

# An Elementary Study of Computational Fluid Dynamics For Various Engineering Applications – A Review

Krutartha Sudhir Jathar<sup>1</sup> , Vivek V.Kulkarni<sup>2</sup>

1. Undergraduate Student, Mechanical Engineering, KLS's Gogte Institute of Technology, Karnataka, India.

2. Assistant Professor, Mechanical Engineering, KLS's Gogte Institute of Technology, Karnataka, India.

\*\*\*

**Abstract:** Right from the 18th century enormous amount of research is going in the field of fluid flow and its application to everyday problems. From the invention of the computer and development in the field of fluid mechanics, its governing equations and advent of numerical methods, CFD has started and has evolved extremely. The design and the development process of any product over the years has become relatively more easy and less time consuming to the orthodox methods, so here the basic intention of this paper is to provide a history, have the importance and feasibility study of computational Fluid Dynamics (CFD) as a computational tool for various analysis of engineering related application based problems. It deals with the current scenario, overall scope of CFD, its relevance for engineering problems, test for its validation of the obtained results and its advantages over the experimental methods.

CFD as a tool can be applied for problems related to turbo-machinery, building performance for different weather conditions, storm analysis, heat exchangers, fully developed turbulent flow in pipes, analysis of fluid flow and airfoil of an airplane, cooling of electronic chips in the processor using forced convection etc.

**Keywords:** CFD, simulation, analysis, validation, turbulence models.

## 1. INTRODUCTION:

Now a days significant attention is given on improving the efficiency and minimizing the fuel consumption of engines due to the global effort of reducing Carbon-dioxide emissions. Coming to Refrigeration systems it is COP (Coefficient of Performance) that matters the most, noteworthy attempts have been made by researchers throughout the world in making these systems more reliable, more eco-friendly, optimizing to get highest output for a given input. Coming to analysis of fluid flow through pipes, it is the reduction in the inlet pumping energy that is most important for such systems to work efficiently.

In 17-18th century scientists through out relied mainly on experimental validation of a theory rather than merely the mathematics. With the advent of computers,

erudite knowledge about fields in which research took place and with development of the mathematics for the theory, a new field which was the unification of mathematical models and computers to obtain solutions for a certain type of a problem. This field has grown immensely since then. A lot of innovation in fields related to this happened in the 19th century, which gave rise to variety of subjects, diverse application fields and lot of paths to approach a given problem. With the discovery of fluid mechanics governing flow equations, numerical methods to solve differential equations in mathematics, this field took off. There are different tools to analyze different type of problems, starting with Finite Element Method (FEM) for structural based problems, Finite Volume Method (FVM) for fluid mechanics, thermal, heat transfer problems.[12]

Although, these computational methods (simulation) are virulent in the industry for design purpose, research and development (R and D), their validation might still be under question. Relevance of results depend upon the field of application analyzing its strengths, limitations and trying to optimize the errors in development of the model under study.

This paper looks into various aspects to be taken into consideration: the field of application, the design, RandD study, specific parameters of interest in the work under study. It also looks into the feasibility of application of CFD principles (methods) to the given research problem, its relevance, its outcomes and their validation with the traditional or neo-traditional methods of obtaining the results of the study. The scope of application of CFD is humongous, right from fluid flow to bio-mechanics using the relative principles, however this paper relies on a review and discussion of mainly the application of CFD principles to core mechanical engineering problems.[12]

## 2. APPLICATIONS OF CFD:

Since the advent of computational fields and ability of solving differential equations numerically, the fields in which CFD is applied has increased. This paper mainly looks at applications from broader domains like automobile, aerospace sectors with concentration in IC

Engines, Turbochargers, Heat transfer problems , aero foils and few applications in non-traditional fields.

### 3. CASE STUDIES:

1. **An-Shik Yang et.al.[1]** employed CFD as a tool in Urban and community planning to simulate, obtain flow parameters and characteristics around different buildings to obtain ventilation. Simulation domain (3kmx2kmx0.6km) was taken. A standard k-  $\epsilon$  two equation turbulence model was used.

2. **D.Bhandari et.al.[2]** carried out work in analyzing fluids(air and water) passing through a closed pipe (internal flow) using CFD software. The results obtained were for fully developed turbulent flow. GAMBIT was used for geometry modeling, Fluent for simulation. The pipe was made of steel for simulation with (length x diameter)of 8m x 0.2m. Inlet velocities of (water, air) are 0.05,1 m/s. A standard k-  $\epsilon$  model was used.

3. **S.Kandwal et.al.[3]** carried out research work for NACA 4412 airfoil, compared simulation results with work conducted by Abbott et.al. GAMBIT and Fluent was used for geometry modeling and simulation. Unstructured mesh was used. Inlet temperature and mach number of 288.17K and 0.15.

4. **Rajesh Bisane et.al.[4]** carried simulation work for 4-stroke C.I Engine and analyzed the exhaust gas system. Diameter of diffuser inlet and engine outlet=0.0254m, 0.15m. A standard k-  $\epsilon$  model was implemented, with GAMBIT for geometry creation. Inlet mass flow for conventional, turbocharged, supercharged was 0.00749,0.0115,0.014 kg/s at 562K,686K,637K respectively. Outlet opening pressure= 1.325,2.89,2.94 bar at 353K,669K,573K respectively.

5. **Rajesh Khatri et.al.[5]** conducted work on analysis of laminar flow over a flat plate. A flat plate 400mm long was maintained at a constant temperature of 333K. The fluid was passed over the plate at a velocity of 2m/s at 300K.

6. **Pulkit Agarwal et.al.[6]** carried out work for heat Transfer from Fins of an air cooled motorcycle engine under varying climates. Fluent was used for simulation. Speed range was 40-72 kmph. The engine was modelled as aluminium cylinder with fins on outer surface and a stroke volume of 150 to 187 cm<sup>3</sup> . Model creation was done using GAMBIT. A fine mesh was created near the fins. A face mesh was done by quad element. The volume was meshed by hex element. A fixed pressure of 101.325kPa was set as BC. Top and bottom surfaces were specified as adiabatic walls and the flow was kept

from left to right. Temperature was specified at the inner surface. Metallic fin was meshed and specified as solid region.

7. **Mukesh Didwania et.al.[7]** carried out work on analysis of heat transfer through two different shaped fins. Air was facilitated by the blower at a certain suitable velocity based on Reynolds number. SOLID WORKS was used to create the geometry model. It had three parts Solid Base, Solid Fin Surface and rectangular duct. All were created independently and assembled. Mesh was generated by ANSYS and it included Tetrahedral, Wedges, Pyramids, Hexahedral and Polyhedral mesh. The bottom wall was kept at a constant temperature of 430 K. No slip condition was applied leading to a zero velocity. The side walls were adiabatic. The inlet air temperature was 280 K. Simulation was done with FLUENT. A standard k-  $\epsilon$  model was employed.

8. **Arularasan R et.al.[8]** conducted work on analysis of a heat sink for cooling of electronic devices to select an optimal heat sink design, studies on the fluid flow and heat transfer characteristics of a parallel plate heat sink was done. Fluent was used for simulation. Parameters involved were fin height, fin thickness, base height and fin pitch which ranged from 16-48mm, 0.8-1.6mm, 4-12mm, 1.5-4mm respectively. A condition of heat input at 100W was assumed at the heat sink base.

9. **Vinod M. Angadi et.al.[9]** conducted work on analysis of heat transfer enhancement of car radiator using nano-fluid as a coolant. The analysis was done using STAR CCM+. Aluminium oxide was used as an additive. Flow rate changed from 2-6 litre/ min . Nano particle volumetric concentration varied from 1% to 6% of base fluid. Fin geometry was modelled on STAR CCM+ with Tube length, space between tubes, thickness, length as 31,1.5,0.3,2 cm. Number of fins used =34. Fin material used : aluminium. Temperature was taken in the range of 1000K-5000K. A 313K was the static temperature taken as initial conditions. Inlet temperature=50 degrees , convective heat transfer coefficient=50 W/m<sup>2</sup>K.

10. **Ravi shankar P R et.al.[10]** carried out work on supercritical aerofoil at different angle of attack with a simple aerofoil. NACA SC(02) 0714 and NACA 4412 aerofoil profiles were used for flow analysis. GAMBIT for geometry creation and FLUENT for simulation. The analysis was done at a Mach number of 0.6. A standard k-  $\epsilon$  model was used. Boundary conditions were defined with properties of fluid being density, viscosity, thermal conductivity and specific heat of air as 1.185 kg/m<sup>3</sup> , 0.0000183 kg/ms , 0.0261 W/mK ,1.004 kJ/kgK respectively. Pressure of 101325 Pa and a velocity of 250 m/s was set. Table shows foil parameters-

**Table 1- Parameters**

Parameters	Thickness	Camber	Lower Flatness	L.E radius	Stall angle	Zero lift angle
NACA SC(02) 0714	13.9%	1.5%	9.4%	2.9%	4.5	-5
NACA 4112	12%	4%	76.1%	1.7%		

**Table 2- Parameters**

Parameters	L.E radius	Stall angle (degrees)	Zero lift angle (degrees)
NACA SC(02) 0714	2.9%	4.5	-5
NACA 4112	1.7%	6	4

**4.RESULTS, DISCUSSION AND REMARKS:**

1.An-Shik Yang et.al. results were- It showed that due to presence of two high rise building and a low height building in between, the wind velocity was at 1m/s. A better ventilation at 1.5-2m/s was obtained eliminating a high rise building.

**Remarks-** Can be implemented with k- $\omega$  or SST models to obtain further accuracy and perform a comparative study.

2. D.Bhandari et.al. results-

**Table 3- Result**

RESULT	Simulation	Experimental	% error w.r.t exp.
Centerline velocity(Air,Water)	1.19,0.061(m/s)	1.22,0.06122 (m/s)	2.45 %; 0.359 %
Skin friction coefficient(air,water)	0.01, 0.009	0.00795,0.01	-25.4% ; 10%

**Remarks-** A k- $\omega$  model is better for fully developed turbulent, it can implemented to obtain more fine results. CFX is less time consuming and utilize less space for same meshing. Type of meshing is not specified, that will decide what package to be used for better results and convergence.

3. S.Kandwal et.al. results-

**Table 4- Result**

RESULT	Simulation	Experimental	% error w.r.t exp.
Coefficient of lift and drag	0.654, 0.001	0.649, 0.007	-0.77% ; 85.1%

**Remarks-** SST model works better for airfoils. Also research could be conducted by simulating using CFX package for unstructured mesh type as it works better.

RESULT	Conventional		Turbocharged		Supercharged	
Location	Inlet	Outlet	Inlet	Outlet	Inlet	Outlet
Gauge Pressure (bar)	1.337	1.82	7.58	1.36	8.13	1.30
Velocity( m/s)	4.439	3.172	2.69	0.56	1.73	1.24
Temperature(K)	563	444	702	658	632	558

**Remarks-** SST model could be used for better results, as it is better for engine analysis. It takes into account k- $\omega$  at inner boundary layer and k- $\epsilon$  at outer boundary layer.

5. Rajesh Khatri et.al. results-

**Table 6- Result**

RESULT	ANALYTICAL	SIMULATION	% error w.r.t exp.
Heat transfer coefficient(W/m <sup>2</sup> K)	8.77	9.28	5.49 %
Heat Flux (W/m <sup>2</sup> )	115.76	121.81	4.96 %

It was observed that boundary layer thickness was maximum for Reynolds number 10,000 and minimum for Reynolds number 50,000 It was observed that the variation of nusselt number was linear till the Reynolds number increased to 5876. The CFD results showed a 5.5% error with the analytical solutions, indicating reliability of the CFD code.

**Remarks-** The type of turbulence model used is not defined. SST model works good for such problems.

6.Pulkit Agarwal et.al. results- - It was observed that with increase in temperature on the fin surface , increasing atmospheric temperature which resulted from decrease in heat transfer due to less temperature gradient. It was also noted that the heat lost at same vehicle speed increased with decrease in atmospheric temperature. With constant temperature, heat transfer increased with velocity.

**Remarks-** Type of turbulence model is not clearly defined. A standard k- $\epsilon$  turbulence model is used for building simulation, but its seen that k- $\epsilon$  SST model produces better results.

7. Mukesh Didwania et.al. results-

**Table 7- Result**

RESULT	Rectangular fin	Circular fin
Heat transfer rate (W)	-406.93	-397.85
Pressure loss (Pa)	0.091872931	0.091870584
Increase in temperature (K)	10.1492	10.15789

It was concluded that Circular fin was optimum fin for maximum heat transfer.

8. Arularasan R et.al. results-

By keeping a minimum amount of fins, a maximum possible fin pitch could be maintained so that the pressure drop would be minimum and air flow would be maximum. Because of the manufacturability and flow velocity or flow bypass constrains, decrease in fin thickness was not feasible. It was found that for a low thermal resistance and low pressure drop in the selected heat sink model, the geometric parameters like the fin height, fin thickness, base height and fin pitch were found to be optimal at 48 mm, 1.6 mm, 8 mm and 4mm respectively.

**Remarks-** Type of turbulence model is not clearly defined. A standard  $k-\epsilon$  turbulence model is used for building simulation, but its seen that  $k-\epsilon$  SST model produces better results. Also a more fine mesh has to be used for accurate results.

9. Vinod M. Angadi et.al. results-

It was observed that with increase in the fluid flow rate and by keeping the base fluid constant without adding any nano-particles, the heat transfer coefficient values kept increasing. Also when nano fluids in certain fractions were added to the base fluid and the flow rates were kept constant , heat transfer coefficient increased. With increase in the value of temperature the heat transfer coefficient value increased. It was suggested to keep a higher magnitude of fluid flow rate clubbed with a higher volumetric percentage of nano- particles additives, which would ensure an enhanced heat transfer.

10. Ravi shankar P R et.al. results-

**Results-** In between angle of attack from 0-15 degrees the drag pressure for supercritical aerofoil was less compared to simple aerofoil , at 15 degrees drag pressure for supercritical aerofoil was least and at 30 degrees the drag pressure for supercritical aerofoil was more than the simple aerofoil. The velocity decrease in the flow field in supercritical aerofoil was less when compared to simple aerofoil.

**Remarks-**  $k-\epsilon$  SST model produces better results. As it takes care of free stream and near wall conditions.

From the case studies and literature survey, it was observed that Computational Fluid Dynamics provides

the convenience of being able to switch off , turn on specific terms of governing equations, conduct the analysis for different conditions, obtain results, start a comparative study, suggest the best method viable etc. This permits the testing of theoretical models , suggesting new paths for theoretical explorations, also provides a platform to test theories which could never possibly have been viable through experimentation. Thus, CFD provides a few major advantages when compared with experimental fluid dynamics:

Lead time in design and development is reduced significantly along with a significant savings in allocation of equipments for experimentation. It can simulate flow conditions which are un-paralleled , not reproducible in experimental model test. It can recondition the parameters and get different outputs and validate. CFD provides more closer, detailed and comprehensive information which can accommodate lot of parameters and make it as close to real life situation as possible. The most prominent feature of it is its cost effective feature compared to experimental fluid dynamics or wind tunnel testing and in a way directly addresses global energy consumption by consuming less power and being highly efficient.

**5. CONCLUSION:**

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.

**ACKNOWLEDGEMENT:**

Author Krutartha Sudhir Jathar, thanks V.V. Kulkarni for guiding and helping in maintaining the flow of work. The author also thanks the Management of KLS's Gogte Institute of Technology, Belgaum (Karnataka), INDIA, for their constant encouragement and unconditional support towards this paper.

**REFERENCES:**

[1] An-Shik Yang, Chih-Yung Wen, Yu-Chou Wu, Yu-Hsuan Juan, and Ying-Ming Su, 'Wind Field Analysis for a High-rise Residential Building Layout in Danhai, Taiwan',

Proceedings of the World Congress on Engineering 2013  
Vol II, WCE 2013, July 3 - 5, 2013, London, U.K.

[2] D.Bhandari, Dr.Singh,' Analysis of fully developed turbulent flow in a pipe using computational fluid dynamics', International Journal of Engineering Research and Technology (IJERT) Vol. 1 Issue 5, July - 2012 ISSN: 2278-0181.

[3] S.Kandwal, Dr.S.Singh, 'CFD study of fluid flow and aerodynamic forces on an airfoil.', International Journal of Engineering Research and Technology (IJERT) Vol. 1 Issue 7, September - 2012 ISSN: 2278-0181.

[4] Rajesh Bisane, Dhananjay Katpatal, 'Experimental investigation and cfd analysis of an single cylinder four stroke c.i. engine exhaust system', IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308.

[5] Rajesh Khatri, Pankaj Agrawal, Mohan Gupta, Jitendra verma, ' Laminar flow analysis over a flat plate by computational fluid dynamics', International Journal of Advances in Engineering and Technology, May 2012. ©IJAET ISSN: 2231-1963- Vol. 3, Issue 2, pp. 756-764.

[6]Pulkit Agarwal, Mayur Shrikhande and P. Srinivasan,'Heat Transfer Simulation by CFD from Fins of an Air Cooled Motorcycle Engine under Varying Climatic Conditions', Proceedings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, 2011, London, U.K.

[7] Mukesh Didwania, Gopal Krishan, Ravikant, 'Study and Analysis of Heat Transfer through Two Different Shape Fins using CFD Tool', International Journal of IT, Engineering and Applied Sciences Research (IJIEASR) ISSN: 2319-4413 Volume 2, No. 4, April 2013.

[8] Arularasan R.and Velraj R. 'CFD analysis in a heat sink for cooling of electronic devices', International Journal of The Computer, the Internet and Management Vol. 16.No.3 (September-December, 2008) pp 1-11.

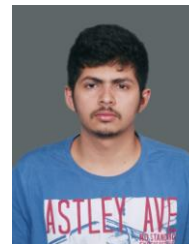
[9] Vinod M. Angadi, R.Nagaraj, Dr.O.D.Hebbal,' CFD analysis of heat transfer enhancement of a car radiator using nanofluid as a coolant', International Journal of Engineering Research and Technology (IJERT) Vol. 3 Issue 8, August - 2014 ISSN: 2278-0181.

[10] Ravi Shankar P R, H.K.Amarnath, Omprakash D. Hebbal, ' Simulations of supercritical aerofoil at different angle of attack with a simple aerofoil using fluent', International Journal of Engineering Research and Technology (IJERT) Vol. 3 Issue 8, August - 2014 ISSN: 2278-0181.

[11] J.O.Ban, U.S Lee, B.H.Ahn, Y.S.Kin, 'Application of Computational Fluid Dynamics to Rotary Compressor Efficiency and Noise', International Compressor Engineering Conference.

[12] Prashant Bhatt, Rajesh Satankar, ' Computational Fluid Dynamics analysis of wind turbine rotor blades- a review', IJCRR Vol 04 issue 21.

## BIOGRAPHIES



### **Krutartha Sudhir Jathar**

Undergraduate Student  
Department of Mechanical Engineering.  
KLS's Gogte Institute of Technology, Belgaum-590008  
Karnataka, (India).



### **Vivek V.Kulkarni**

Assistant Professor  
Department of Mechanical Engineering.  
KLS's Gogte Institute of Technology, Belgaum- 590008  
Karnataka(India).