



CFD Simulation in the Oil and Gas Industry

By : *Engr. Mohd Nazir bin Ramli, Grad. IEM*

A WORKING DEFINITION OF COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD - a computational technology that enables you to study the dynamics of things that flow. Using CFD, you can build a computational model that represents a system or device that you want to study. Then you apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore, CFD is a sophisticated computationally-based design and analysis technique.

CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid structure interaction and acoustics through computer modeling.

Using CFD software you can build a 'virtual prototype' of the system or device that you wish to analyse and then apply real-world physics and chemistry to the model, and the software will provide you with images and data, which predict the performance of that design.

THE BENEFITS OF CFD

There are three compelling reasons to use CFD software: insight, foresight and efficiency.

- **Insight**

If you have a device or system design which is difficult to prototype or test through experimentation, CFD analysis enables you to virtually crawl inside your design and see how it performs. There are many phenomena that you can witness through CFD, which wouldn't be visible through any other means. CFD gives you a deeper insight into your designs.

- **Foresight**

Because CFD is a tool for predicting what will happen under a given set of circumstances, it can quickly answer many 'what if?' questions. You provide a set of boundary conditions, and the software gives you outcomes. In a short time you can predict how your design will perform, and test many variations until you arrive at an optimal result. All of this can be done before physical prototyping and testing.

- **Efficiency**

The foresight you gain from CFD helps you to design better and faster, save money, meet environmental regulations and ensure industry compliance. CFD analysis leads to shorter design cycles and your products get to market faster. In addition, equipment improvements are built and installed with minimal downtime. CFD is a tool for compressing the design and development cycle allowing for rapid prototyping.

APPLICATIONS IN OIL AND GAS INDUSTRY

In oil and gas industry we can use this powerful predictive tool in many applications ranging from drilling to production and processing such as to analyse;

- Pressure contours on stator and rotor in a down-hole turbine.
- Contours of volume fraction of oil in a hydrocyclone.
- Mud flow behavior for optimum cuttings removal in drill bit design.
- Contours of gas volume fraction in a gas-liquid separator.

- Assessment of wind-driven ventilation through a platform.
- Wind-induced surface pressures on the platform.

For instance, in exploration-drill bit, CFD analysis can predict the coverage of lubricating mud over the cutting surfaces of a drill bit. The results will show a maldistribution of mud that causes uneven wear on the drill bit, thus shortening its life.

These predictions are borne out by the actual worn drill bits, which exhibit a very similar wear pattern. These results inform new designs which can be tested with further CFD simulations before being manufactured.

CFD can also be applied to a host of problems such as:

- Oil production in reservoir, including flow around downhole injectors (i.e. Coiled Tubing Fill Cleanout job).
- Drilling through complex fluids, such as mud.
- Cavitation in rotating equipment and pipe bends.
- Flows involving gas-solids, liquid-solids or liquid-liquid mixtures.
- Gas dispersion in mud, in and around plants or an offshore platform for safety analysis.

CONCLUSION

CFD helps in predicting the performance of a particular design and offers suggestion to improve the existing designs. A typical CFD simulation might involve complex geometry, which allows us to check that the model is a reasonable representation of the engineering situation and check for any errors that might have been introduced in generating the geometries.

The simulation results in data for the velocities, pressure, turbulence, temperature and other data at each of the millions of points in the mesh. This data can be used to determine important engineering parameters such as drag coefficients, heat transfer rate and maximum temperature depending on the purpose of the calculations. The numerical analysis of the CFD results will tell us what the engineering parameters of interest are, but it is the graphical illustration of the results that explains why. It is true in CFD more than anywhere else that a single graphic can replace a thousand words.

The main benefit is that numerical experiments can be performed to illustrate the effects of design and retrofit changes, without the need for a costly and time-consuming prototype program. ■

REFERENCES

- [1] Ramli, Mohd Nazir., Computational Simulation of Turbulence Convective Heat Transfer in Multiple Rectangular Duct. Thesis, UiTM, Shah Alam, Malaysia, May 2006
- [2] Versteeg H.K. and Malalasekera W., An Introduction to Computational Fluid Dynamics (The Finite Volume Method). Longman Scientific and Technical, United Kingdom, 1995.
- [3] Bradshaw P., An Introduction to Turbulence and Its Measurement. Pergamon, Oxford, 1971.
- [4] Abbot M.R. and Basco D.R., Computational Fluid Dynamics – An Introduction for Engineers. Longman, 1989.
- [5] Shaw C.T., Using Computational Fluid Dynamics. Prentice Hall, UK, 1992.
- [6] Hinze J.O., Turbulence. McGraw-Hill, New York, 1959.
- [7] Schlichting H., Boundary Layer Theory. McGraw-Hill, New York, 1968.
- [8] Launder B.E. and Spalding D.B., Mathematical Models of Turbulence. Academic Press, New York, USA, 1972.
- [9] FLUENT News, 2006