Anysys fluent simulation of turbulent flow engineering essay

The characteristics of fluid flow with sudden expansion in a 1: 2 diameter ratio pipe are investigated using ANSYS Fluent. Results show fluid re-circulates just after expansion, length of recirculation zone approximates to 0. 35m. Velocity, turbulence intensity and pressure vary along pipe length in accordance with Bernoulli’s principle. Influence of change in turbulence models on accuracy is also investigated with the Reynolds Stress model providing the relatively best fit although other turbulence models (realizable k-Îµ and SST k-Ï‰) provide reasonably close fitting models. Results were checked for mesh independence and validated.

Computational Fluid dynamics (CFD) involves computational simulation of fluid flows in different situations employing numerical solution of basic flow equations e. g. the continuity equation and other equations over a discretized unit reference (Versteeg, and Malalasekera, 2007). The usage of CFD transcends the traditional scope of chemical engineering profession into wider areas such as oceanography, biomedical engineering electrical circuitry, etc (Fairweather, 2011).

Sudden expansion in pipes involves fluid flow from a smaller hydraulic diameter to a larger one. Flow separation usually occurs in a sudden expansion scenario, where a part of the fluid flows in opposition to the main fluid flow. This are called eddies, and are strong contributors to the irreversibility of practical flows as energy is dissipated by this eddies. Thus it is of great significance to be able to model eddies in a sudden expansion flow adequately and observe the characteristics of this recirculation zone (efluids, 2011; Gharegbagi and Ali, 2011; Mahmud, 2011: Roy, et al 2010).

Sudden expansion is a simple looking but intriguing case of fluid flow in pipes. Sanmiguel-Rojas (2010) implies that not many significant studies have been done on instabilities encountered in steady, turbulent, sudden expansion fluid flow with respect to spatial structure of piping with D2/D1 = 2. However, previous remarkable work in this field includes Roy, et al (2010) and Mansoori and Bazargan-Lari (2007).

Examples of scenarios in which the above phenomenon occurs include; Flows into a tank, oil drilling and extraction, plug flow reactors, combustion engines, aerodynamics, etc.

Software

ANSYS Fluent is a commercial CFD package that models flow via the finite-volume method (a variation of the finite difference method) created by the company Fluent (now part of ANSYS Inc.). Pre-processing of the case study (meshing) was done on Gambit which comes along with Fluent (now ICEM). The version of Fluent employed in this report is 12. 1 (CFD-online, 2011; Weidner, 2011; ANSYS, 2009).

This report covers the Reynolds Average Navier-Stokes (RANS) modelling of turbulent flow with sudden expansion in a 1: 2 diameter piping, using the pressure based solver and the second order upwind difference scheme in ANSYS Fluent. Effects of changes in turbulence models on computational time, and accuracy would be examined, visual plots would be used to describe and analyse modelling results.

SIMULATION METHODOLOGY

Fig 1: diagrammatic representation of simulation process (Fairweather, 2011)

Nature of Fluid flow under consideration

Calculating the Reynolds number of the flow helps to determine the nature of the flow. At Normal Temperature and pressure (P = 101. 325 kgm-2, T = 288. 16 K)

Generally it is accepted that flows with a Reynolds number (Re) > 4000 are turbulent in nature. Therefore it is established that the flow under consideration is a turbulent flow

Reynolds-Averaged Navier Stokes (RANS)

RANS involves the time averaging of the equations that govern turbulent fluid flow to capture information on variations that occur on a minute scale while avoiding horrendously lengthy computation times. RANS represents variations as a mean such that

; ; ; and P

RANS is employed in obtaining the equations that were numerically solved in this report assuming constant velocity and viscous flows (Fairweather, 2011).

Geometry:

The geometry consists of two pipes of diameter ratio 1: 2 joined together through which fluid flows with no bends as shown below

Fig 2: geometry of pipe showing mesh grid/mesh discretization

Governing Equations

Continuity equation:

Momentum equation (x-direction only)

Where:; ; ; ; ; ;

TURBULENCE MODELS

Realizable k-Îµ model

The k-Îµ model is a two equation model that assumes a linear relationship between Reynolds stress and rate of strain. It has the advantages of fast computation time, wide usage and extensive validation. However, it predicts badly the length of eddies for complex flows. The realizable k-Îµ model is an update to the model based on observed strengths and weaknesses of the standard k-Îµ model (Fairweather, 2011; ANSYS, 2009).

Below is a mathematical representation of the standard k-Îµ model

Where:

Î¦ = k or Îµ; SÎ¦= source term for k or Îµ; Sk= G-Ï Îµ (production rate of k-destruction rate of k); SÎµ= (C1G-C2Ï Îµ)(Îµ/k) = (production rate of k-destruction rate of k)

;

N. B. for this simulation: ; and

SST k-Ï‰ model

The k-Ï‰ model is also a two equation model based on the Wilcox k-Ï‰ model. It is suitable for wall bounded flows and free shear flows as it performs low Reynolds number corrections, computation time is relatively fast and accuracy is better than the k-Îµ model in most cases. Ï‰ is specific dissipation rate and is analogous to a ratio of Îµ/k. The SST k-Ï‰ model is an improved version of the standard k-Ï‰ model (ANYSYS, 2009).

Reynolds Stress Model

This is a very rigorous model, with seven equations unlike the preceding 2-equation models. It provides more accuracy where other models are faulty e. g. impinging flows and can predict fluid flow for a lot of cases closely without any dedicated / individual adjustments. However, computing costs are large (Fairweather, 2011)

The first six equations of the RSM model can be condensed into the equation below

Where:

; ; ;

The seventh equation (turbulence dissipation rate) is

N. B. in this simulation: ; and

Numerical methods

The discretization employed is the finite volume method. It is a variant of the finite difference method. This scheme splits up the domain into discrete control volumes over which the control equations are resolved using a truncated Taylor series expansion. Finite volume method is the most established of Discretization schemes in CFD modelling. Convective fluxes were evaluated with the second order upwind-difference scheme (Fairweather, 2011; Versteeg, and Malalasekera, 2007).

Boundary conditions

Table 1: boundary conditions for numeric solution (adapted from Versteeg, and Malalasekera, 2007)

Realizable k- Îµ model

SST k-Ï‰ model

Reynold Stress model

Inlet

k = 0. 01148438 m2s-2

Îµ = 0. 02888982 m2s-3

k = 0. 1148438 m2s-2

Ï‰= 27. 95085

Rij =

Îµ =

Outlet

;

;

;

Interior

k = 0 ; Îµ = 0

k = 0 ; Ï‰ = 0

Rij = 0 ; Îµ = 0

Walls

law of the wall

Law of the wall

Wall functions

Convergence criteria and levels

For all the equations solved by each model, a uniform convergence criterion of 1. 0 x 10-4 was used for every equation solved. The value represented an informed compromise between acceptable accuracy and realistic computation time (ANYSYS, 2009). It is worthy of note that for the RSM model, this relatively stringent criterion caused the number of iterations to exceed 14, 000 without any obvious improvement in results as shown in fig 2. Therefore a cap of 4, 000 iterations was placed on the RSM calculations. Results show there was no ensuing negative impact on accuracy of numerical solution.

Fig 3: Iteration length for RSM model showing

Mesh Independence test

The table below shows that results from the modelling experiment are similar and essentially the same within three (3) decimal places of precision irrespective of mesh size employed. Also since assurance of mesh independence cannot be guaranteed by mere reduction in cell size (Sloan et al, 1986), an attempt was made at adaptive meshing to attenuate important flow variations and phenomenon with the same results obtained.

Table 2: Grid/Mesh independence of simulation

Gambit Mesh/Grid size

Volume of unit cells

Mass flow rate at inlet [kgs-1]

Mass flow rate at Pressure-outlet [kgs-1]

Error

Percentage

Difference (%)

5

439, 993

0. 016809944

0. 016809996

-5. 22E-08

3. 09 x 10-4

7

163, 311

0. 01678467

0. 016784551

1. 19E-07

7. 08 x 10-4

10

55, 182

0. 016728994

0. 016729204

2. 1E-07

1. 255 x 10-3

10b

100, 693

0. 016728994

0. 016728895

-9. 9E-08

5. 9 x 10-4

15

16, 750

0. 016609019

0. 016608695

-3. 24E-07

1. 95 x 10-3

N. B. 10 b means mesh size 10 with boundary layer mesh added (adaptive meshing)

Grid optimization (Mesh finesse Vs Time trade off)

The greater the volume of unit cells in grid per geometry, the better the accuracy of numeric analysis. However, within the scope of grid independence, results are relatively uniform irrespective of mesh size. The cost of finesse of grid is computation time could be noticed with the case of mesh size 5 (439, 993 cells) which took almost forever to compute using the RSM model and had to be terminated. Thus mesh 10 (55, 182 cells) and 10b

(100, 693 cells) were employed for analysis with other mesh sizes serving as validation checks

RESULTS AND ANALYSIS

Part 1

Taking a close look at flow close to the walls of the pipe, we see the effect of sudden expansion resulting in backflow of fluid creating velocities in the opposite direction (red box). Recirculation zone is approximately 0. 37m in length. We also can see how the fluid adjust to changes in geometry with a sharp rise velocity to fill the voids created by liquid moving backwards then a gradual decrease as pressure pile us towards the exit of the pipe

Fig 4: velocity variation along length of pipe close to the walls showing effects of recirculation

Fig 6 shows the variation in turbulence intensity. It can be seen that the flow becomes more turbulent around the recirculation zone with dead (stagnant) flow occurring just at the corners of the pipe. Fig 7: displays the total pressure variations in the pipe. It can be noted that sudden expansion causes a drop in total fluid pressure. Fig 8 shows the radial velocity and profile. It can be noted that velocity variation in the radial direction is minimal, which is typical of plug fluid flow depicted by fig 5. Fig 9 is a streamline plot of axial velocity, velocity variation along the axial direction is more dominant than in the radial direction, also worthy of note is the length of the recirculation zone (black box) and the reattachment zone.

Fig 5: stages of flow development at different positions on pipe length

Fig 6: Turbulence intensity profile of fluid along length of pipe

Fig 7: Total pressure profile of fluid along length of pipe

Fig 8: Radial velocity profile of fluid

Fig 9: streamline plot of axial velocity of fluid

Part 2

Fig10(a-c) shows axial velocity profiles for different turbulent models in order of increasing complexity (‘ realizable k-Îµ’ âŸ¶ ‘ SST k-Ï‰’ âŸ¶ ‘ RSM’). Curves get smother showing a more gradual response of the fluid to changes and also approach exact solution, as model complexity increases. However, all the essential features of the fluid flow are well represented by all models.

Fig 11(a-c) displays turbulence intensity variations, more variation details are captured as model increases in complexity. Worthy of note is that the SST k-Ï‰ model provides a more detailed picture of turbulent intensity variation in reference to the other models picking up intensities as low as 5. 42 x 10-5 %, while the realizable k-Îµ picks up a minimum of 0. 336% and RSM 1. 45%

Fig 12(a-c) shows streamline plot of axial velocity, though length of recirculation zone remains approximately the same the representation of velocity magnitude in recirculation zone varies visibly for each model.

Fig 13(a-c) is the radial velocity profile; the SST model indicates larger radial velocities along pipe length than both than both the realizable k-Îµ and the

RSM models. For all models radial velocity variation is dominated by axial velocity variations

Fig 10a: k-Îµ model

Fig 10b: SST model

Fig 10c: RSM model

Fig 11a: k-Îµ model

Fig 11c: RSM model

Fig 11b: SST model

Fig 12a: k-Îµ model

Fig 12b: SST model

Fig 12c: RSM model

Fig 13a: k-Îµ model

Fig 13b: SST model

Fig 13c: RSM model

VALIDATION OF RESULTS

For CFD, convergence of numerical iterations does not really count for much as Versteeg and Malalasekra (2007) put it “ results are at best as good as the physics embodied in it, or at worst as good as the skill of the operator”.

Thus, validation of results becomes extremely important. The results obtained herein would be validated thus:

Bernoulli's equation

For an ideal fluid flow Bernoulli's equation enables us to calculate the velocity at any point in the pipe (assuming constant flow rate, and negligible friction losses). Therefore we can validate output velocity from fluent using this principle (Roymech, 2011).

Where vin = 1. 73855 ms-1, P1= 101. 325 kgm-2, P2= 101. 325 kgm-2, g = 9. 81 ms-2; Ï = 1. 225 kgm-3; z1 = 0. 1m; z2 = 0. 1m;

Therefore

Mass flux variation results from Fluent

The third mechanism for validation will be the CFD package fluent itself. Analysis of the computation results as presented in table 4. 0, show that value of errors resulting residuals is very low (less than 0. 0095%) indicating conservation of mass during numerical calculations which lend credit to suitability and accuracy of model.

Table 3: comparison of percentage error of each model

MODEL/mesh volume

K-EPSILON (%)

SST K-OMEGA (%)

REYNOLD STRESS (%)

5

0. 000309

0. 00352

N/A

7

0. 000708

0. 004468363

0. 000673233

10

0. 001255

0. 007867

0. 001124

10 b

0. 000153

0. 00258

0. 001488

15

0. 00195

0. 000783

0. 00927

N. B. 10 b means mesh size 10 with boundary layer mesh added

Research journals

In addition to the above validation processes, the results of modelling experiment reported in this work were compared with previous research works such as (Roy, et al 2010), (Mansoori and Bazargan-Lai 2007) and (Teyssandiert, 1973). Results obtained corroborated foregoing analysis and results obtained it the above mentioned papers.

CONCLUSION

In summary, CFD modelling of sudden expansion flow in a 1: 2 diameter ratio piping posses the following characteristics.

Sudden expansion in pipe flow results in local pressure losses

Flow fully develops into plug flow before exit at outlet and majority of the variations occur axially along reactor length

Recirculation of fluid occurs after sudden expansion for a lengthspan of approximately 0. 35m along pipe

Viscous effects along wall boundaries help dissipate energy of turbulent eddies

The realizable k-Îµ model predicts the size and strength of recirculation zone poorly, but as flow develops into plug flow, the model's accuracy remarkably improves with reference to the other models tested.

Turbulence models become better with increase in complexity of model from k-Îµ to SST k-Ï‰ to RSM. Ability of other models to better the k-Îµ model in recirculation zone prediction can be attributed to embedded corrections for boundary layer flow, turbulent kinetic energy and dissipation rates.