ADVANCED TOOL FOR FLUID DYNAMICS-
CFD AND ITS APPLICATIONS IN
AUTOMOTIVE, AERODYNAMICS AND
MACHINE INDUSTRY

N. Bhagat
LeLogix Design Solutions Pvt. Ltd., Greater Noida

Shashi Kant and Amit Tiwari
LeLogix Design Solutions Pvt. Ltd., Greater Noida

ABSTRACT

Today Automotive, Aerospace and Machine industry is striving for better Efficiency and Design. Advanced tools like Computation Fluid Dynamics (CFD) may be used for improving the fuel efficiency of these and hence controlling the atmospheric air pollution. In this paper, CFD analysis software is used a) to study fluid flow and detect the cavitation in centrifugal pump to find out safe operating conditions b) to find out effect of front shape to improve drag coefficient of a car. The results of the simulation shows, how CFD can be used to study flow distribution, pressure loss, thermal distribution (cooling and climate control) in the field of Automotive, Aerospace and Machine industries.

Key words: CFD, Fluent, Drag Coefficient.


1. INTRODUCTION

The aim of this paper is to create awareness about new technologies available in the field of Fluid Mechanics and show its applications in Machinery and Automotive. This paper also aims to make this new technologies interesting for the students so that they are motivated to make use of it.
ABOUT CFD
Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena.

CFD is used in all stages of the design process:

- Conceptual studies of new designs
- Detailed product development
- Troubleshooting
- Redesign

CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation.

Following are some of the areas, where CFD is being used

- HVAC
- Automobile
- Food Processing
- Marine
- Aerospace
- Electronics

Advantages of CFD
With the availability of high speed computer, CFD has changed from high level mathematics to an essential tool in almost every branch of fluid mechanics. Now CFD results can give reliable, more confident and consistent results. Some of the advantages are listed below

- It provides a detailed understanding of flow distribution, mass and heat transfer, particulate separation etc. Consequently, all these will give plant managers a much better and deeper understanding of what is happening in a particular process or system.
- It makes possible to evaluate geometric changes and answer “What if” questions in much less time and cost as compared to laboratory testing.
- It has become almost mandatory in simulating conditions, where it is not possible to take detailed measurements such as high temperature or dangerous environment like in an oven.

2. METHODOLOGY
Basically CFD analysis involves three major tasks called Pre-Processing, Solving and Post Processing.

Pre-Processing: All the tasks that take place before the numerical solution are called pre-processing. This includes defining the problem, creating its 3D model, meshing, and applying physical operating condition called boundary conditions.

Processing: Processing involves solving mathematical equations of fluid flow until inacceptable convergence is achieved. Usually it requires the computer to solve many thousands of equations and might take few hrs. to few days.

Post-processing: When the model has been solved, the results can be analysed both numerically and graphically. Post-processing is about visualisation either in simple 2-D to 3-D representations.
Case-1: Cavitation in Centrifugal Pump

Centrifugal pump is a machine in which mechanical energy is converted into pressure energy by means of centrifugal force acting on the fluid. Pressure energy is used for lifting fluids from lower level to higher level. It is a very good example where CFD may be used to study performance of flow, which is turbulent and three dimensional in nature, needs to be predicted, before its manufacturing and its actual use.

The purpose of this study is to simulate fluid flow and detect the cavitation in centrifugal pump.

Cavitation is a disadvantage to the pump as it lowers the performance and reduces life of centrifugal pump. Hence its analysis is a very important aspect of a centrifugal pump. Following are the step for a CFD analysis of a Pump.

There are many different software available to create 3D model. Most famous of them are SolidWorks, Creo, Inventor, CATIA, UG-NX etc. In current paper, we have used SolidWorks software from Dassault systemes to create 3D model. Figure-1 shows 3D model of the fluid in the Pump
After meshing of the model of fluid in pump, Fluent is used as solver. The boundary conditions are applied. Results like pressure diagram and velocity diagram are obtained at different operating speed by taking turbulent modelling. These pressure and velocity diagrams are checked to detect the cavitation in centrifugal pump, hence finding out safe range of operating at different flow rate and operating speed:

Pressure diagram shows energy conversion taking place in different parts of the pump. Diagram clearly shows that pressures near the hub is higher than shroud. Velocity diagram shows kinetic energy at different areas.

Fig. 3 shows results at 1500 rpm and mass flow rate 35 Kg per sec.

Fig. 4 shows results at 1800 rpm and mass flow rate 35 Kg per sec.
Fig. 5 shows results at 2100 rpm and mass flow rate of 35 Kg per sec.

Red portion in the pressure diagram shows very less pressure near the shroud and sudden pressure rise shroud to hub.

The same can be seen in the Cavitation diagram. Red areas in Cavitation diagram shows bubbles or voids created because of low pressure in the pump, which may create shock waves and harm the centrifugal pump.

With Fluent, a CFD tool, it is easy to detect the cavitation occurring at different parts of the Centrifugal pump and finding out safe range of operating conditions like rotating speed. Students from B. Tech and M. Tech may take their thesis subjects like finding out optimisation of flow rate, optimisation of blade shape to avoid cavitation.

2.2. Case-2: Drag Coefficient of Car & Jeep

Nowadays with increase in competition in automobile sector, vehicle aerodynamics plays very important role in designing the outer shape of a vehicle. Manufacturer are finding CFD analysis a better tool instead of wind tunnel testing to reduce the testing time and keep the cost of R&D low.

This paper shows comparison of Drag Coefficient between a Sedan car and a Jeep. Drag coefficient is a very important aerodynamic performance characteristic. It is a measurement of resistance of an object in any fluid like air or water etc.

In this work, SolidWorks is used to create 3D model of Sedan Car and a Jeep. To reduce the overall computational cost and time, the vehicle is modelled as symmetric. Figure 6 shows the 3D models.
SolidWorks model of Car and Jeep is imported in Ansys-Fluent. Also a block of 21000x7000x500 size is generated in SolidWorks and imported in Fluent. This block represents the wind tunnel.

Now to simulate air flow around the vehicles, 3D model of the car/jeep is subtracted from the wind tunnel block. 3D model is meshed within Fluent.
The boundary condition like inlet Air flow for both Car with the speed 40m/s is given to the fluid inside the wind tunnel. Using k-epsilon model which is turbulence model.

Fig-8 shows Mesh is generated with Ansys Workbench. Unstructured meshes with tetrahedral cells and relevance centre is fine, Advance Size function with proximity are used.

Fig no. 9 shows (Definition-According fluid Dynamics force applied per unit area. Practical Reason - we are finding this result to view how much pressure applied on front of car by air. we have more for jeep it means jeep have less efficiency compared to the sedan car, this pressure will apply in opposite direction of car moving thus will oppose to the car speed, because the sedan car have less front impact pressure which will move faster compared to the jeep.) Pressure with Mesh Element in which we can see red colour shows Maximum Value, max. Pressure for the sedan car
applied by air 1045 Pa this is on less area and for Jeep max. Pressure is 1064 Pa that is on more area comparision to Sedan Car.

Figure 9

Figure 10 Shows simmiller to Fig no. 9 Pressure Distribution but in Isometric View in which we can see red colour shows Maximum Value.

Figure 10

Fig no.11 shows (Definition-According fluid Dynamics, turbulence kinetic energy is defined as mean kinetic energy per unit mass related with eddies of object in turbulent flow. Practical Reason- this energy shows turbulance generated by the edges of body of the car. We have more turbulence kinetic energy area like as in sedan car at back porsion of body it means our car have another force back side that intensify the speed and Jeep have maximum values but turbulence kinetic energy area is less compared to sedan so Jeep have another force from back side less that less intensify the speed compared to Sedan car) turbulence kinetic energy in which we can see red colour shows Maximum Value , max. Turbulence kinetic energy for the sedan car 90.67 J/kg and for Jeep max. Turbulence kinetic energy is 211 J/kg

Figure 11
Figure 12 Shows (Velocity is defined as a vector expression of the displacement that an object or particle undergoes with respect to time. Practical Reason-if we have more velocity of air applied it apposed to speed of car more, if we have velocity of air applied is less, it apposed to speed of car is less.) Velocity with Vector (for direction of air flow) in which we can see red colour arrows shows Maximum Value, max. velocity distribution for the sedan car 57.52 m/s and for Jeep is 61.29 m/s.

![Figure 12](image12.jpg)

Figure 13 Shows (Definition-The coefficient of drag (Cd) defined as measure of the force of air resistance on an object. Practical Reason-if we have less Cd value, it offered less resistance force which makes the vehicle (Sedan car) faster than jeep as obtained below.) Coefficients of drag, in left side diagram for Sedan car and right side for Jeep. For Sedan Car Maximum Coefficient of drag Value is 0.5225 and for jeep Coefficient of drag value is near about 0.72.

![Figure 13](image13.jpg)

The iterations were carried up to the point where the change in the value of drag coefficient was found negligible. This is called solution convergence.

In case of car, drag coefficient converges at 0.5225. Same process is repeated with the jeep. In this case solutions converge at 0.75, which is quite higher as compared to the Sedan car.

3. CONCLUSION

CFD is an advanced tool for designing, where aerodynamic shape plays a very important role like in the case of Automotive, Aerospace, marine etc.

Students of Mechanical branch may take subjects like finding out drag coefficient for Wing of aeroplane, aerodynamic flow over a ship superstructure for their M. Tech and B. Tech thesis.
REFERENCES

[2] Paul D. Bates and Stuart N. Lane and Robert I. Ferguson, (2005), Computational Fluid Dynamics