INTRODUCTION

The use of computational fluid dynamics (CFD) to predict internal and external flows has risen dramatically in the past decade. In the 1980s the solution of fluid flow problems by means of CFD was the domain of the academic, postdoctoral or post graduate researcher or the similar trained specialist with many years of the grounding in the area. The widespread availability of engineering workstations together with efficient solution algorithms and sophisticated Pre- and post-processing facilities enable the use of commercial CFD codes by graduate engineers for research, development and design tasks in industry. The codes that are now on the market maybe extremely powerful, but their operation still requires a high level of skill and understanding from the operator to obtain meaningful results in complex situations. The long learning curve, previously including apprenticeships of up to four years - more widely known as MPhill and PhD studies - meant that the users of the 1980s were, through their own experiences, very conscious of the limitations of the CFD. However, the pressure on engineers I industry to come up with solutions to problems implies that there is not always the time available for the new type of user of the 1990s to learn about the pitfalls of CFD by osmosis and frequent failure.

WHAT IS CFD?

Computational Fluid Dynamic or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. Some examples are:

- Chemical Process Engineering: mixing and separation, polymer molding…

- Environmental Engineering: distribution of pollutants and effluents…
- Biomedical Engineering: blood flows through arteries and veins…

From the 1960s onwards the aerospace industry has integrated CFD techniques into the design, R&D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engines, combustion chambers of gas turbines and furnaces. Furthermore, motor vehicle manufacturers now routinely predict drag forces, under –bonnet air flows and the in car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

There are several unique advantages of CFD over experiment-based approaches to fluid systems design:

- Substantial reduction of lead times and costs of new designs
- Ability to study systems where controlled experiments are difficult or impossible to perform (e.g. very large systems)
- Ability to study systems under hazardous conditions at and beyond their normal performance limits (e.g. safety studies and accident scenarios)
- Practically unlimited level of detail of results

The variable cost of an experiment, in terms of facility hire and/or man-hour costs, is proportional to the number of data points and the number of configurations tested. In contrast CFD codes can produce extremely large volumes of results at virtually no added expense and it is very cheap to perform parametric studies, for instance to optimize equipment performance.