CFD ANALYSIS FOR FLOWMETER

Harsh r Patel[1], Darshan Ganesan K.[2], Faizan M. Saiyed[3]

[1] Student, Mechanical Department, Vadodara Institute of Engineering, Gujarat, India
[2] Student, Mechanical Department, Vadodara Institute of Engineering, Gujarat, India
[3] Student, Mechanical Department, Vadodara Institute of Engineering, Gujarat, India

Abstract

CFD flow simulation methods have been used to model the flow through the proving facilities. Simulations have been run to represent 10 inch meters. It has been found that significant swirl is generated within the system pipe circuit which creates the turbulence and as result we get the wrong interpretation at flow meter. This errors can be resolved using two methodologies out of which is using tube bundles i.e. the tube bundles installed in the 10 inch lines effectively remove this swirl. However, flow conditions at the flowmeters' inlets are not ideal with skewed and flattened velocity profiles being predicted at the meters' inlets. This is particularly true of the 10 inch flowmeters.

Keywords – *cfd, flowmeter, turbulence, laminar flow, tubebundle.*

---

1. INTRODUCTION.

CFD is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. The pipes on the way has many elbows, joints, T-ends/joints, etc. which resists the flow of any fluid in a pipe. Due to these projections the flow which is supposed to be laminar turns out into turbulent and this turbulent flow enters the flow meter. In spite of coverings inside and coatings the eroded surface is also responsible for the change of flow of any fluid. The sensors present, though highly reactive to flow sometimes fail to recognize the quantity except light waves flow meter which are most costly. Hence the main aim of our project is to convert turbulent flow into laminar flow before entering the flow-meter.

1.1 IMPLEMENTATION.

A. Modulation in pipe.

As we know, the main aim of our project is to convert turbulent flow into laminar flow before entering into flow-meter. So we need to implement a structure like laminar tubes in pipe section after having turns within elbow or T-Joints. These laminar tubes are to be placed before flowmeter which reduces the turbulence by converting turbulent flow into laminar flow.

B. Process Input Parameters.

The below values have been officially provided by Daniel Industries plant situated at Hajira as a problem statement. These values include Mass Flow Rate (Q_{max}), Maximum Pressure that can be sustained by our designed model (P_{max}), Working Pressure (P_{work}), Design Temperature (T_{design}), Working Temperature Of Gas (T_{gas}), Surrounding Temperature ($T_{ambient}$) and Velocity in X Direction.

TABLE-1: PROCESS INPUT PARAMETERS

PARTICULARS	**VALUE**	**UOM**
Q max.	30	M3 / h
P max.	92	BAR
P work.	45-60	BAR
T design.	-10 - +55	‘C
T gas.	+5 +30	‘C
T ambient	+15 +45	‘C
Velocity	10 in X-direction	m/s

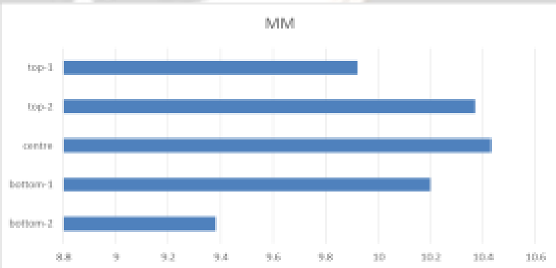
C. Error Calculation.

TABLE-2: ERROR CALCULATION

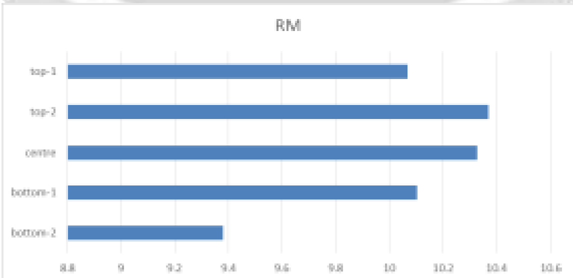
	TOP-1	**TOP-2**	**CENTRE**	**BOTTOM-1**	**BOTTOM-2**
MM	0.75	0.74	0.74	0.75	0.77
RM	0.75	0.74	0.74	0.75	0.77
MUT	0.27	-0.06	-0.23	-0.13	0.22

As per given data we came across the *swirl angle error* i.e. the medium has turbulent flow before entering flow meter which is to be avoided. The above values are for Master Meter (MM), Reference Meter (RM) and Meter Under Test (MUT). The three meters here are installed as per practical application seen. The error seen here is the practical values are beyond the optimum required values; the value should not exceed 0.3 which we have overcome using our *tube bundles* using Meter Under Test.

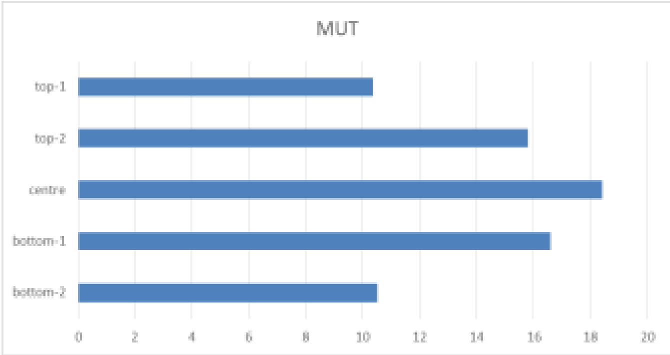
D. Graph Representation.



GRAPH-1: VELOCITY IN MM



GRAPH-2: VELOCITY IN RM



GRAPH-3: VELOCITY IN MUT

Thus, we get the more clear information regarding the errors at various installed meters through graphical representation.

2. MODELLING WORK DETAIL.

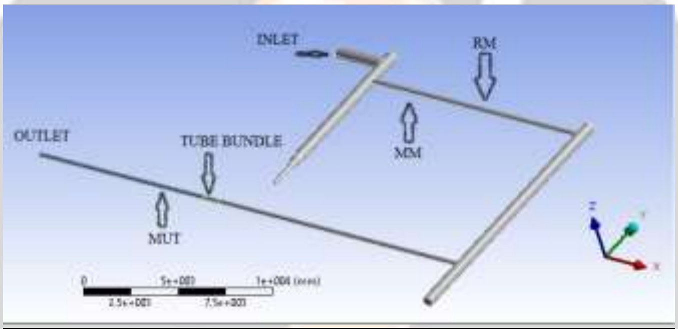


FIG -1: MODELING WORK FOR 10"X600

The figure shown above gives the detailed sketch as per practically observed by us of various meters, junctions and turns.

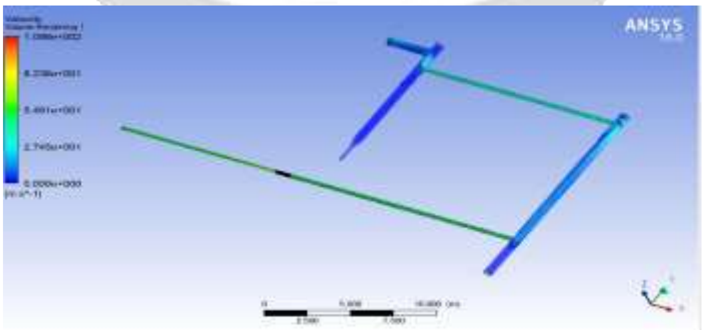


FIG -2: CFD WORK FOR 10"X600

Useful Equations.

(Q1-Q2)/Q1 = Error. (E)

$(\pi/4)D^2$ = Area. (A)
AV = Continuity Equation. (Q)

A/a = Number of tubes. (N)

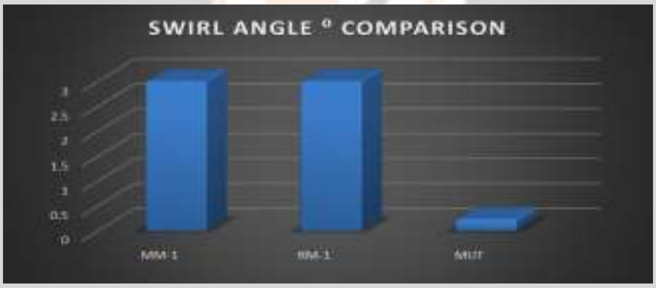
Swirl Angle Plane =

A cos[{velocity (u) * normal (X) + velocity (v) * normal (Y) + velocity (w) * normal (Z)} / Velocity].

2.1 Swirl Angle Calculation

TABLE-3: SWIRL ANGLE CALCULATION

SWIRL ANGLE COMPARISON	
LOCATION	**ANGLE ^{0}**
MM-1	2.999
RM-1	2.998
MUT	0.23



GRAPH-4: SWIRL ANGLE COMPARISION

The above comparison shows that the swirl angle is created more at MM – 1 and RM – 1. This is because the fluid directly enters the flow meter after the bents whereas in MUT, the swirl angle is negligible as the fluid passes through *tube bundles* first then in flow meter.

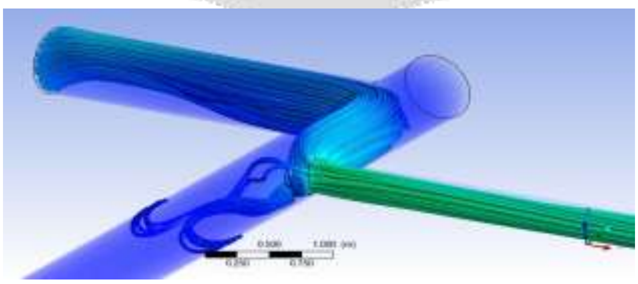


FIG -3: INLET TO MASTER METER-1

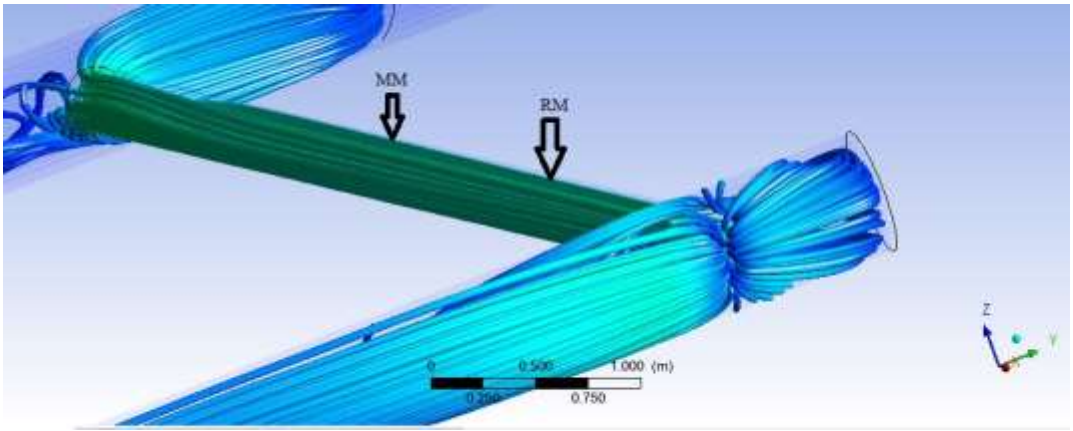


FIG -4: FLOW THROUGH M.M. AND R.M.

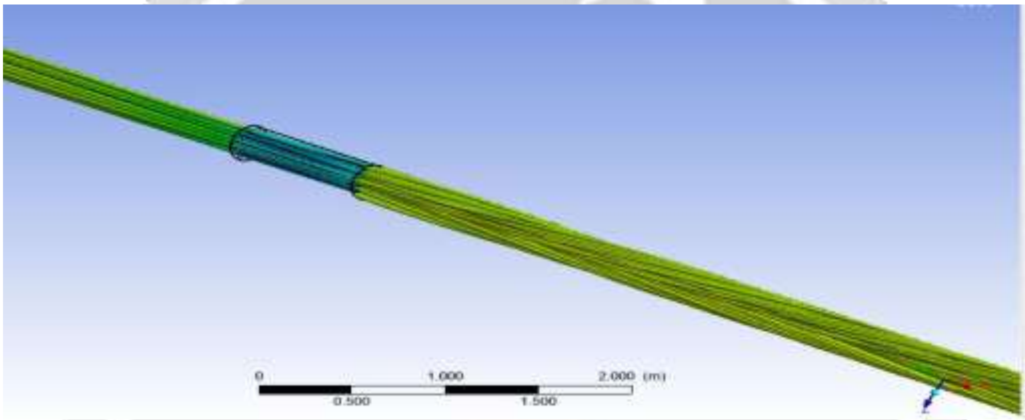


FIG -5: Tube Bundle Before M.U.T.

Illustrated figures give the information as follows;

Fig. 3 shows, swirl is created before Master Meter – 1.

Fig. 4 shows, the position of Master Meter and Reference Meter and generation of swirl after Reference Meter.

Fig. 5 shows, flow through tube bundles.

3. CONCLUSION.

Thus we conclude from above figures that, the predicted flow pattern in the CFD for simulations of the 10”X600 master meter in line with the 10” test meter or the 10” reference meter.

Also it shows that, flow conditions entering at inlet, the master meter has a swirl, or velocity profile distortion (maximum at the mixing point of inlet.)

The tube bundle effectively remove the swirl but the velocity profile entering the test and reference flow meters is definitely skewed.

4. REFERENCES.

1. Anusha Rammohan, Aditya Bhakta, Vinay Natrajan, John Ward, and Manoj Kumar -“Flow swirl and flow profile measurement in multiphase flow” (GE Global Research, Bangalore, India,GE Oil and Gas - Measurement and Control, Groby, UK).

2. **Dinesh Singh – (**M. Tech. Scholar, NRI Institute of Research and Technology, Bhopal, Indi) *“* Swirl Diffuser Design and Performance Characteristics for Air Flow in Air Conditioning of an Automobile-A Review”.*-ISSN 2278 – 0211*.

3. Akay A. Islek - (Master of Science in the Woodruff School of Mechanical Engineering). “The Impact of Swirl in Turbulent Pipe Flow”- 2004.

4. C G Kim, B H Kim, B H Bang and Y H Lee-“Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model” - 2015 IOP Conf. Ser.: Mater. Sci. Eng. 72 042044 - (http://iopscience.iop.org/1757-899X/72/4/042044)

5. Arun S , Rakesh P-(Department of Mechanical Engineering,College of Engineering Trivandrum (CET) Thiruvananthapuram) “Computational Evaluation of Spray Characteristics in Pressure Swirl Atomizers”-International Journal of Scientific & Engineering Research, Volume 5, Issue 7, July-2014 ISSN 2229-5518.

6. Ramazan-“ Cfd Simulation Of Swirling Effect In S-Shaped Diffusing Duct By Swirl Angle 20^0” [Ramazan / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622].

7. Dorin Stanciu-[Dept. of Engineering Thermodynamics, Polytechnic University of Bucharest] “The Influence of Swirl Angle on the Irreversibilities in Turbulent Diffusion Flames”(Int. J. of Thermodynamics ISSN 1301-9724-Vol. 10 (No. 4), pp. 143-153, December 2007.

***