REVIEW QUESTIONS

1.1 From which industry has CFD emerged from?

The emergence of CFD came from the high-technology engineering industries of aerospace and astronautics. It's application has since been adapted for use in many engineering applications.

1.2 Which three disciplines CFD is derived from?

CFD integrates three disciplines:

- Fluid Mechanics – this involves the study of fluids at rest and in particular fluids in motion. This includes the fluid flow behaviour and flow processes such as heat and mass transfer, and chemical reactions.
- Mathematics – this involves the physics of fluid motion governed by a set of mathematical equations (partial differential equations).
- Computer Science – this involves computational techniques that are used to solve complex mathematical equations computationally.

1.3 Traditionally, CFD has been used to solve aerospace and automotive engineering applications such as drag and lift for airplanes and cars, what examples can you think of where CFD is being used within non-traditional fluid engineering applications?

Some examples of non-traditional engineering applications include:

- bio-medical engineering that deals with blood flow and respiratory flow;
- sports performance such as swimming techniques, aerodynamic cycling, and yachting/rowing boat and oar design;
- renewable power generation through wind power blade designs and heat transfer of solar power production;
- civil and environmental engineering that deals with air and water flow around infrastructures (oil rigs or high skyscrapers), coastal water movements, wind and fire simulations.

1.4 What are some of the advantages of using CFD?

Some advantages of CFD include:

- Through computational science, specific terms in the governing equations can be studied in a more detailed fashion which is difficult through analytical mathematics.
- CFD results can be used to complement experimental and analytical approaches which are a cost-effective means of simulating real fluid flows.
- The ability of simulating flow conditions that are not reproducible in experimental tests found in geophysical and biological fluid dynamics, such as nuclear accident scenarios or scenarios that are too huge or too remote to be simulated experimentally (e.g. Indonesian Tsunami of 2004)
- CFD results are detailed and provides comprehensive visual information when compared to analytical and experimental methods.

- CFD simulations can speed up experiments through virtual testing of alternate evaluated over a range of dimensionless parameters that may include the Reynolds number, Mach number, Rayleigh number and flow orientation.

.

1.5 What are the limitations and disadvantages of using CFD?

The main limitation of CFD exists in the solution of the mathematical equations that govern fluid motion by numerical methods. Numerical errors occurs which creates differences between the computed results and reality. Visualization of numerical solutions may present attractive images that appear to provide a sense of realism but there is a danger that it is an erroneous solution due to numerical errors. Hence, CFD simulations require the user to have some knowledge of the three facets of CFD (computational science, mathematics and fluid dynamics) in order to properly analyze and make critical judgment on the computed results.

1.6 What CFD measurements can be obtained to assist the designs for safety and comfort in passenger airplanes?

Data regarding the lift and drag, and the impact of trailing vortices for the safe operation of aircraft taking-off and landing on the runway can be obtained to assist designs for safety issues. The internal cabin ventilation, heating and cooling system can be simulated and measured to enhance the passenger and crew comfort.

1.7 How is CFD being used as a research tool, a design tool and an educational tool in academic fields, such as Thermal-Fluids?

CFD can be employed as:

- a research tool to perform *numerical experiments*. Comparative visualization of the numerical calculations with experimental results allows a deeper investigation into the observed flow structures and physical aspects of a flow field, similar to a real laboratory experiment, e.g. obtaining heat transfer correlations;
- a design tool, parameters can be isolated and tested to effects of different designs, e.g. design of heat exchangers;
- an educational tool that provides visualization capability enhancing the students' intuition of the flow behavior, e.g. visualization of convective flows.

1.8 How can CFD be applied and used to improve cost-effective design procedures in the automotive industry?

CFD simulations have the ability to shorten design cycles through optimization of existing engineering components and systems to improve energy efficiency and meet strict standards and specifications. CFD can be used to determine the small changes in pre-existing designs such as the effects of local geometry changes on the aerodynamic forces. This procedure reduces the dependence on time-consuming expensive clay models and wind-tunnel experiments and delivers quicker design turn-around.

1.9 The bio-medical science field is turning to CFD to resolve flows within human airway and vascular systems. What advantages does CFD hold over experiments in obtaining these numerical results?

Experimental testing for the human airway and vascular system is intrusive, difficult and expensive while ethical issues have to be considered. CFD can provide flow simulations of the human body, and also virtual prototyping to recommend the best design for surgical reconstructions such as carotid endarterectomy. This can lower the chances of post-operative complications, assist in developing better surgical procedures and deliver a good understanding of biological processes as well as more efficient and less destructive medical equipment such as blood pumps.

1.10 What details can CFD capture in the simulation of hydro-cyclones, a process commonly used in the minerals industry?

CFD can capture the separation of the solid particles from the liquid due to the imparted centrifugal forces inside the hydro-cyclone. Individual particles can be tracked throughout the system through different multiphase (here liquid and solid particles are two different phases) flow approaches.

1.11 How is CFD being used in the civil and environmental industry?

Infrastructure in the civil industry can use CFD to test the flow of fluids, especially for large water tank and pipe flow in dam designs. CFD has been used to predict the pollutant plume dispersed from a cooling tower subject to wind condition which helps to ascertain the effects of the exhaust plume on the surrounding structures. Also architectural structures exposed to environmental elements can be tested (e.g. the Itsukushima Torii (Gate) ???.

1.12 What two applications see CFD being used in the power generation industry? What kind of data is collected and how is this useful in increasing the efficiency in power generation?

Turbine blade designs for wind generation are tested through CFD. The amount of torque created under varying wind conditions and the positioning of the turbines throughout a complex terrain with steep inclines, is obtained to maximise power generation. Also simulations of combusting fluidized coal beds as well as boiler operation can provide the necessary data regarding, fuel consumption, pollutant emissions, slagging and degradation of the tubes, thus improving the efficiency of power generated through coal.

1.13 How can CFD influence the way swimmers in improving their swimming strokes?

CFD has helped to “design” the optimum stroke to achieve peak propulsive performance for elite swimmers. Investigations evaluated the flow around the hand and forearm of a swimmer during the propulsion phases of the freestyle and butterfly strokes. Steady-state lift and drag forces for evaluated at different angles of attack ranging from -15 to 195 degree to find the optimum stroke.

1.14 What competitive edge can CFD give to a cycling team?

The design of the bikes’ forks and handlebar arrangement were evaluated to see the effects of the rider’s positioning on the bike on the drag that is caused by the overall shape of the cyclist. Additionally a streamlined design for the aerodynamic helmet was tested and created to accommodate different head styles achieving the ultimate cycling efficiency.

1.15 In the future, to what extent will CFD be involved within the product development process in manufacturing?

In the future, CFD will be an integral part in many manufacturing processes. Its main advantage is the ability to cut time and costs through virtual modeling instead of prototyping. By performing time-dependent simulations every possible design aspect and engineering judgment can be consistently exercised on the spot through *real-time* assessments on proposed customized design simulations in selecting the most optimum vehicle.

1.16 What is the future of CFD?

Through the increase in computing power, CFD techniques are infiltrating more industries and different areas of study. Complex fluid mechanics problems such as jet flames, buoyant fires, multi-phase and/or multi-component flows are now being progressively applied especially through the availability of multi-purpose commercial CFD computer programs. Advancements will be made to more accurate turbulence models such as the Large Eddy Simulation (LES) which will become the preferred methodology for many turbulence investigations of fundamental fluid dynamics problems. Also more unsteady simulations with large complex geometries, such as car aerodynamics with dynamic movements like cornering and turning will be feasible in the future.