Review on Computational Fluid Dynamics (CFD)

PRATYUSHA GAJBHIYE[1], CHANDAN VERMA[2], SHILPA VINCHURKAR[3]
[1, 2] *PG Student, Department of Mechanical Engineering, G.H.R.C.E, Nagpur, Maharashtra, India*
[3] *Assistant professor, Department of Mechanical Engineering, G.H.R.C.E, Nagpur, Maharashtra, India*

Abstract - In this article we provide a an overview on computational fluid dynamics (CFD) applications. The basic principles of CFD are reviewed and are discussed with some examples of their application. The development of specialized tools such as SOLIDWORKS and ANSYS has significantly expanded the application domain of simulation, especially computational fluid dynamics (CFD) in engineering. Analysis is a high level of computational fluid dynamics in general engineering and specific areas (thermal machinery, electrical appliances and systems, electronics and assemblies in industrial and building ventilation, turbomachinery, automotive and agricultural machinery, greenhouse industry, etc.).

Indexed Terms- CFD, CAD Software, etc.

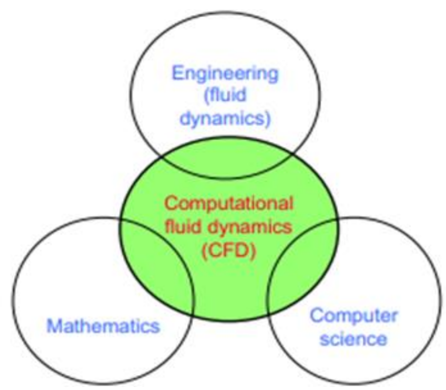
I. INTRODUCTION

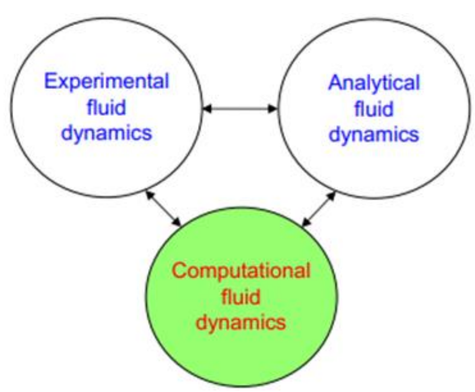
The development of advanced CAD systems has made it possible to expand the fields of application of simulation technology, especially computational fluid dynamics (CFD). This method constitutes one of the branches of Mechanics of Fluids that uses numeric methods and algorithms to solve and to analyse problems on the flow of fluids. The fact that CFD was developed later than other CADs used for thermal stress analysis is due to the inherent difficulties encountered in the equations that describe fluid flow. Today, with the great potential of computers and the development of numerical solutions, CFDs have become very practical and efficient tools for analyzing flow situations, resulting in invaluable analytical tools and designs. Will be. Computers are used to perform the millions of calculations required to simulate the interaction of liquids and gases with complex surfaces projected by engineering. This means that CFD should not be considered a tool used by highly qualified professionals in universities and research centers, it is already in use in numerous industries. Using simplified equations and high-performance computers often yields only approximate results. However, ongoing research allows you to integrate software that speeds up computations and reduces margins of error, while at the same time allowing you to analyze increasingly complex situations as transonic fluids and turbulent flows.

II. WHAT IS COMPUTATIONAL FLUID DYNAMICS?

It is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. The different disciplines contained (i.e., Engineering fluid dynamics, Mathematics and Computer science) within computational fluid dynamics. As shown in figure.

III. APPROACH OF COMPUTATIONAL FLUID DYNAMICS





CFD has also become one of three basic methods or approaches that can be used to solve fluid dynamics and heat transfer problems. As demonstrated in figure, each approach is strongly interlinked to each other and does not lie in isolation. Traditionally, both experimental and analytical methods have been used to study different aspects of fluid mechanics and help engineering design equipment and industrial processes involving fluid flow and heat transfer.

IV. LITERATURE REVIEW

1. Bert Blocken, Ted Stathopoulos, Jan Carmeliet. "CFD simulation of the atmospheric boundary layer: wall function problems". In this paper by using CFD simulation he analyse the wall surface. wall surface has a rough and smooth surface.
2. H. P. A. Calis, J. Nijenhuis, B. C. Paikert, F. M. Dautzenberg, C. M. van den Bleek. "CFD modelling and experimental validation of pressure drop and flow profile in a novel structured catalytic reactor packing". in this paper author is study that how flowing fluid behave between the two surfaces

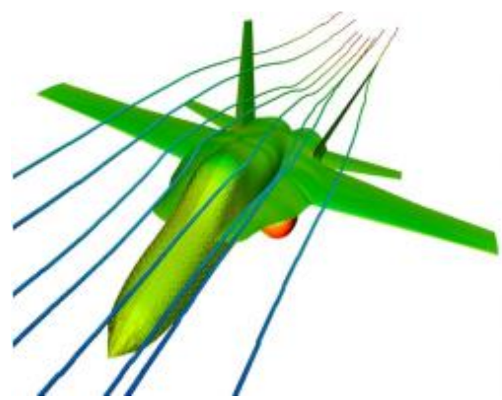
At different pressure.

V. ADVANTAGES OF COMPUTATIONAL FLUID DYNAMICS

1. Construction and solution of the governing equations and the study of various approximations to these equations.
2. CFD complements experimental and analytical approaches by providing an alternative cost-effective means of simulating real fluid flows.
3. Reduce design and manufacturing lead times and costs compared to experimental approaches.
4. CFDs can simulate flow conditions that cannot be reproduced by experimental tests of geophysical and biological fluid dynamics, such as: nuclear accident scenario, or a scenario that is too large or too far to be experimentally simulated (eg, the 2004 Indonesian tsunami).
5. CFDs can provide comprehensive information that is fairly detailed and visualized.

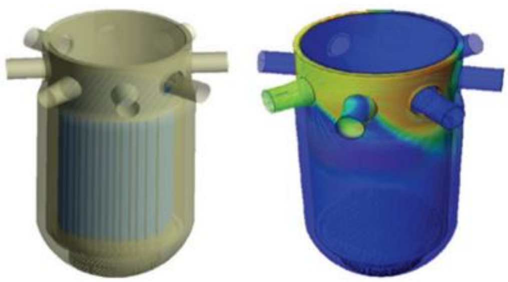
VI. APPLICATION OF COMPUTATIONAL FLUID DYNAMICS

1. As a research tool: CFDs give you a better understanding of the physical events or processes that occur when fluid flows around and inside a given object.. Similar to wind tunnel testing, CFD can be used as a research tool for conducting numerical experiments.
2. As a design tool: Engineering design and analysis environment in major industries. CFD for optimizing product development and processes, and / or predicting the performance of new designs before manufacturing or implementation. As a viable design tool, CFD has helped provide important and substantive insights into the equipment and flow characteristics in processes needed to increase production, improve lifespan, and reduce waste. Increasing the processing power of computers will undoubtedly revolutionize the use of CFD in new and existing industries.
3. Aerospace: CFD certainly has a long and illustrious history of development and application in the aerospace and defense industries. The simulation of fluid path lines in the vicinity of an F18 jet (left) and prediction of pressure coefficient contours at a 10-degree angle of attack around a supersonic missile system with grid fins (right) are illustrated in figure.



These examples are just a few of the many applications in aerodynamic design and military applications. Efforts to better understand and mitigate the noise generated by the safe operation of heavy guns and military helicopters when launching missiles that can affect airframes and tail rotors are increasingly being used by CFDs in military applications. There are some other operational issues that we are addressing. CFD strictly meets a wide range of physical modeling requirements for studying complex phenomena associated with aerospace and defense-related designs.

4. Nuclear safety: The need for more accurate computational models for relevant safety analyses of nuclear facilities has sparked an escalating interest in CFD to feasibly predict several important flow phenomena. Figure illustrates the case of a pump start-up due to a strong impulse-driven flow at the inlet nozzle where the horizontal part of the flow dominates in the downcomer in a pressurized water reactor.



VII. APPLICATION OF COMPUTATIONAL FLUID DYNAMICS

1. We are witnessing the renaissance of computer simulation technology in many industrial applications.
2. State-of-the-art models for simulating complex fluid problems such as flash fire, buoyancy fire, polyphase and / or multi-component flow, are increasingly available, especially with the availability of commercial multipurpose CFD computer programs. It has been applied.
3. In addition, significant advances in virtual technology and electronic reporting allow engineers to quickly view and challenge CFD forecasts and make the necessary assessments and decisions for a particular engineering design.
4. Ultimately, the industry will move to zero prototype engineering for new product development as CFD becomes a part of the design process.

CONCLUSION

The great potential and versatility of CFDs that make up tools that can solve very complex problems such as pressure prediction, velocity profiling, and aerodynamic research, especially in general engineering and multiple applications in agricultural engineering. CFD is a valuable tool for analysis and simulation to improve the design of various machines and plants. Of note is its application in determining the effects of wind on the airflow of agricultural atomizers and in greenhouse climate studies.

REFERENCES

[1] Computational fluid dynamics the basic with application by John David Anderson.

[2] Computational fluid dynamics a Practical approach by Jiyuan Tu, Guan-Heng Yeah, Chaoqun Liu.

[3] https://en.wikipedia.org/wiki/Computational_fluid_dynamics

[4] Bert Blocken, Ted Stathopoulos, Jan Carmeliet. “CFD simulation of the atmospheric boundary layer: wall function problems”. Atmospheric Environment 41(2): 238-252. © Elsevier 2007.

[5] H. P. A. Calis, J. Nijenhuis, B. C. Paikert, F. M. Dautzenberg, C. M. van den Bleek. “CFD modelling and experimental validation of pressure drop and flow profile in a novel structured catalytic reactor packing”. Chemical Engineering Science 56 (2001) 1713}1720.