ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

IJARMATE
Your ult-MATE Research Paper !!!

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 2, Issue 3, March 2016

CFD PROCESS – A DETAILED STUDY

Sreesha.M.[1], Seeniammal.A.[2], Reena.J.[3], Nathiya[4]

U.G. Scholars, Department of Aeronautical Engineering, PSN College of Engineering and Technology (Autonomous), Tirunelveli, India [1,2,3,4]

***Abstract*— The standard deviation of the three dimensional spout plans were contrasted with their individual 2D standard deviations to get the best outline which creates more uniform mach number of the perfect gas that turns out. The standard deviation of the spout model 3 is the slightest. Yet, contrasting with 3 dimensional spouts model 2 has huge diminishment in standard deviation, around 31% lessening. Thus it was presumed that the spout model 8 which was intended for creating Mach number 3 was the most effective spout which delivers greatest consistency at the spout exit.**

***Index Terms*—Mach number, CFD, Goad Gears, Discretization.**

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses computational techniques like numerical methods and algorithms to simulate various phenomenon associated with fluid flow. It has emerged a third approach in fluid mechanics after experimental and theoretical fluid mechanics. Advancement in computing technology and development of efficient numerical schemes has propelled the field of CFD. It has helped us simulate complex phenomenon, like turbulence, shock waves, phase transformation and heat transfer, occurring within the fluid flow. CFD involves solving various partial differential equations (PDE's) that govern the fluid flow with the ultimate goal of understanding the physical events occurring in and around fluid flow. It has wide range of applications starting from aerodynamics to biotechnology. Whatever the application might be, carrying out a CFD analysis involves following general steps.

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. CFD analysis complements testing and experimentation and it reduces the total effort required in the laboratory.

The result of CFD analyses is relevant engineering data used in

- Conceptual studies of new designs.
- Detailed product development.
- Trouble shooting.
- Redesign.

CFD - how it works

- Analysis begins with a mathematical model of a physical problem.
- Conservation of matter, momentum, and energy must be satisfied throughout the region of interest.
- Fluid properties are modelled empirically.
- Simplifying assumptions are made in order to make the problem tractable Provide appropriate initial and boundary conditions for the problem.

CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanics in the fluid region of interest. The solution is post-processed to extract quantities of interest like temperature distribution, heat transfer, pressure loss, etc.

II. CFD PROCESS

There are essentially three stages to every CFD simulation process:

- Pre-processing
- Solving
- Post Processing.

The framework of computational fluid dynamics analysis is shown in Fig.1.

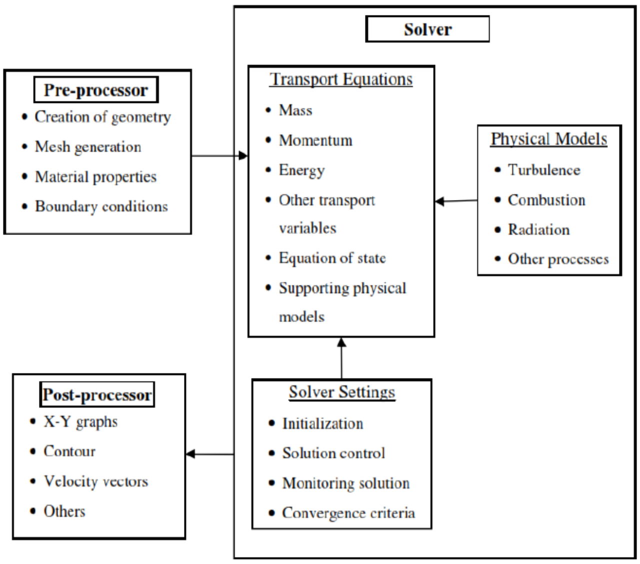

Fig.1. CFD Analysis Framework

Preprocessing is the first step in building and analyzing a flow model. It includes building the model within

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 2, Issue 3, March 2016

a computer-aided design (CAD) package, creating and applying a suitable computational mesh, and entering the flow boundary conditions and fluid materials properties.

CAD geometries are easily imported and adapted for CFD solutions in ICEM CFD, Hypermesh, Gambit etc. 3D solid modelling options in hypermesh allow for straight forward geometry construction as well as high quality geometry translation. HyperMesh is a high-performance finite element pre-processor to prepare even the largest models, starting from import of CAD geometry to exporting an analysis run for various disciplines. HyperMesh enables engineers to receive high quality meshes with maximum accuracy in the shortest time possible. A complete set of geometry editing tools helps to efficiently prepare CAD models for the meshing process. Meshing algorithms for shell and solid elements provide full level of control, or can be used in automatic mode.

The CFD solver does the flow calculations and produces the results. We provide four general-purpose products: FLUENT, CFX, CFD-POST and POLYFLOW. FLUENT is used in most industries. POLYFLOW are also used in a wide range of fields, with emphasis on the materials processing industries.

The FLUENT CFD code has extensive interactivity, so you can make changes to the analysis at any time during the process. This saves you time and enables you to refine your designs more efficiently. Our graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modelling process faster. It is also easy to customize physics and interface functions to your specific needs. In addition, FLUENT's adaptive and dynamic mesh capability is unique among CFD vendors and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow.

The FLUENT solver has repeatedly proven to be fast and reliable for a wide range of CFD applications. The speed to solution is faster because our suite of software enables you to stay within one interface from geometry building through the solution process, to post processing and final output. FLUENT's performance has been tried and proven on a variety of multi-platform clusters. Our parallel computing capability is flexible and enables you to solve larger problems faster.

Post Processing is the final step in CFD analysis, and it involves the organization and interpretation of the predicted flow data and the production of CFD images and animations. All of Fluent's software products include full post processing capabilities. Our post processing tools enable you to provide several levels of reporting, so you can satisfy the needs and interests of all the stakeholders in your design process. Quantitative data analysis can be as sophisticated as you require. High-resolution images and animations help you to tell your story in a quick and impactful manner.

Fluent's CFD data exports to third-party postprocessors and visualization tools such as Insight, Field view and Tech Plot. In addition, FLUENT CFD solutions are easily coupled with structural codes such as ABAQUS, MSC and ANSYS, as well as to other engineering process simulation tools.

The governing equations are based on the conservation of mass, momentum and energy. The conservation equations are related to the rate of change in the amount of that property within an arbitrary control volume to the rate of transport across the control volume surface and the rate of the production within that volume. The Navier-Stokes equations for compressible flows are examples of governing equations.

Continuity equation: describes the conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V}\right) = 0$$

Linear Momentum Conservation:

$$\frac{\partial\left(\rho \vec{V}\right)}{\partial t} + \nabla \cdot \left(\rho \vec{V}\vec{V} - \underline{\underline{T}}\right) = \rho \vec{b}$$

Energy Conservation in total energy form:

$$\frac{\partial (e_o \rho)}{\partial t} + \nabla \cdot (e_o \rho \vec{V} - \underline{\underline{T}} \cdot \vec{V} + \dot{\vec{q}}_c + \dot{\vec{q}}_r) = \rho(\vec{V} \cdot \vec{b} + \dot{\vec{q}})$$

Where: ρ is the fluid density, is the flow velocity, T is the stress tensor, is the body forces, is the internal energy, is the conduction heat transfer flux and is the radiation heat transfer flux and is the heat source.

Most flows of engineering significance are turbulent in nature. Flow structure in the turbulent regime is characterized by random, three-dimensional motion of fluid particles in addition to the mean motion, which is macroscopic mixing of fluid particles from adjacent fluid layers.

The Reynolds averaged Navier-Stokes (RANS) equations are a mathematical model of turbulent flow that introduces additional terms in the governing equations that need to be modelled in order to include the turbulence effects. The RANS equations govern the transport of the averaged flow quantities, with the whole range of the scales of turbulence being modelled. The RANS-based modelling approach therefore greatly reduces the required computational effort and resources and is widely adopted for practical engineering applications.

Among available CFD models, the RANS approach commonly based on turbulent kinetic energy (k) closure schemes is used for engineering applications. It is increasingly used in simulations of flow. The most widely used RANS models are two equation models, which solve two transport equations [10]. The k-ε model and its variants are the best known among these models, which require the solutions of (k) equation and dissipation rate equation (ε) The k-ω model and its variants, where (ω) is the specific dissipation rate are also very used. In this project the standard k-ε model is used.

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

IJARMATE
Your ult-MATE Research Paper !!!

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 2, Issue 3, March 2016

III. METHODOLOGY

Methodology which is used in CFD is shown in the below (Fig 3.2):

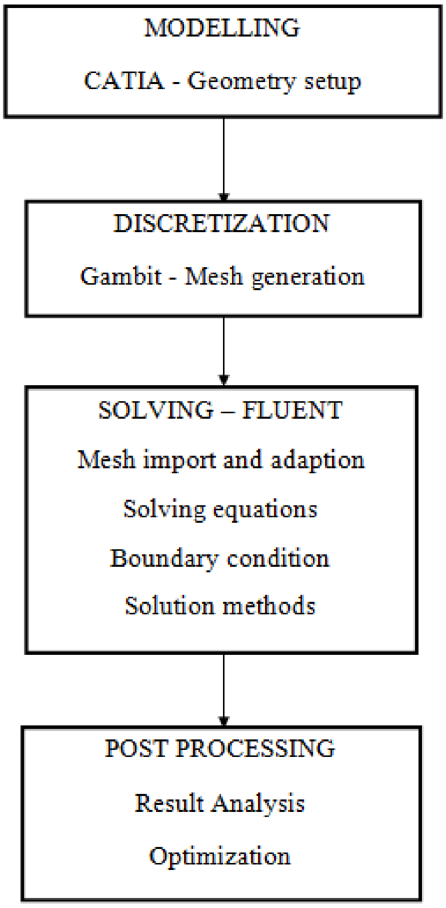

Fig.2. Methodology

The below table gives the real design inputs (Dimensions) of the 12 nozzles that have been designed (Table I).

TABLE I

NOZZLE DESIGN DIMENSIONS

SHAPE	X INLET	X EXIT	INLET RADIUS	EXIT RADIUS	INLET ANGLE	EXIT ANGLE
1	9.5	10	1.5	1.7	-3	4
2	4.0	4.9	1.5	1.7	-7	8
3	2.5	3.2	1.5	1.7	-11	12
4	2.0	2.6	1.5	1.7	-14	15
5	13.3	46	1.7	4.2	-3	4
6	5.7	23	1.7	4.2	-7	8
7	3.6	15	1.7	4.2	-11	12
8	2.8	11.9	1.7	4.2	-14	15
9	19.0	139	2.0	10.7	-3	4
10	8.1	69	2.0	10.7	-7	8
11	5.1	46	2.0	10.7	-11	12
12	4.0	36	2.0	10.7	-14	15

REFERENCES

[1]Anandraja Perumal, "Computational Analysis of De Laval Nozzle"

[2] Mohan Kumar G, Dominic Xavier Fernando, R. Muthu Kumar (2013), "Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation"

[3] M.A.Fathima, M.Gnana Soundarya, M.L.Jothi Alphonsa Sundari, B.Gayathri, Praghash.K., Christo Ananth, "Fully Automatic Vehicle For Multipurpose Applications", International Journal of Advanced Research in Biology, Ecology, Science and Technology (IJARBEST), Vol 1,Special Issue 2,November 2015, pp: 8- 12.

[4] M.A.Fathima, M.Gnana Soundarya, M.L.Jothi Alphonsa Sundari, B.Gayathri, Praghash.K., Christo Ananth, "Fully Automatic Vehicle For Multipurpose Applications", International Journal of Advanced Research in Biology, Ecology, Science and Technology (IJARBEST), Vol 1,Special Issue 2,November 2015, pp: 8- 12.

[5] G.R.Nagpal. "Power Plant Engineering" Khanna Publishers, Delhi.

[6] T.Thiyagarajan. "Fundamentals Of Electrical And Electronics Engineering" Scitech Publishers Chennai.

[7] R.S.Khurmi, J.K. Gupta "A Text Book Of Machine Desine" S.Chand & Company Ltd, Delhi.