International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 12 Dec 2021, pp: 1421-1423 www.ijaem.net ISSN: 2395-5252

Computational Fluid Dynamics of Turbulent Flow

K. D. Manwar[1], D. S. Pattebahadur[2]

[1](Assistant Professor, Civil Department, Jawaharlal Darda Institute and Technology, Yavatmal, India)
[2](Assistant Professor, Civil Department, Jawaharlal Darda Institute and Technology, Yavatmal, India)

Date of Submission: 15-12-2021 Revised: 27-12-2021 Date of Acceptance: 30-12-2021

ABSTRACT: This paper deal with the variety of application of CFD.The objective of CFDsolve and validate Partial Differential Equations (PDEs) and discretize PDEs into an algebra problem and obtain a model that can predict quantities of interest, such as fluid velocity, for use in engineering designs of the system being modeled.Much software offers the possibility of solving fully nonlinear coupled equations in a production environment.CFD iterative Methodsare used because the cost of direct methods is too high and discretizationerror is larger than theaccuracy of the computer arithmetic.

KEYWORDS –computational fluid dynamics,jet flow, turbulence modeling, turbulent flow

I. INTRODUCTION

Over the past few decades, Computational Fluid Dynamics (CFD) has emerged as a powerful tool for the design and optimization of new products and processes. It is widely used in a variety of applications and industries such as chemical, petroleum, aerospace, automotive, polymer processing, medical research, construction, meteorology, and so forth. Some of the key offerings include the design reliability and reduction in the time and cost of the product development.A significant research is being carried out in a quest to model the physics accurately and to develop robust and efficient numeric (solution algorithms and accurate discretization methods).The interaction between the CFD and disciplines like solid mechanics and optimization algorithm is also evolving.

II. LITERATURE REVIEW

Khatim et al performedCFD study for cross flow heat exchanger with integral finned tube.The objective of the present work is to simulate the 3-D geometry for cross flow smooth and finned tube heat exchanger with using hot water inside the tube and cooling air outside the tube by using computational fluid dynamic (ANSYS-FLUENT-15). The fundamental basis of most of CFD problem are the solution of (mass, momentum and energy) equation.

Patil et al design, analysis of flow characteristics of exhaust system and effect of back pressure on engine performance. This paper deals with the exhaust system designed and through CFD (Fluent) analysis, a compromise between two parameters namely, more maximization of brake thermal efficiency with limited back pressure. The analysis has been carried out on two designs an existing one that is EDS – I with 0° inlet cone angle and a modified one that is EDS – I with 90° inlet cone angle, results are subsequently compared. With appropriate boundary condition and fluid properties, Boundary conditions used at inlets mass flow rates and Temperatures of Fluid are applied and at outlets pressure outlet is applied. Domain surface is used as a wall with 'No Slip condition' and heat transfer coefficient of 45w/m2 ºk and wall surface roughness as 0.00508 mm is used. The conclusions may be drawn from the present study. The Exhaust system is successfully designed. Through CFD analysis, the backpressures of various Exhaust diffuser systems are studied. The increase in inlet cone angle increases the pressure of the flow which leads to reduce the recirculation zones. Installation of the EDS – II increases the brake thermal efficiency and decreases the backpressure.

III. OBJECTIVE

The objective of CFD is to model the continuous fluids with Partial Differential Equations (PDEs) and discretize PDEs into an algebra problem, solve it, validate it and achieve simulation based design.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 12 Dec 2021, pp: 1421-1423 www.ijaem.net ISSN: 2395-5252

IV. BASIC EQUATION OF CFD

Incompressible Navier-Strokes Equations	
Mass:	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$
Momentum:	$\rho\frac{Du}{Dt}+\frac{\partial p}{\partial x}=\mu\nabla^2 u$
	$\rho\frac{Dv}{Dt}+\frac{\partial p}{\partial y}=\mu\nabla^2 v$
	$\rho\frac{Dw}{Dt}+\frac{\partial p}{\partial z}=\mu\nabla^2 w$
Energy:	$\rho\frac{De}{Dt}=\frac{\partial Q}{\partial t}+k\nabla^2 T+\tau_{ij}\frac{\partial u_i}{\partial x_j}$

V. TURBULENCE MODELING

In computational modeling of turbulent flows, one common objective is to obtain a model that can predict quantities of interest, such as fluid velocity, for use in engineering designs of the system being modeled. For turbulent flows, the range of length scales and complexity of phenomena involved in turbulence make most modeling approaches prohibitively expensive; the resolution required to resolve all scales involved in turbulence is beyond what is computationally possible. The primary approach in such cases is to create numerical models to approximate unresolved phenomena. This section lists some commonly used computational models for turbulent flows.

Turbulence Models Available in FLUENT
One-Equation Models
Spalart-Allmaras
Two-Equation Models
Standard k–ε
RNG k–ε
Realizable k–ε
Standard k–ω
SST k–ω
Direct Numerical Simulation (DNS)
Large Eddy Simulation

VI. APPLICATION OF CFD

Industrial application: CFD is used in wide variety of disciplines and industries, including aerospace, automotive, power generation, chemical manufacturing, polymer processing, petroleum exploration, pulp and paper operation, medical research, meteorology, and astrophysics.

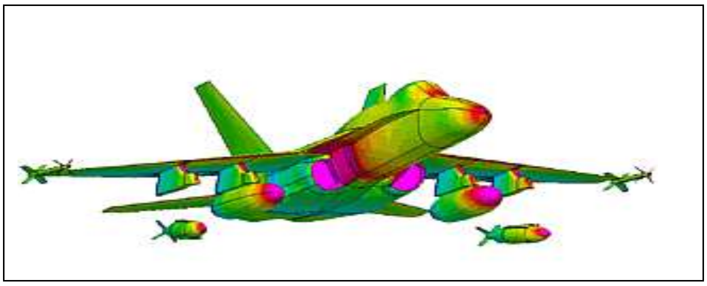

fig. 1 plane-body interaction

Two dimensional transfer chute analyses using a continuum method: Fluent is used in chute designing task like predicting flow shape, stream velocity, wear index and location of flow recirculation zones.

Bio-medical engineering: The following figure shows pressure contours and a cutaway view that reveals velocity vector in a blood pump that assumes the role of heat in open-heart surgery.

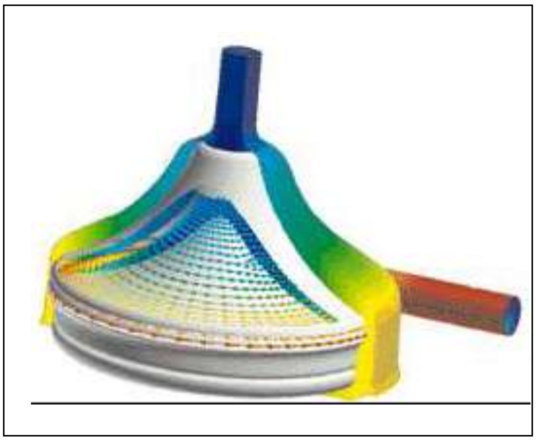

fig.2 pressure contours in blood pump

Blast Interaction with a Generic Ship Hull: The figure shows the interaction of an explosion with a generic ship hull. The structure was modeled with quadrilateral shell elements and the fluid as a mixture of high explosives and air. The structural elements were assumed to fail once the average strain in an element exceeded 60 percent.

Automotive Applications: Influence of the rear center and B-pillar ventilation on the rear passenger comfort is assessed. The streamlines marking the rear center and B-pillar ventilation jets are colored in red. With the rear center and B-pillar ventilation, the rear passengers are passed by more cool air. In the system without rear center and B-pillar ventilation, the upper part of the body, in particular chest and belly is too warm.

VII. CONCLUSION

Nearer the conditions of the experiment to those which concern the user, more closely the predictions agree with those data, the greater is the reliance which can be prudently placed on the predictions.CFD iterative Methods like Jacobi and Gauss-Seidel Method are used because the cost of direct methods is too high and discretization error is larger than the accuracy of the computer arithmetic.Much software offers the possibility of solving fully nonlinear coupled equations in a production environment. In the future we can have a multidisciplinary, database linked framework accessed from anywhere on demand simulations with unprecedented detail and realism carried out in fast succession so that designers and engineers anywhere in the world can discuss and analyze new ideas and first principles driven virtual reality.

VIII. AKNOWLEDGEMENTS

We express our profound gratitude to our head of department Prof. R.N.Pantawane and Principal Dr. R.S.Tatwawadi, for significance and helpful guidance during completion work. We also thankful to all my teaching and non-teaching faculties of department.

REFERENCES

[1] Zena K. Kadhim, Muna S. Kassim, Adel Y. Abdul Hassan, CFD study for cross flow heat exchanger with integral finned tube, International Journal of Scientific and Research Publications,6(6),2016, 668-677.

[2] Mahesh T. Dhotre, Nandkishor Krishnarao Nere, Sreepriya Vedantam, Mandar Tabib, Advances in Computational Fluid Dynamics,International Journal of Chemical Engineering, Volume 2013.

[3] Atul A. Patil, L.G. Navale, V.S. Patil, Design, Analysis of Flow Characteristics of Exhaust System and Effect of Back Pressure on Engine Performance, International Journal of Engineering, Business and Enterprise Applications (IJEBEA),(2014), 99-103.

[4] Abubaker E.M Elbalsohi, Junling Hu, Ruoxu Jia, Simulation of Turbulent Flow in an Asymmetric Air Diffuser, ASEE 2014 Zone I Conference, April 3-5, 2014, University of Bridgeport, Bridgpeort, CT, USA.

[5] Suhas V. Patankar, Numerical Heat Transfer And Fluid Flow(McGraw-Hill Publishing Company)

[6] David C. Wilcox, Turbulence Modeling for CFD, (DCW Industries Inc., California, USA, Second Edition)

[7] Fluent 5 User's Guide Vol. 2,July 1998.