Presenting author Biodata

Name : A. Rajiv

Designation : Asst. Engineering Manager (Instrumentation)

Company : L&T Construction, Chennai, India

Qualification : Bachelor of Engineering in Electronics and Instrumentation Engineering under Anna University, Chennai, India

Area of Expertise **:** Process field Instruments and control system

Significant Achievements:

- Working as an Instrumentation Design Engineer for Cooling Water system of ITER (International Thermonuclear Experimental Reactor) project. ITER is an international nuclear fusion research and engineering megaproject, which will be the world's largest magnetic confinement plasma physics experiment.

- Working as a designer for centralized based plant control system with new advanced technologies in the project.

- Also handling selection of field instruments for the application, sizing of annubar and orifice for the process requirement.

Number of Papers Published in Journals:

Number of Papers Published in Conferences:

Optimal approach to calibrate an annubar, for very large fluid flow

A. Rajiv
L&T construction,
Heavy Civil Infra IC, EDRC,
Nuclear Design
Chennai, INDIA,
Email: rajivar@lntecc.com

Hariram Patel
L&T construction,
Heavy Civil Infra IC, EDRC,
Nuclear Design
Chennai, INDIA,
Email: haripatel@lntecc.com

Abstract

To monitor fluid flow in specific applications and large size pipes, an *averaging Pitot tube* probe known as *Annubar***,** are preferred over the conventional flowmeter as later one is bulky and causes more energy loss with inflated flow disturbance. Motive of the present study is to infer the optimized approach to calibrate annubar which experiences large fluid flow. It is prerequisite to calibrate Annubar at test laboratory to persuade performance guarantee before installation at site. As per project specific requirement few annubars experiences very high flow rates to the extent that the available test facilities are inadequate for their simulation. Possible substitutes for above problem can be; 1st is, extract a curve from experimental data, based on it anticipate pressure drop values for large fluid flow rates. 2nd is, simulate the facility on virtual platform i.e. Computational fluid dynamics (CFD) technique and calculate the pressure drop numerically. Where first method is just a trend based anticipation and second method is a numerical approximation and may not be able to generate the expected results even at the expense of many skilled man hours. In this manuscript an optimized approach is proposed and described using a project specific problem to induce explicit solution by utilizing experimental trend and then validating the same using optimized CFD technique. Eventually a credence in the calibration results is stabilized with significant reduction in man hours, calculation time and thereby cost associated.

The study conducted here is limited to incompressible fluid i.e. water and can be explored for compressible fluid like air and other gases. Proposed calibration method can help engineers involved in this profession to deal with similar kind of problems effectively and to unfold other dimensions of the problem and inherent solutions.

1. Introduction

The annubar flow measuring instruments are light in weight, easy installation and causes less energy loss as compare to conventional flow measuring instruments and hence are more often applied in engineering practices for flow measurement. The basic principle behind of flow rate measurement by annubar is using Bernouli's principle by sensing the pressure differential across an obstruction placed in the flow path. Constructional details of a typical diamond shaped annubar and important factors affecting to annubar factor are discussed in details by V. Seshadri et al.[1]. The competency of any measuring instrument will be assured by means of well proven testing methods.

Present study is conducted to address a real engineering issue related to annubar calibration for large fluid flow which are difficult to simulate in test laboratory due to inadequate facility. To account this concern, a CFD based optimized approach is proposed to calibrate an annubar against very large water flow.

2. Problem Definition

It is prerequisite to calibrate annubar at test laboratory to persuade performance guarantee before installation at site. For small and medium range of fluid flow, it is easier to simulate the annubar calibration at test laboratory in compliance with ISO 4185.

But as the flow rate increases the task of calibration turns complicated and cost associated in the calibration setup (complying ISO 4185) of annubar for large flow rates gets amplified. In order to save cost & time without negotiating with accuracy of the calibration, a well proven simplified alternatives to be established.

3. Alternative Solution available

There can be two possible substitutes for above problem;

Method 1:- A curve can be extracted from experimental data for small/medium flow rates, based on it pressure drop values for large fluid flow rates can be projected. For example, look at the experimental data (Flow rate vs Pressure drop) of an annubar (for 1800DN pipe) calibrated in a test laboratory as shown in Table 1.

Based on experimental data shown in Table 1 irrespective to flow rates (serial number 1 to 4), prediction of pressure ΔP for flow rates (serial number 5 to 18) is presented in the graph shown below using exponential, linear and 2nd order polynomial trend. By analyzing the graph a polynomial curve can be concluded as best fit against available experimental data, however in the absence of experimental data, best fit can never be proved. It means any of the three trends plotted in figure 1 can be true curve in the absence of experimental data if no secondary validation be carried out, and hence this method of trend based prediction must not be used directly for prediction of pressure drop values for higher flow rates.

Table 1 Annubar Calibration

Sr. No.	Qa (m3/h)	V (m/sec)	ΔP_EXP (pascal)
1	11333.71	1.23	2012.80
2	11976.51	1.30	2247.10
3	12793.77	1.39	2570.10
4	12873.97	1.40	2600.40
5	13230.39	1.44	2745.90
6	13231.54	1.44	2758.40
7	13736.51	1.49	2971.40
8	13736.58	1.49	2966.80
9	13759.84	1.50	2984.70
10	14323.69	1.56	3227.10
11	14351.40	1.56	3241.50
12	15187.08	1.65	3616.50
13	15726.22	1.71	3880.00
14	15741.35	1.71	3889.80
15	16396.91	1.78	4195.20
16	16418.41	1.78	4199.40
17	17060.28	1.85	4569.70
18	17068.17	1.85	4577.00

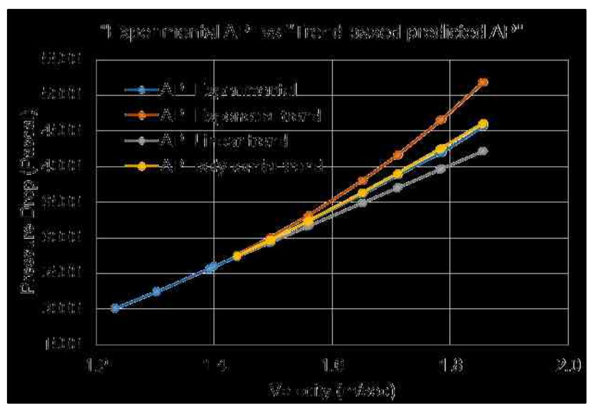

Figure 1 ”Experimental ΔP” vs “Trend predicted based ΔP”

Method 2:- Simulate the facility on virtual platform (CFD technique) and calculate the pressure drop numerically, but this method comes with following issues:

a) Modelling an annubar inside a pipe and generating an accurate mesh over it requires many skilled man hours and huge computational capabilities in search of accurate results with respect to kind of model adopted for the simulation. Adopting 3D model may produce precise results at the expense of time, cost and computational capabilities. Instead one may think that clarity of solution and accuracy of results may get negotiated if 2D model will be adopted for the simulation. Hence it needs to be addressed in order to justify with time and cost constraints of competitive industry without dropping accuracy of the solution.

b) CFD simulation is a numerical approximation and may not generate expected results for pressure drop prediction. In fact D. Wecel et al.[3], has reported that CFD code cannot calculate flows with a laminar/turbulent transition" is another reason based on which CFD results should not be used directly. To understand it in a better way, CFD simulation results of an annubar in 1800DN pipe against experimental data is presented in graph shown below (Figure 5). Almost a difference of 60% can be observed for CFD results against experimental data.

4. Offered Solution:

Issues with method-1 discussed in previous section is addressed in this manuscript by proposing a secondary validation of trend curve using "CFD based prediction" as follows.

Proposed solution for the issues with Method-2 are described here;

i). To handle the complicated geometry and meshing, the factual information available from physics of flow across annubar is to be scrutinized; As per Flow Measurement Engineering Handbook [4], annubar factor formula is;

$$[K = Q_a/F_a Y A\sqrt{2\rho\Delta P}]$$

Wherein for incompressible fluids at ambient temperature, thermal expansion factor 'F' and expansibility factor 'Y' can be taken as unity and therefore the annubar factor 'K' depends largely on shape and size of the annubar due to the phenomenon of wake and vortices formation behind the body[5].

For a fully developed turbulent flow(Figure 2) maximum velocity of fluid lies at the central plane of pipe and hence dominance of turbulence (wake and vortices formation behind the body) if any, is expected to occur at central plane of fluid at first and then it can spread over the other areas far from centerline of pipe. It means, while predicting pressure drop for high flow rates, any abnormality/divergence in the pressure drop trend can be captured on central plane in advance. Hence modelling 2D geometry (central plane of fluid inside the pipe along cross section of annubar) can be considered as conservative approach for CFD simulation as it will capture the uncertainties in advance. Also it will capture characteristics of the diamond shape of annubar consistently for the ascending velocities coming at the central plane of the fluid. Considering 2D geometry for simulation will reduce the simulation time, cost and skill level required significantly which can be justified with competitive industry requirements.

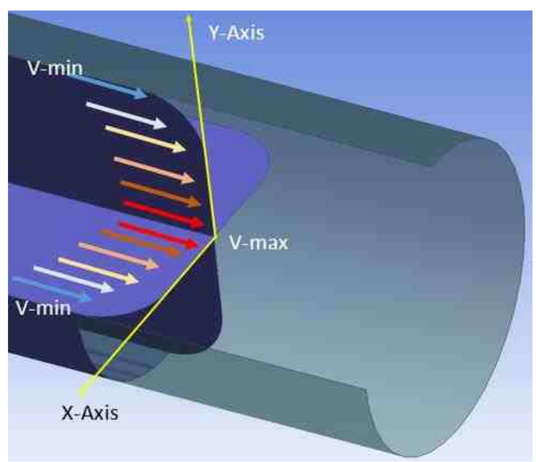

Figure 2 Velocity profile of a fluid inside a pipe

c) It is true that the CFD results are approximate and cannot meet to actual expected, however it is observed that the value of proposed non-dimensional coefficient Kp (Ratio of Experimental pressure drop(ΔP_EXP) and Numerical pressure drop(ΔP_CFD)) remains constant and can be useful for predicting actual pressure drop values corresponding to very high flow rates precisely.

5. Numerical simulation:

Annubar shown in Figure 3 is designed and manufactured by General Instruments India Private Limited to measure flow rates range from 11000 m3/hr. to 27000 m^3/hr. same is calibrated for 1800DN pipe at the test laboratory large flow laboratory (FCRI Palakkad, India) for the flow rate range from 11333 m^3/hr. to 17000 m^3/hr. (Refer Table 2), flow range above 17000 m^3/hr. falls outside the ambit of experimental facilities available and hence could not be calibrated in the laboratory. For the calibration of the same annubar for flow rates greater than 17000 m3/hr., proposed approach is applied and presented here as an example problem.

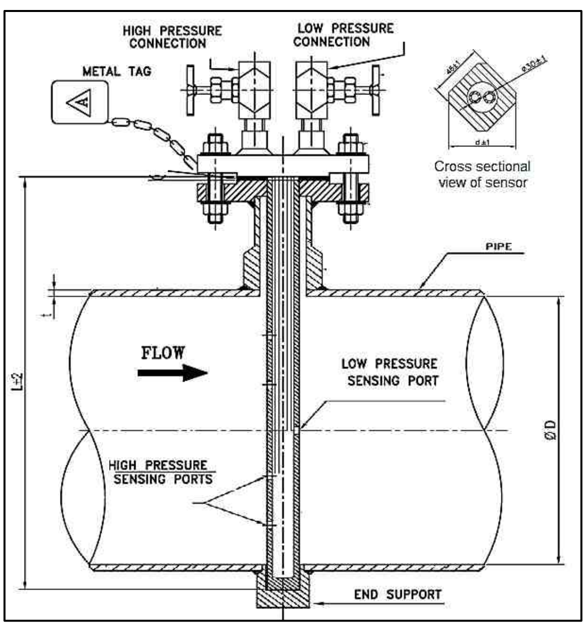

Figure 3 Typical annubar arrangement inside a pipe

For Numerical simulation of annubar in 1800DN pipe, geometry of central plane along the cross section of annubar is considered for modeling in CFD based Fluent code. Meshing adopted at pipe surface boundaries and near annubar element were sized appropriately to capture the turbulence effects. Standard K-Epsilon turbulence model is used to run the problem.

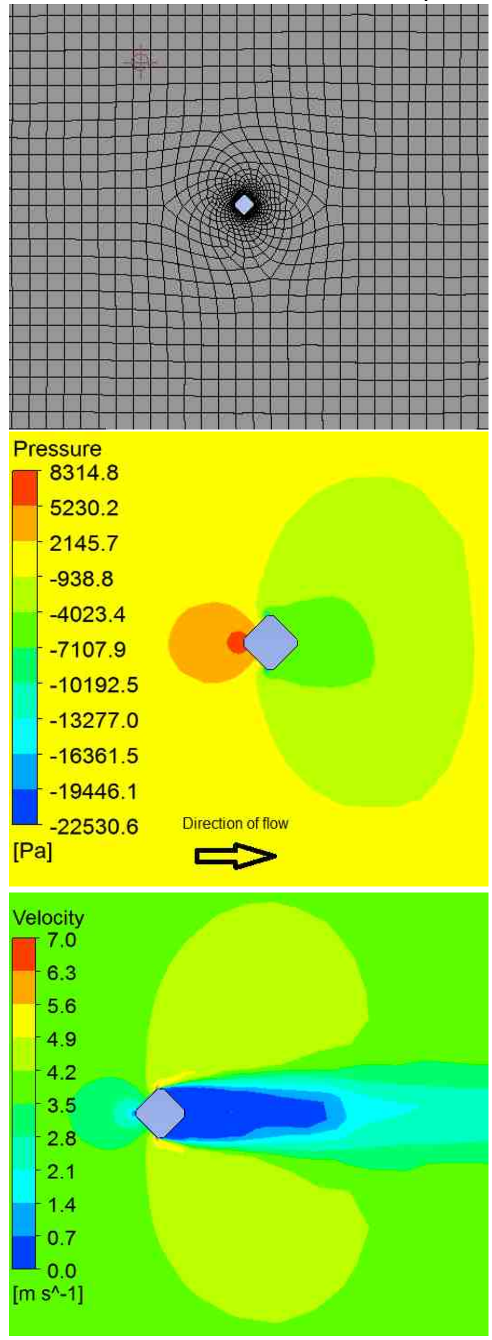

Figure 4 Typical Meshing, Pressure contour and velocity contour profile across the annubar

Typical meshing, pressure contour and velocity contour are presented through Figure 4, at first CFD simulation is carried out for the flow range 11000m^3/hr. to

17000m^3/hr. using appropriate boundary conditions, there after a new non-dimensional coefficient Kp is introduced based on the equation as shown here.

$$Kp = (K1 + K2 + K3 \ldots Kn)/n$$

$$Kn = (\Delta P\_EXPn / \Delta P\_CFDn)$$

Where, n =1, 2, 3...6.

ΔP_EXPn = *Pressure drop values from experiment*

ΔP_CFDn = *Pressure drop values from CFD simulation*

Once value of *Kp* is derived from the above formula, CFD simulation of annubar for flow rates beyond 13000 m^3/hr. up to 27000m^3/hr. is carried out. Essence of the activity is that the CFD based prediction of pressure drop values for the flow range from 13230 m^3/hr to 17000m^3/hr is overlapping the experimental curve which in turn gives us confidence on practicality of proposed approach which is used here to predict pressure drop values corresponding to very high flow rates i.e. in the range of 17000 m^3/hr to 27000m^3/hr.
Final values of predicted pressure drop for individual flow rates is been arrived from formula as shown below.

$$\Delta P\_PREm = \Delta P\_CFDm \ x \ Kp$$

Where m = 7,8,9,10……m.

ΔP_CFDm = *Pressure drop values from CFD simulation*

ΔP_PREm = *Predicted Pressure drop values*

CFD based prediction of pressure drop is tabulated and plotted below along with Trend (polynomial trend) based prediction curve, a converging behavior of both of these curves establish a credence in the results in spite of just a trend based prediction(as discussed above with respect to figure 1). Thanks to CFD we could save the cost of extra pump to be installed at testing lab to arrange the additional flow rate of 10000m^3/hr.

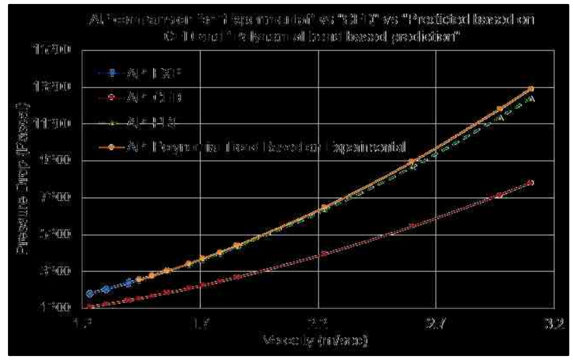

Figure 5 ΔP comparison "Experimental" vs "CFD" vs "Predicted"

Based on the predicted pressure drop values annubar factor is calculated and presented in Table 2 below.

6. Conclusion:

A project specific problem and its solution is described here to induce explicit solution by utilizing proposed approach. Eventually a credence in the calibration results is stabilized with significant reduction in man hours, calculation time and thereby cost associated. Proposed approach of annubar calibration for very high flow rates found suitable for the industrial use where it becomes impractical to arrange huge flow in laboratory, wherein it justifies with time, cost and skill constraints of competitive industry. However an annubar must be calibrated in a laboratory for lower/medium flow rates to capture manufacturing defect present, if any. Hence significance of laboratory based calibration cannot be ignored completely. The study conducted here is limited to incompressible fluid i.e. water and can be explored for compressible fluid like air and other gases. Proposed calibration method can help engineers involved in this profession to deal with similar kind of problems effectively and to unfold other dimensions of the problem and inherent solutions.

Table 2 Comparison " ΔP Experimental" vs " ΔP from CFD" vs " ΔP Predicted based on proposed approach"

Sr. No.	Qa (m3/h)	V (m/sec)	ΔP_EXP (pascal)	ΔP_CFD (pascal)	ΔP_PRE (pascal)	Kn	K__EXP or PRE	Category
1	11333.71	1.23	2012.80	1282.44	2018.35	1.570	0.6129	Data used as basis for prediction
2	11334.27	1.23	2012.70	1282.44	2018.35	1.569	0.6130	
3	11976.51	1.30	2247.10	1428.20	2247.75	1.573	0.6130	
4	11975.38	1.30	2245.70	1428.20	2247.75	1.572	0.6131	
5	12793.77	1.39	2570.10	1626.98	2560.59	1.580	0.6123	
6	12873.97	1.40	2600.40	1647.28	2592.54	1.579	0.6125	
7	13230.39	1.44	2745.90	1737.87	2735.11	-	0.6126	Data used to establish the credence in prediction
8	13231.54	1.44	2758.40	1739.57	2737.79	-	0.6113	
9	13736.51	1.49	2971.40	1872.93	2947.68	-	0.6114	
10	13759.84	1.50	2984.70	1880.73	2959.95	-	0.6111	
11	13736.58	1.49	2966.80	1874.03	2949.41	-	0.6119	
12	13736.62	1.49	2970.80	1874.03	2949.41	-	0.6115	
13	13740.85	1.49	2974.80	1874.03	2949.41	-	0.6113	
14	14323.69	1.56	3227.10	2035.69	3203.83	-	0.6118	
15	14351.40	1.56	3241.50	2043.56	3216.21	-	0.6116	
16	15181.51	1.65	3616.70	2286.80	3599.04	-	0.6125	
17	15187.08	1.65	3616.50	2286.80	3599.04	-	0.6127	
18	15726.22	1.71	3880.00	2452.03	3859.08	-	0.6126	
19	15741.35	1.71	3889.80	2454.92	3863.63	-	0.6124	
20	16396.91	1.78	4195.20	2664.76	4193.88	-	0.6142	
21	16418.41	1.79	4199.40	2670.58	4203.04	-	0.6147	
22	17060.28	1.86	4569.70	2882.80	4537.04	-	0.6123	
23	17068.17	1.86	4577.00	2885.34	4541.04	-	0.6121	
24	20481.80	2.23	NA	4147.65	6527.70	NA	0.6151	Data for which testing facility is inadequate
25	23895.44	2.60	NA	5631.86	8863.59	NA	0.6158	
26	27309.07	2.97	NA	7345.70	11560.89	NA	0.6162	
27	28512.00	3.10	NA	8004.26	12597.35	NA	0.6164	
					Average	**1.574**	**0.6129**	

Acknowledgment

We extend our gratitude to Mr. Karthik P, Engineering Manager (Instrumentation) at L&T Construction Nuclear division for his immense support.

Also we appreciate "Fluid Control Research Institute", Palakkad (Kerala, India) for extending their support in conducting laboratory based testing.

References

[1]. V. Seshadri et al. 2004, "Analysis of the effect of body shape on annubar factor using CFD" ELSEVIER Measurement 35, 25–32.

[2]. Bolesław Dobrowolski et al. 2005, "A mathematical model of the self-averaging Pitot tube a mathematical model of a flow sensor". ELSEVIER Flow Measurement and Instrumentation 16, 251–265.

[3]. Daniel Wecel et al. 2008, "Experimental and numerical investigations of the averaging Pitot tube and analysis of installation effects on the flow coefficient" ELSEVIER Flow Measurement and Instrumentation 19, 301–306.

[4]. R.W. Miller 1989, Flow Measurement Engineering Handbook, second ed., McGraw-Hill Publishing Company, New York, pp. 11.27–11.40.

[5]. H. Schlichting 1998, Boundary Layer Theory, eighth ed., Mcgraw Hill Book Company, New York.