an unstructured rough mesh).

- The problem is very sensitive to grid type and turbulence models while changing the velocity-pressure coupling method or equations spatial discretization scheme was affectless.
- Second method with largest number of cells have the lowest standard deviations which means that it also is a suitable choice for undetermined problems.

References

[1] G. Technologies, "GT-SUITE Flow Theory Manual," GT-SUITE, Ed., ed, 2016.

[2] D. E. Winterbone and R. J. Pearson, *Design Techniques for Engine Manifolds: Wave Action Methods for IC Engines*. Professional Engineering Pub. Limited, 1999.

[3] W. R. Dean, "XVI. Note on the motion of fluid in a curved pipe," *The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science,* vol. 4, no. 20, pp. 208-223, 1927/07/01 1927.

[4] J. R. Womersley, "Method for the calculation of velocity, rate of flow and viscous drag in arteries when the pressure gradient is known," *The Journal of Physiology,* vol. 127, no. 3, pp. 553-563, 1955.

[5] D. S. Miller, *Internal Flow Systems*. BHRA (Information Services), 1990.

[6] N. I. Abou-Haidar and S. L. Dixon, "Pressure Losses in Combining Subsonic Flows Through Branched Ducts," no. 79047, p. V001T01A041, 1992.

[7] S. Maharudrayya, S. Jayanti, and A. Deshpande, "Pressure losses in laminar flow through serpentine channels in fuel cell stacks," *Journal of Power Sources,* vol. 138, no. 1-2, pp. 1-13, 2004.

[8] V. KUMAR, "SIMULATION AND FLOW ANALYSIS THROUGH DIFFERENT PIPE GEOMETRY," Bachelor of Technology, Civil Engineering, NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA,, 2010.

[9] J. M. Desantes, J. J. López, M. Carreres, and D. López-Pintor, "Characterization and prediction of the discharge coefficient of non-cavitating diesel injection nozzles," *Fuel,* vol. 184, pp. 371-381, 2016/11/15/ 2016.

[10] S. V. C. G.B.Nimadge, "CFD ANALYSIS OF FLOW THROUGH T-JUNCTION OF PIPE," *International Research Journal of Engineering and Technology (IRJET),* vol. 4, no. 2, 2017.

[11] ANSYS, "Fluent Theory Guide," ed, 2015.

[12] Y. A. Çengel and J. M. Cimbala, *Fluid mechanics fundamentals and applications*. Boston, Mass: McGraw-Hill Higher Education, 2006.