International Journal of Scientific Research and Engineering Development-– Volume 6 Issue 3, May-June 2023

Available at www.ijsred.com

RESEARCH ARTICLE OPEN ACCESS

Computational Fluid Dynamics analysis of Fluid application in Mechanical Engineering and Civil Engineering fields – A Review Article

Basheer Khan*, Colin Quintyn**, Antalov Jagnandan***, Shawn Jagnandan****

*(Department of Mechanical Engineering, University of Guyana, Georgetown, Guyana
Email: kbasheer899@gmail.com)

** (Department of Civil Engineering, University of Guyana, Georgetown, Guyana
Email:nativeguyanese@gmail.com)

*** (*Department of Mathematics, Physics and Statistics University of Guyana, Georgetown, Guyana*
Email: antalovjagandan@gmail.com)

**** (*Department of Mathematics, Physics and Statistics University of Guyana, Georgetown, Guyana*
Email:jagnandanshawn@gmail.com)

--************************----------------------------------

Abstract:

Computational Fluid Dynamics (CFD) is widely used in various areas of fluid mechanics in both mechanical engineering and civil engineering fields. CFD is applied to areas such as centrifugal pumps, jet engines, hydraulic engineering, intelligent building design with the focus on ventilation and fluid flow in pipes and design of irrigation systems. Therefore, engineers will be able to understand how a system behaves and can modify parameters of existing system in CFD which will result in predicting optimum operating conditions. This paper explored and summarizes CFD application in mechanical engineering and civil engineering fields.

Keywords **—Computation Fluid Dynamics (CFD), Fluid Mechanics, Mechanical engineering, Civil Engineering**

--************************----------------------------------

I. INTRODUCTION

Fluid mechanics fundamentally look at the behaviour of fluids at rest and in motion which applies the principles of general mechanics to liquids and gases. There are two categories of fluid mechanics, fluid statics and fluid dynamics. Fluid statics looks at fluids at rest, however it covers fluids that are moving as a whole but the elements of the fluid do not move relative to one another. Fluid dynamics is the study of fluids in motion [1].

Fluid mechanics plays a pivotal role in engineering. The natural gas, cold water and sewage systems piping are designed and calculated from fluid mechanics, it is also important for the piping of heating and air conditioning systems. In addition, fluid mechanics has many applications in vehicles such as moving the fuel to the cylinders from the fuel tank, the hydraulic brakes, automatic transmission, power steering, pumping the fuel and fuel injectors or carburettors, mixing the air and fuel in the cylinders and the combustion gases that go into the exhaust pipes are purged. Furthermore, fluid mechanics is essential in the design, calculation, testing and building of boats, submarines, aircraft, jet engines, rockets, wind turbines, the cooling of electronic parts, biomedical devices, crude oil, natural gas and transport water. Also, it is of vital importance in the design of bridges and buildings to ensure they will be able to resist damage from wind and water loading. Significantly, fluid mechanics is important in the human body. The design and manufacture of artificial hearts, the machines for breathing and, the dialysis system all use fluid dynamics principles [2]. Complex systems are analysed using Computational Fluid Dynamics (CFD) which includes fluid-gas, fluid-solid and fluid-fluid interaction. ANSYS CFX mode is used to determine the streamlines and pressure coefficients in the solid-air interaction analysis, these parameters are used in a numerical computational process. The process involves formulating the initial and final boundary conditions, establish the flow pattern, producing the mesh, modelling the system, determine simulation strategy, input the files and parameters and run the simulation [3].

II. Application of fluid mechanics in Mechanical Engineering – Centrifugal pumps

Centrifugal pumps are commonly used in various industries, the agriculture sector and, for domestic purposes, the pump system will need to move fluid over a wide flow distance depending on the need. The design of the impeller is dependent on understanding the loading operating conditions and the internal flow behaviour in the different parts of the pump. In previous research, a numerical approach was done and it focuses on the pump near design state. Insufficient studies were conducted on the off-design parameters that will cause the performance of the pump to increase or decrease [4].

Computational Fluid Dynamics (CFD) software was applied to the design of turbines and water pumps some 30 years ago. Its development aids in pump design because the intricate flow patterns through the various parts of the pump can be tested and studied at different working conditions resulting in increased performance. [5]

It began with the finite element method built into the CFD and categorized by 3 Dimensions (3D).

Quasi-simplified Euler solution and 3D flow solution potential. The complexity of CFD increased over the years from 3D Euler solution to Reynolds Average Navier – Stokes (RANS) simulation of the single blade passages utilizing finite volume methods, extend to whole machines steady simulations and recently using advanced turbulence models to solve unsteady RANS equations. The most researched area currently is looking at the effects of fluid-structure interaction and 2 phase flow. [6]

Researchers used finite volume method and a structured grid system for the equations solution to simulate the flow in the centrifugal pump through the impeller. The head capacity curve, the pressure distribution and flow patterns were all predicted using the CFD technique. The results showed that the boundary layer variables could not be investigated because of insufficient grid size. This method was applied to get a good understanding of the flow of the liquid at numerous operating positions in the pump [7]. Shah et al. [8] performed a simulation on the centrifugal pump of 200 m3/hr in steady-state conditions by applying the RANS equations. It was reported that non-uniformities exist in different pump parts for the off design conditions which led to a reduction in efficiency. Researchers investigated the pump performance by using the identical outlet diameter and changing the blade angles. The testing was done by a CFD commercial finite volume code applied to the numerical simulation of 3D and incompressible Navier-Stokes equations. It was observed that at nominal capacity, the head was improved by 6% but there was a reduction of 4.5% in hydraulic efficiency when the outlet blade angle was changed from 200 to 500. In addition, the increased blade angle coupled with high flow rates resulted in a substantial increase in the hydraulic efficiency [9].

Anagnostopoulos [10] looked at how to solve the RANS equations for a centrifugal pump impeller 3D turbulent flow simulation and decided to develop a numerical model. The model involved controlling a number of design variables for the impeller geometry which resulted in the impeller shape being modified in order to test various configurations. The outcome of this parametric testing revealed that optimizing the impeller geometry will lead to a noteworthy increase in hydraulic efficiency. Figure 1 shows the Modified shroud geometry and its effect on efficiency and head.

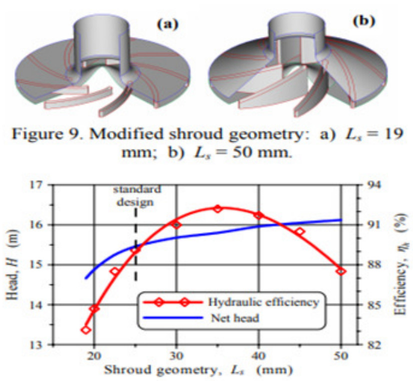

Fig. 1Modified shroud geometry and its effect on efficiency and head

A numerical study was conducted on the mixed flow pump by carefully looking at the effects of changing the hub curve profile and stator angle at part load and duty point. The results from this analysis showed that:

- The power capacity curves obtained and the head nature were similar to the standard mixed flow pump.
- A +5% range of the duty point was predicted for the pump efficiency, however, it was at the off-design conditions more variations were observed.
- It was noticed that after changing the hub curve profile and matching the stator angle the efficiency was improved by 15. Figure 2 shows the actual and predicted efficiency versus capacity curves done by CFD [11].

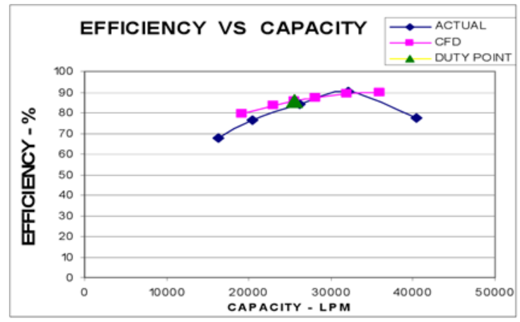

Fig. 2 Actual and Predicted efficiency versus capacity curves done by CFD

III. Application in Mechanical Engineering – CFD Analysis of Jet Engine Combustion Chamber to Diminish the Formation of NO_X

The CFD analysis revealed that at the combustor outlet there is a 38% total temperature reduction for the geometry combustor proposed concept for the jet engine. Importantly, CFD showed that there is a reduction of both NOx and CO emissions of 55% and 40% respectively.

To achieve optimizing the enthalpy and cut harmful compounds emission, it is promising to develop many alternatives to manage the changes in the dilution hole areas of the jet engine combustion chamber. Using the three- dimensional maps for the NOx and CO emissions and the outlet combustion chamber temperature can be utilized to create the best control of the dilution holes' zone variable area.

The volume of air in the primary zone can be easily controlled by managing the geometry of the combustion chamber resulting in the stoichiometric factor being regulated. Therefore, during the engine transient operating conditions, it is possible to stop the blow off of the flame meaning with a sudden increase or sudden decrease of the amount of fuel. Looking at the case of the amount of fuel suddenly reduced, a large amount of air is supplied to the combustion chamber by the gas turbine rotor which has a huge inertia [12].

IV. APPLICATION IN CIVIL ENGINEERING

CFD analysis can be used in hydraulic engineering in areas such as free surface non-stationary embankment dams flow numerical modeling, particle migration modeling and flow going through granular filter dams, research on hydropower engineering to determine the choice on the foundation elevation dam and utilized in the extracting of the municipal water system. Computational Fluid Dynamics can also be used to study irrigation systems which are made up of a pipe network used to transport water to irrigate crops. The significant parameters that needs to be considered and accurately predicted in CFD are friction losses, other losses in the pipe network and the velocities of the flow to achieve the optimal working conditions and the adequate water uniformity that is to be fed on the irrigated area. In building design and construction CFD numerical analysis is essential for intelligent buildings design and ventilation design systems for sports activities [13].

V. CONCLUSION

CFD analysis has been found to be an excellent simulation numerical tool to predict performance in centrifugal pumps at the off-design and design conditions. It is essential in the thrust axial prediction, in parametric and cavitation analysis, the interaction of all the different pump parts analysis and the analysis of the pump diffuser. From an engineering perspective, to generate a good estimation of the centrifugal pump performance apply the URANS equations along with two equations k-ε model turbulence. The use of CFD software in jet engine analysis can lead to an increase in the jet engine operation efficiency, the dilution holes' area can be facilitated as an active anti-surge system and also a lean control system blowout. In addition, the three –dimensional maps used in the combustor outlet to obtain the total temperature and the NOx and CO emissions can be applied to create the dilution holes' variable area control system. CFD is also used in civil engineering discipline to determine the fluid flow in natural ventilation, force ventilation, in the environment and in the design of smart buildings. CFD software modelling is used in hydraulic systems and structures and the study of how wind circulate around a building.

REFERENCES

[1] B. Massey and J. W. Smith, Mechanics of Fluids, 1998.

[2] Y. A. Cengel and J. F. Cimbala, Fluid Mechanics: Fundamentals and Applications, McGraw-Hill, 2014.

[3] R. Raj, A. Rukhaiyar, B. Jayant and K. Dahiya, *CFD Analysis for wind flow characteristics of varying cross section tall building using ANSYS,* Research Square , 2022.

[4] L. Hedi, K. Hatem and Z. Ridha, 2010. [Online]. Available: https://www.semanticscholar.org/paper/Numerical-Flow-Simulation-in-a-Centrifugal-Pump-Hedi-Hatem/900bec03ee7c9f2a4f00dbee59659776182e19e0.

[5] S. R. Shah, S. V. Jain, R. N. Patel and V. J. Lakera, "CFD for Centrifugal Pumps: A Review of the State-of-the-Art," *Elsevier,* vol. 51, pp. 715-720, 2013.

[6] H. Keck and M. Sick, "Thirty years of numerical flow simulation in hydraulic turbomachines," *Acta Mech,* vol. 201, pp. 211-229, 2008.

[7] M. D. Mentzos, . A. E. Filios, . D. P. Margaris and . D. G. Papanikas, "CFD Predictions of flow through a centrifugal pump impeller," in *Proceedings of International Conf.Experiments/Process/System Modelling/Simulation/Optimization*, Athens, 2005.

[8] S. Shah, S. Jain and V. Lakhera, "CFD based flow analysis of centrifugal pump," in *Proceedings of internaltion conference on fluid mechanics and fluid power*, Chennai, 2010.

[9] E. C. Bacharoudis, A. E. Filios, M. D. Mentzos and D. P. Margaris, "Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet," *The Open Mechanical Engineering Journal,* vol. 2, no. 1, pp. 75-83, 2008.

[10] J. S. Anagnostopoulous, "CFD Analysis and Design Effects in a Radial Pump Impeller," *WSEAS TRANSACTIONS ON FLUID MECHANICS,* vol. 1, no. 7, pp. 763-770, 2006.

[11] K. Patel and N. Ramakrishnan, "CFD analysis of mixed flow pump," in *International ANSYS Conference Proceedings*, 2006.

[12] S. M. Pasalkar, M. S. Shaikh and M. M. Pipulyawala, "Design and CFD Analysis of Combustion Chamber of Jet Engine to Reduce Formation of NOx," *International Research Journal of Engineering and Technology,* vol. 7, no. 6, pp. 5172-5180, 2020.

[13] A. Sultana and Q. Sultana, "Application of Computational Fluid Dynamics in Civil Engineering-A," *International Journal of Current Engineering and Technology,* no. 9, pp. 84-87, 2021.