Advancement in Computational Fluid Dynamics

Mr. Suraj R. Kudale[1] Prof. R. K .Bawane[2]
[2]Guide
[1,2]Department of Mechanical Engineering
[1,2]PCET'S Pimpri Chinchwad College of Engineering And Research, Pune, India

Abstract— As we all know that the increasing digitization has lead to do all the things by computers. From washing clothes to writing papers in all things we need the help of computers. In fluid mechanics for small applications is easy but for bigger applications like design of car, plane etc. The conventional numerical methods become to lengthy. Hence a new branch is introduced which helps us to slove this fluid mechanics equations to easily. This branch is known as computational fluid dynamics. In CFD basically computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. Thus, we can just put the values of boundary conditions and get all the related results in this method. The scope in the subject is very huge. And various developments related to the same are yet to be seen.

Key words: Computational Fluid Dynamics (CFD)

I. Introduction

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary condition with highspeed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels. In addition, previously performed analytical or empirical analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as flight tests.

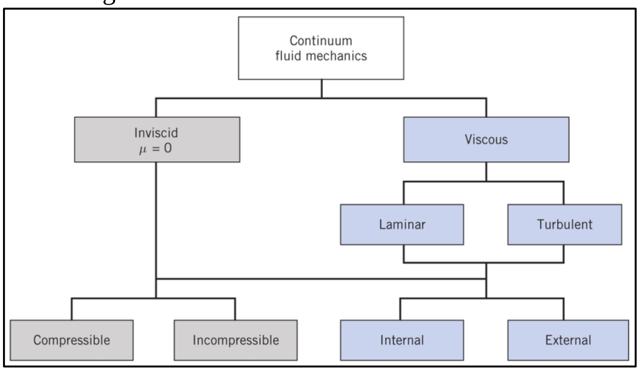

Fig. 1.1: Basic components of fluid mechanics

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering and fluid flows, and engine and combustion analysis.

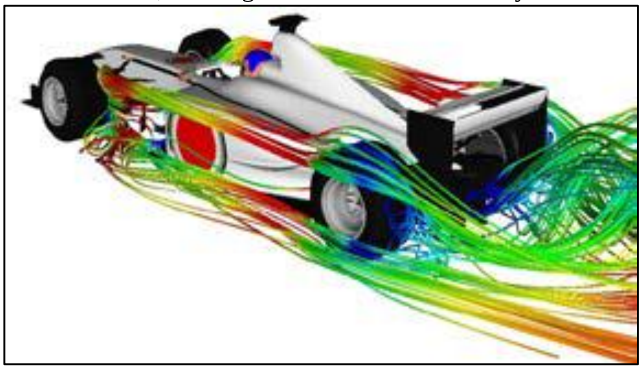

Fig. 1.2: CFD analysis model

II. Stages of CFD:

A. During Initial Stage of Processing:

1) The geometry (physical bounds) of the problem defined.
2) The volume occupied by the fluid is divided into the small discrete cells.
3) Physical modeling in terms of all the parameters (i.e. kinematic, thermodynamic, heat transfer etc.) (Motion equation+ enthalpy + radiation + species conversion).

B. Processing Stage:

1) Boundary conditions are defined involving specified fluid behaviour and properties of boundaries of the problem are defined.
2) The simulation is started and the equations are solved iteratively as steady state or the transient state.

C. Pre-screening and Preparation:

1) After identifying the list of problems and recognizing them, they must be screened and put them into the decreasing order of profit and increasing order of investment as well as difficulty of breaking technology. This process is carried out to collect technical data, maintenance data and Performance specifications.
2) Technical data helps to generate in the stages including dimensions of parts to be analyzed, surface finish interfaces, tolerances and performance and testing quality assurance requirements. However engineering drawings of the models are prepared for the prototypes and drafted with the help of different software exiting in CAD (i.e. COMPUTER AIDED DESIGN).

D. Numerical Simulation:

1) System-level CFD problems (Includes all components in the product)
2) Component or detail-level problems (Identifies the issues in a specific component or a sub- component)

3) Different tools for the level of analysis Coupled physics (fluid-structure interactions).

E. Significance of topic

1) The flaws in the model can be detected
2) Detail analysis of fluid forces on vehicles can be done.
3) Unwanted drag forces can be reduced.
4) The efficiency of vehicle can be increased.

III. LITERATURE REVIEW

A. A.S. Adkine (2016) (ICGTETM):

Computational Fluid Dynamics is a powerful way of modeling fluid flow, heat transfer, and related processes for a wide range of important scientific and engineering problems. A fully integrated numerical method for flutter analysis with a coupled fluid structure interaction is presented. The technique replaces a hands-on process guided by experience to yield accurate and reliable low fidelity models. The Computational fluid dynamics tool, ANSYS has been used to analyze the engine mounting bracket. The results obtained from the static structural and modal analysis shows that ERW-steel is better than ERW-1 steel. From the results it can be said that the ERW-2 steel is safe for the required application.

B. Thabet (2018) (IJSRD):

The aim of this assignment is to perform a mathematical and CFD analysis on the flow over Ahmed Body and compared the numerical results with the available experimental data. The study will be carried out using both Solidworks and ANSYS software to create the geometry and obtain the numerical solution, respectively.

C. Bin Xia & Da-Wen Sun (2002) (FRCFT):

This paper reviews the application of CFD in the food processing industry. CFD can be used as a tool to predict food processes as well as to design food processing equipment. There has been considerable growth in the development and application of CFD recently in the area of drying, sterilisation, mixing and refrigeration. However, the simulation results should be validated by experiments because CFD use many approximate models as well as a few assumptions. Although there are still some obstacles such as inability in accurate simulation of large 3-D problems on an affordable computer, in particular, in large-scale sophisticated plants, the trend of widespread application of CFD in the food processing industry will continue in the 21st century.

D. J Abhinesh (2014) (IJME):

In the process of redesigning, exterior styling with improved aerodynamics of existing intercity bus plying on Indian roads, a detailed computational analysis has been done. The Two prototype bus body has been modeled for performing numerical analysis using CFD software. Model No.1 is the existing Volvo intercity bus model and Model No.2 is that we altered and modified the existing model. Velocity given to the fluent analysis is 100kmph. The resultant drag force of the baseline model is 0.8 and the modified model we get is 0.7. By these modifications the coefficient of drag is reduced by approximately 10%. We reduced the drag force, results in increased performance of the bus and reduced fuel requirement.

E. Vivek V. Kulkarni (2015) (IRJET):

The entire paper presented Computational Fluid Dynamics as a tool for different research cases and real time problem solving. The range of problems encountered or tackled is from air flow simulation around buildings for planning cities to engine related or heat transfer problems. After studying the cases and reviewing literature, suitable remarks were made. As it is seen, it can be concluded that the wide variety of application of CFD is commendable, the on-going and previously done research compliment each other through validation and from the case studies it is seen that validation is satisfactory and is in close agreement with the experimental results. Hence CFD as a tool for simulation can be considered reliable for research works, or specific problem solving

IV. METHODOLOGY

Broadly following methodology will be followed to complete this seminar

1) Defining the reason for seminar topic: As now-a-days everything is getting computerized, this topic has its potential and importance in field of design.
2) Hypothesis: Predictive statement of performance was made in order to thoroughly study the topic in required direction.
3) Conceptual design: Before finalization the work process concept map was developed according to the availability of instruments and time.
4) Execution of experimental work: Various papers and information sources were referred as well practical process was tried to understand using vedio references.
5) Data analysis and developing mathematical modeling: Thus using various sources the data was analyzed and mathematical model was also developed.
6) Result conclusion: Various data, result tables among the sources were analyzed and result is drawn.
7) Future scope: Based on the conclusions, future work recommendations will be made.
8) Report Writing: Finally the complete seminar was be reported in the form of report.
9) Research Work: A systematic research work will be published in a journal.

V. RESULT & CONCLUSION

A. Static Pressure Plot

In this we discussed on the analyzed results that is static pressure distribution on the vehicle, drag coefficient and velocity vector flow pattern for each case respectively.

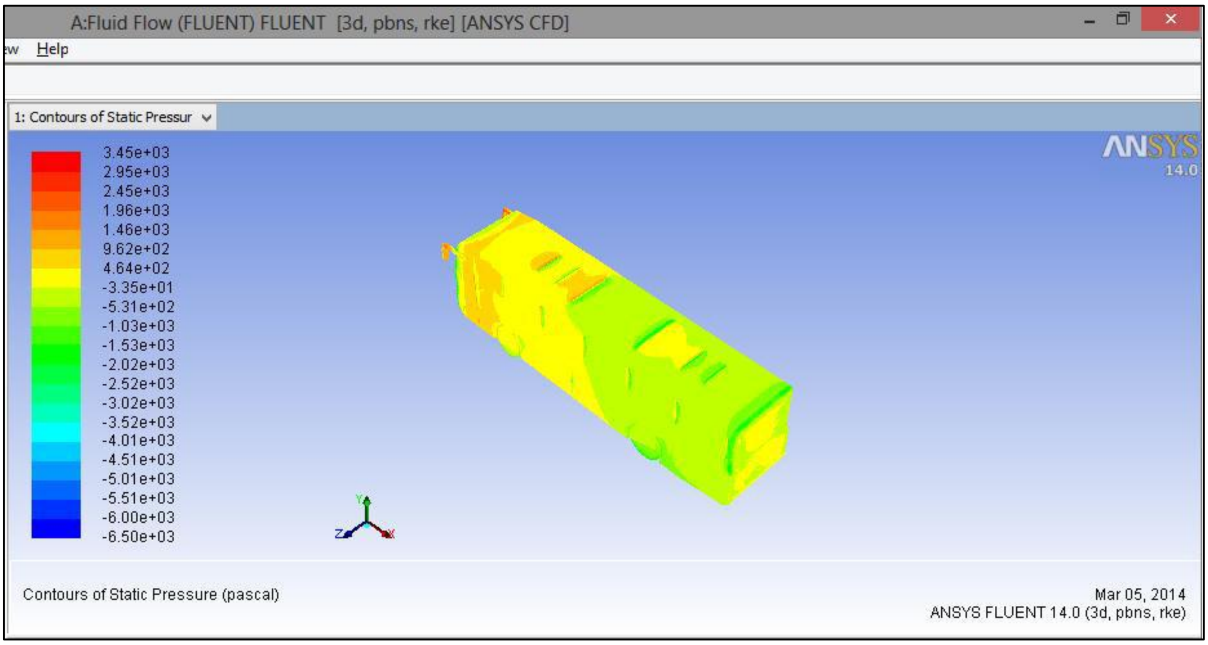

Fig. 4.1: Static pressure plot

B. Drag Coefficient

From the analysis we have found the force in the X axis direction which is nothing but the drag force. Upon substituting the drag force value we can find the drag coefficient (Cd) value respectively. Cd = 0.5* drag force/ ρAV2

Outputs	Values
Coefficient of Lift	0.03686
Coefficient of Drag	0.32346
Coefficient of Moments	0.35953

Table 4.1: Coefficient values

C. Alternative Material for Engine Mounting Bracket:

ANSYS software was used for stress analysis and results are tabulated in Table-4.1

Parameters	ERW- 2	Aluminum alloy	ERW-1
Von-Mises stress (max)(MPa)	82.96	64.588	68.87
Total deformation (mm)	1.086	3.88	1.90

Table 4.2: Stress Distribution among ERW2, Aluminum and ERW-1

This shows that the defining alternative material engine mounting bracket, in which aluminum alloy were studied along with ERW-1 and ERW- 2 steel. After analyzing the results, it can be anticipated that ERW-2 can be proffered over Aluminum and ERW-1.

VI. Conclusion

1) Computational Fluid Dynamics is a powerful way of modeling fluid flow, heat transfer, and related processes for a wide range of important scientific and engineering problems.
2) A fully integrated numerical method for flutter analysis as well as for determining drag coefficient.
3) The technique replaces a hands-on process guided by experience to yield accurate and reliable low fidelity models.

Acknowledgement

I have great pleasure in submitting the seminar Report on the topic, "Advancement in Computational Fluid Dynamics"

It gives me immense pleasure to record my debt of gratitude and my warmest regards to Pimpri Chinchwad College of Engineering & Research.

I would like to say thank to my guide, Prof. R. K. Bawane, Mechanical Engineering Department, Pimpri Chinchwad College of Engineering, Pune.

Also I would like to say thanks to Prof. Dr. Sham Mankar, (HOD) Mechanical Department.

References

[1] Sachin Thorat and G Amba Prasad Rao (1999), "Conducted a Research on Computational Analysis of Intercity Bus with Improved Asthetics and Aerodynamic Performance on Indian Roads".
[2] Edwin J Saltzman, Robert R Meyer, Mc Callen and K Salari (2007), "International Papers Carried Out Studies on Reducing the Drag of Trucks and Buses".
[3] Peterbilt (2009), "Motors Company Presents a White Paper on Heavy Vehicle Aerodynamics and Fuel Efficiency".
[4] Mc Callen (2004), "Investigated on a Reduction in Drag Value Until the Front Leading Edge".
[5] G Buresti, G V Lungo and G Lombard (2007), "Carried out a Research on Methods for the Drag Reduction of Bluff Bodies and Their Application to Heavy Road".
[6] Panu Sainio, Kimmo Killstrom and Matti Juhala (2007), "From Aalto University Conducted a Research on Aerodynamics Possibilities for Heavy Vehicles".
[7] E Selvakumar, M K Murthi and A Muthuvel (2013), "Conducted Research on Aerodynamic Exterior Body Design of Bus".

[8] P.Wesseling, Principles of Computational fluid dynamics, Springer link (2001).John W. Slater, CFD analysis NASA OFFICIAL, Tuesday 17, 2008.
[9] ZHANG Baoqiang, CHEN Guoping a, GUO Qintao, "Static Frame Model Validation with Small Samples Solution Using Improved Kernel Density Estimation and Confidence Level Method" (2012) Chinese Journal of Aeronautics, vol. 25 pp 879-886.
[10] ZHANG Man, FU Zhenbo, LIN Yuzhen, LI Jibao"CFD Study of NOxEmissions in a Model Commercial Aircraft Engine Combustor" (2012), Chinese Journal of Aeronautics vol. 25 pp 854-863.
[11] Gareth A. Taylor b, Michael Hughes, Nadia Strusevich, KoulisPericleous"Finite volume methods applied to the computational modelling of welding phenomena" (2002) Applied Mathematical Modelling vol.26 pp 309–320.