ISSN: 2321-8169
232 – 234

Calculation of Swirl Angle Using CFD

Harsh R. Patel
Student, Mechanical Department.
Vadodara Institute Of Engineering.
Gujarat, India.
harsh9435.patel@gmail.com

Faizanali M. Saiyed
Student, Mechanical Department.
Vadodara Institute Of Engineering.
Gujarat, India.
farhanshaiyad453@hotmail.com

Abstract - CFD flow simulation methods have been used to model the flow through the proving facilities. It has been found that significant swirl is generated within the system pipe circuit. 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, swirl angle, tube bundle, flowmeter, laminar flow.*

****

I. 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 and swirl are generated. Hence the main aim of our project is to calculate the swirl angle and try to remove the error.

II. IMPLEMENTATION

2.1 Process Input Parameters

TABLE 1: PROCESS INPUT PARAMETER

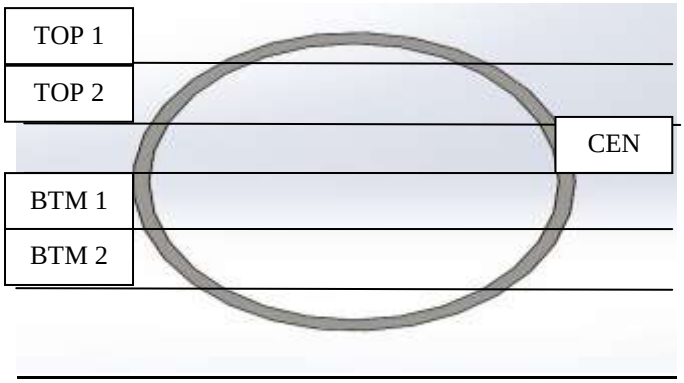
PARTICULARS	VALUE	UOM
Q max.	30	M3 / h
P max.	92	BAR
P work.	45-60	BAR
T design.	-10 - +55	'C
T gas.	+5 +30	'C
T ambient	+15 +45	'C
Velocity	10 indirection	m/s

These are the input parameters used for analysis, given by the external faculty.

2.2 Velocity at Different Location

TABLE 2: Velocity Table

LOCATION	INLET	MM	RM	MUT
bottom-2	**10**	**9.38**	**9.38**	**10.51**
bottom-1	**10**	**10.2**	**10.102**	**16.58**
Center	**10**	**10.43**	**10.327**	**18.41**
top-2	**10**	**10.37**	**10.37**	**15.79**
top-1	**10**	**9.92**	**10.067**	**10.36**



This velocity table illustrates various velocities at different points over the total cross section area of pipe from top to bottom.

ISSN: 2321-8169
232 – 234

2.3 Error Calculation

TABLE 3**:** For Master-Meter

MM	
LOCATION	**ERROR %**
	MM
TOP-1	0.75
TOP-2	0.74
Centre	0.74
BOTTOM-1	0.75
BOTTOM-2	0.77

TABLE 4: For Reference Meter

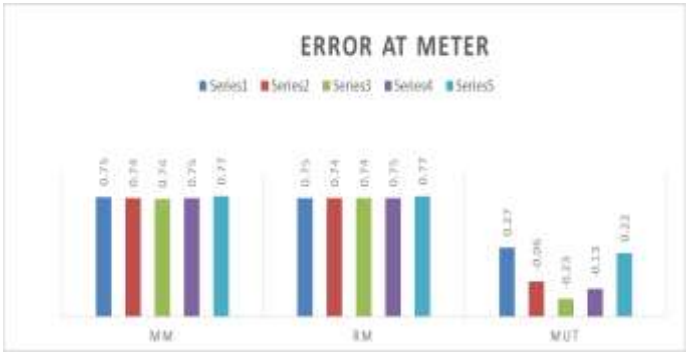
RM	
LOCATION	**ERROR %**
	RM
TOP-1	0.75
TOP-2	0.74
Centre	0.74
BOTTOM-1	0.75
BOTTOM-2	0.77

TABLE 5: For MUT

MUT	
LOCATION	**ERROR %**
	MUT
TOP-1	0.27
TOP-2	-0.06
Centre	-0.23
BOTTOM-1	-0.13
BOTTOM-2	0.22

The table 3 and 4 shows the output without implementation and table 5 gives the result after implementation of tube bundles. As a result we lowered the swirl angle created due to turbulent flow.

2.4 Graph Representation



GRAPH 1:- VELOCITY IN MM

The above graph shows the comparison of swirl angle error generated at Master Meter (MM), Refrence Meter (RM) and Meter Under Test (MUT)

III. MODELLING WORK DETAIL

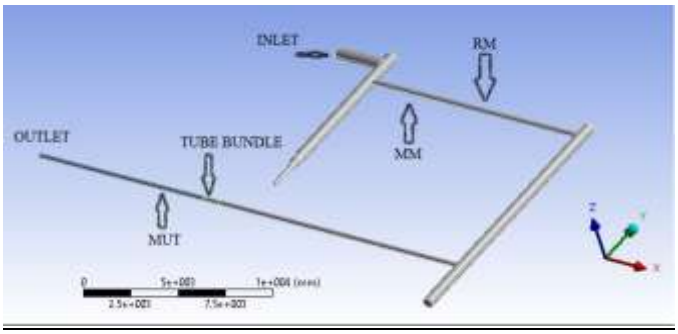


FIG. 1:- MODELING WORK FOR 10”X600

3.1 Abbreviations and Acronyms

MM: Master Meter
RM: Reference Meter
MUT: Meter Under Test

3.2 Equations

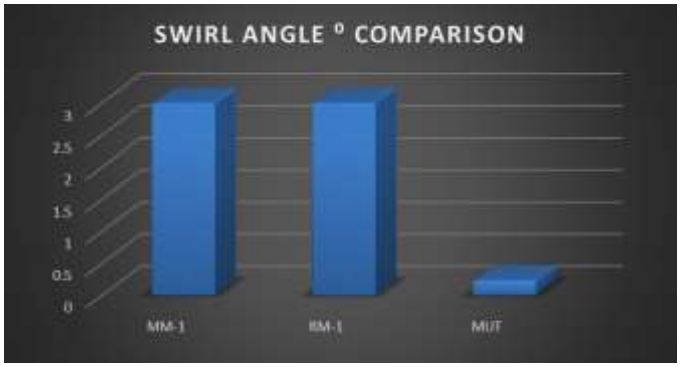
(Q1-Q2)/Q1 = Error
(Π/4)D*D = Area
Q =AV = Continuity Equation
A/a = Number of tubes

3.3 Swirl Angle Calculation

TABLE 3: SWIRL ANGLE CALCULATION

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

The table shows the different value of swirl angle calculated using CFD data.



GRAPH 2:-SWIRL ANGLE COMPARISION

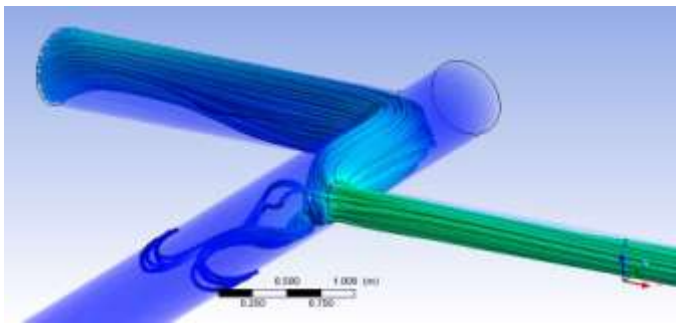


FIG. 2 :- INLET TO MASTER METER-1

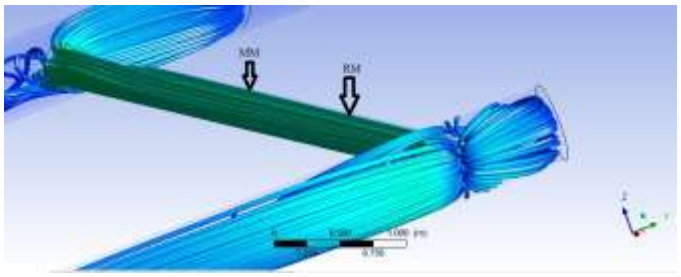


FIG.3: FLOW THROUGH M. M.-1 & REF.M.

The above figure shows the swirl generated with the help of streamline function.

CONCLUSION

Above figures show 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.

Above figure (fig. swirl point-3&4) shows that flow conditions entering at inlet, the master meter has a swirl, or velocity profile distortion (maximum at the mixing point of inlet.)

REFERENCES

[1] SIEMENS JOURNAL "CFD Post processing of industrial Gas Turbine Exhaust Diffuser & Optimization of a one dimensional Analysis Tool." Leolein P. Mouale, University of North Dakota

[2] MASTER'S THESIS "Flow Simulation of an Axisymmetric Two –Dimensional 3rd Generation DLE Burner" Simon Bruneflood Master of Science in Engineering Technology Space Engineering Lulea Engineering of technology.

[3] 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).

[4] 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.*

[5] Akay A. Islek - (Master of Science in the Woodruff School of Mechanical Engineering). "The Impact of Swirl in Turbulent Pipe Flow"- 2004.

[6] 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)

[7] 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.

[8] 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].

[9] 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.