COMPUTATIONAL FLUID DYNAMICS

David Hunt
TESSELLA SUPPORT SERVICES PLC

Issue V1.R1.M4
October 2005

1 INTRODUCTION

The need to control and predict the movement of fluids is a common problem. The study of this area is called fluid dynamics and the systems that are studied range from global weather patterns, through aircraft aerodynamics to the way blood circulates. Computational Fluid Dynamics (CFD) takes these problems and solves them using a computer.

CFD and its application is a rapidly developing discipline due to the continuous development in the capabilities of commercial software and the growth of computer power. CFD is already widely used in industry and its application is set to spread.

This guide aims to provide an introduction to CFD and an overview of current software and techniques, including ways in which smart IT based methods can increase productivity.

2 HOW DOES CFD WORK?

The physics which governs fluids is relatively simple; the laws of motion and thermodynamics with a little bit of chemistry. However, the solutions are very complex and this makes analytical methods (pencil and paper) largely unusable for industrial applications. A common engineering approach to such complex dilemmas is to replace the problem with a number of smaller less complex problems. With the advent of computers this approach became practical and in the late 1960s Computational Fluid Dynamics was born.

2.1 The Science

To explain the idea behind CFD lets take an example of an aeroplane. As the plane moves along the air must move out of its way. The way in which the air flows depends on the plane’s shape. The flow can be smooth but more likely it will contain vortices, shockwaves and other disturbances (see figure 1).

To model the behaviour of the fluid (in this case air) the volume is split into many smaller sub-volumes, called a mesh (or grid). A mesh can be simply the same sub-volume repeated throughout the space or, more usually, it can be moulded around the object that is being modelled and so can be complicated (see figures 2 and 3).

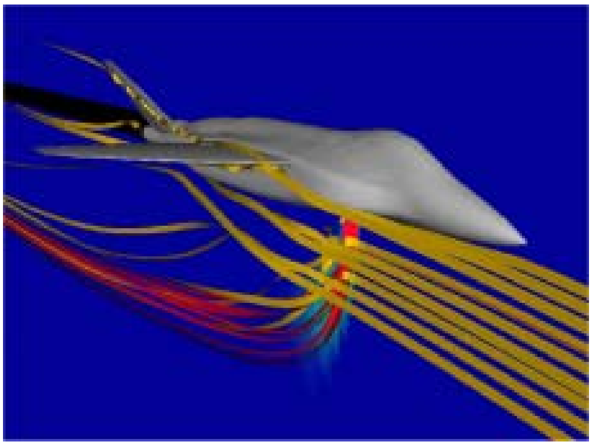

Figure 1: Flow associated with a jet in cross flow [2]

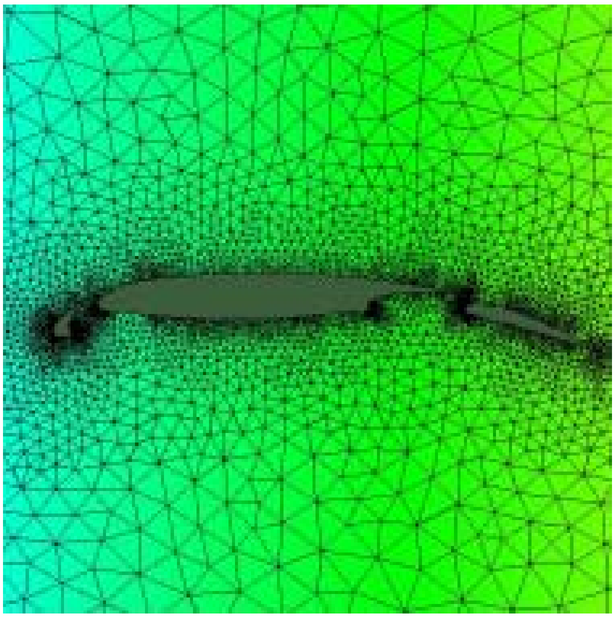

Figure 2: Mesh Around a High Lift Aerofoil [8]

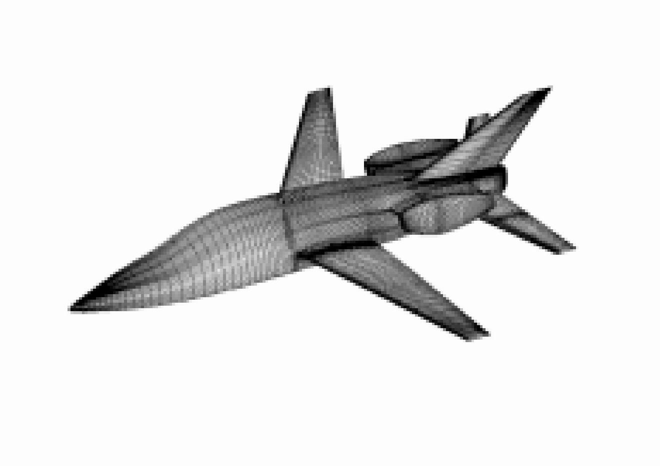

Figure 3: Mesh on the Surface of an Aircraft [2] (The grid extends into the volume around the aircraft – not shown)

Splitting the volume into sub-volumes is elegant in theory but hard in practice. The art is to create a mesh with exactly the right sub-volumes. If the sub-volumes are too big then the solution will have errors; if the volumes are too small then the calculation will take too long to be useful in a design process. In 1990 CFD was an activity for owners of CRAY super-computers, but by 2000 the same problems could be solved in a fraction of the time on PCs. As a result the use of CFD has gone from an elite minority in 1990 to widespread use today.

The airflow within each sub-volume is simple enough that it can be modelled using only the conservation equations of mass, momentum and energy. Other effects can be added to make the model more realistic if required, such as chemical reactions or heat exchange.

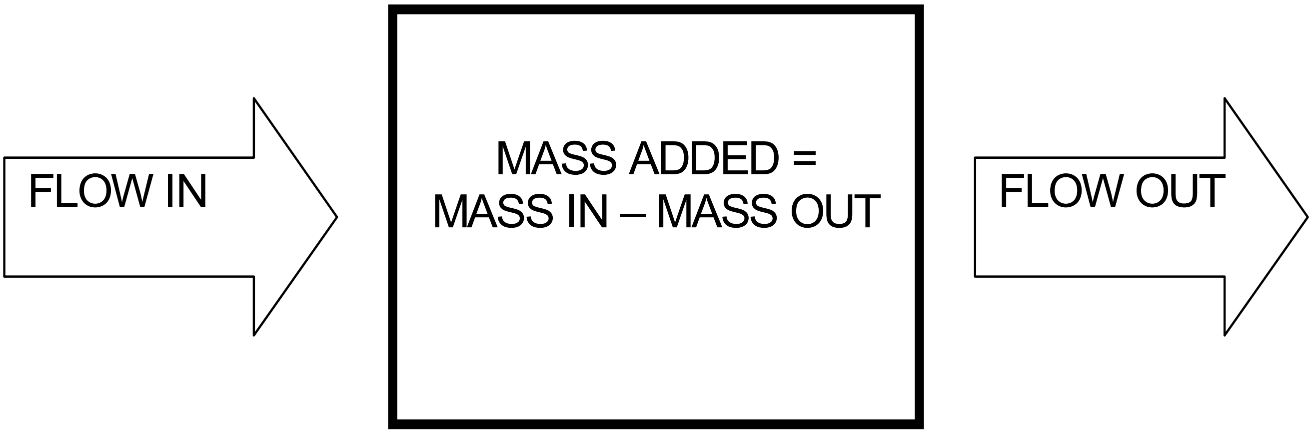

Figure 4: Conservation of mass applied to a sub-volume

The airflow is analysed by starting from an initial flow, which can be either a guess at the solution or a specific initial condition. Using this initial flow the conservation equations are used to predict the flow a short time later. A new prediction is then made from the newly calculated flow. In this way the evolution of the airflow can be solved.

Some flows, such as those around a wing, are naturally steady. For these steady flows, the process is repeated until the solution doesn’t change from one time to the next, i.e. the solution has ‘converged’. The opposite of this is when the flow is unsteady, such as the flow behind the back of a car or the inside of a car engine. For these flows, the solution never settles down and the aim of CFD is to track how the flow changes through time. Most applications of CFD involve steady flows because they require less computer power.

This explanation of using the conservation equations as a flow solver is a simplification; there are other more complex solvers. However, these methods produce similar outputs and, more importantly, CFD systems look the same to a user regardless of the solution technique.

Having obtained a flow solution, the user is presented with the flow at every point in the mesh. This is a vast amount of data and not very useful on its own. The last phase of the CFD process is to extract from these data the information that the user actually wants, e.g. the forces on the aircraft.

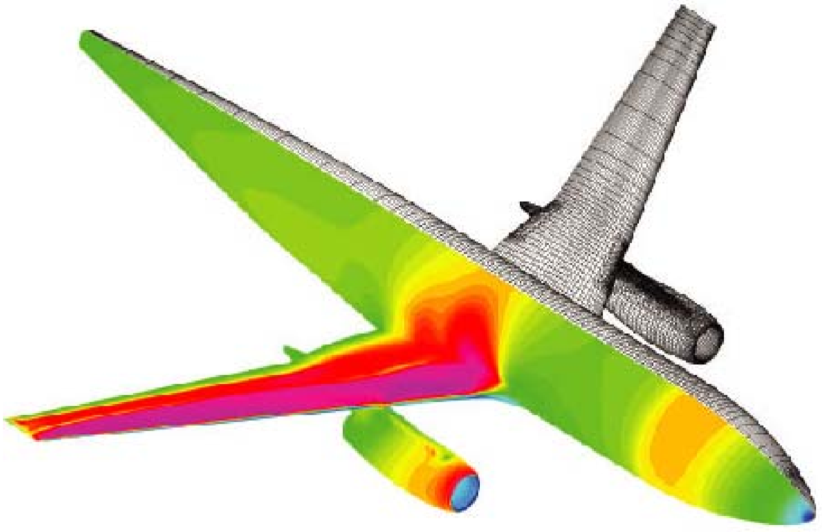

Figure 5: Forces on an aircraft due to airflow [2]

2.2 Summary of the CFD Process

Using CFD is a process with three stages:

(1) Pre-processing
  - represent objects in the flow domain, e.g. the aircraft surface, using CAD tools
  - split the flow domain into sub-volumes to create a mesh

(2) Obtaining a Flow-Solution
  - run a flow-solver using the mesh for flow conditions specified by the user

(3) Post-processing
  - extract and visualize the flow data from the results of the flow-solver (see figures above for examples)

The first stage is the most time consuming because the quality of the mesh determines the quality of the flow solution. In addition, mesh generators can be temperamental and it can take some effort to get any mesh at all!

2.4 Accuracy

CFD suffers from one big draw back - accuracy. In reality, most problems are far too complex to be solved accurately. For an aircraft, an accurate solution on today's top-end computers may take several centuries! The physics is simple but the flow is extremely complex. Over an aircraft wing the flow near the surface is turbulent; it is constantly changing with time and contains eddies which may be less than a millimetre in size. Modelling this complexity is expensive, so approximations are used which do not reproduce this complexity in the solution but 'model' the effects of complex phenomena on the flow. These 'models' are used to represent things like turbulence and chemical reactions. They are usually not very good and so the absolute accuracy of CFD tends to be poor due to use of a coarse mesh and inadequate 'models'.

For many applications however, such as building design, the level of accuracy required is very low and CFD results are more than adequate. Also, CFD usually gets the qualitative picture correct, which is useful in helping people understand what is happening within a flow. For applications which require high accuracy, such as aircraft design, CFD tends to be reasonably accurate at predicting the differences between one design and another. It is these 'deltas' which drive the design phase and more accurate techniques are used to estimate the absolute level of forces on the aircraft.

As a result of the accuracy of CFD being relatively poor, highly dependent on mesh quality and the models used, effective use of CFD relies heavily on experienced users. In essence, the CFD process requires expert intervention at every stage. Automation of the process is only possible where almost identical calculations are being carried out. Software solutions, which reduce the level of expertise needed to use a CFD system, will prove to be an important aspect of future CFD systems.

3 WHICH CFD SYSTEM?

When choosing a CFD package to use for a particular application, the user is faced with a bewildering array of options. For a comprehensive review of commercial and publicly available CFD packages, see the CFD-online web site [1]. There is a huge spectrum of fluid dynamic applications and this is mirrored in the variety of CFD systems available, from general-purpose CFD software through to very specific application-oriented software. There are some CFD systems that will provide solutions only for pipe networks whilst others will work only with 2D aerofoils. General-purpose software is more versatile but specialist software tends to produce results with less user input or at reduced cost or using specific specialist techniques.

There are three sources of CFD software and each will now be considered in turn.

3.1 Commercial / Off-the-Shelf Packages

These are CFD packages developed and supported by commercial organizations. Vendors tend to have very good sales teams who can provide advice on how to use their software. The leading general purpose products in the UK are:

- *STAR-CD* from CD-ADAPCO, London
- *FLUENT* from FLUENT plc, Sheffield
- *CFX* from ANSYS Inc.
- *CHAM* from Phoenix, London.

Each system has pros and cons and each aims to be usable for as wide a range of applications as possible, although certain industries prefer specific products.

In addition there are specialist codes such as PIPENET and MG-AERO which offer specialist support to niche markets.

3.2 Bespoke Systems

The advantages of specialist software over general-purpose packages lead many organizations, both industrial and academic, to develop CFD systems in-house for their own user-base. Aerospace organizations, for instance, tend to rely heavily on bespoke CFD because they require high accuracy results with minimal CPU cost, which general-purpose software cannot deliver.

When considering CFD requirements it's worth investigating the existence of bespoke software in other organizations, especially academic institutions and research associations. The owners and developers of these systems are often very enthusiastic about increasing the user-base of their software. Some bespoke codes are now being offered commercially, such as the SAUNA and SOLAR systems, which offers a CFD suite for aerospace applications. SAUNA and SOLAR are developed and marketed by the Aircraft Research Association [2].

3.3 Freeware and Shareware

There is a proliferation of freeware and shareware CFD systems. These range from undergraduate projects, made available on the web, to more sophisticated systems. Often these packages lack the ease of use associated with commercial and some bespoke systems. One of the best offerings is Visual3, a freeware CFD-specific visualization tool developed by MIT [3]. Visual3 is used in a number of commercial organizations. However, before Visual3 can be used it requires a small amount of code to be written to provide an interface between the Visual3 library and each specific CFD data format. Even so, Tessella could perform such tasks for a fraction of the cost of an off-the-shelf visualization package.

Publicly available software may require some effort in installation and validation before it is fit for the purpose, but again, Tessella could do this for far less than the cost associated with buying in commercial software.

4 NEW DEVELOPMENTS IN CFD

CFD still faces many challenges in terms of accuracy, usability and applicability. As a result, there is a lot of research and development required for CFD to reach its full potential. There are a number of areas in which this development effort is bringing new technology to the user.

4.1 CAD to CFD

Getting geometries into a form that a CFD package can understand is still a big issue. Many commercial codes come with geometry manipulation tools. However often companies wish to seamlessly knit their existing CAD software with their CFD system. Unfortunately, the available tools are often insufficient for this purpose. In such cases, specific software is required to provide the bridge. Tessella's experience of customising CAD systems can provide the skills required to achieve this.

4.2 New Grid Technologies

Historically, most meshes have been created using tetrahedral cells, block-structured hexahedral cells or a locally refining Cartesian approach. Different mesh types have advantages and disadvantages related to cost, efficiency for flow-solutions and automation of mesh-generation. In recent years, there have been considerable efforts to develop the hybrid approach, which mixes different mesh types [2,4].

Prismatic meshes offer a different solution to the mesh generation problem. Surfaces are represented using a collection of 2D shapes, usually a triangulated grid, and these shapes are marched away from the surface to form layers of prisms. Away from the surface, tetrahedra are used to fill in any gaps in the mesh. This approach is well suited to viscous applications, which require high mesh density normal to solid surfaces. However, this approach is limited by its ability to march away from complex geometries, such as intersections between multiple surfaces.

A more flexible approach uses techniques derived from the hybrid and marching strategies. The SOLAR CFD system, a bespoke system developed by ARA, BAE SYSTEMS and QinetiQ [5] uses this technology. Surface meshes are generated using predominantly unstructured quadrilaterals. These meshes are marched away from the surface, to generate predominantly hexahedral near-field meshes. Each new layer of mesh is modified to maintain quality even in the vicinity of the most challenging geometries. Cartesian meshes are then used to fill the flow-domain away from surfaces. The combination of these different technologies offers high quality meshes suitable for viscous applications without the need for the intensive user interaction, as is currently the norm.

4.3 Flow Solvers

The basic flow solver technologies have changed little in recent years. The main advance has been to take flow solvers that were previously only capable of solving low-speed flows, and extend their range to include high-speed flows, and vice-versa. It seems unlikely that there will be any major advances in this respect.

The innovation at the moment is in the development of models that have been used to represent the physics associated with different applications. Many new models have appeared in commercial packages, widening the range of problems to which they can be applied, including combustion models, multi-phase flows, particulate modelling and support for non-Newtonian fluids. In addition, most commercial packages now offer a full range of turbulence models from the basic $k\text{-}\varepsilon$ model with wall functions through to complex Reynolds-stress models. There has been little progress in the development of significantly better turbulence models in recent years. However, a number of existing models have been extended to deliver a capability that is more robust and applicable to a wider range of flows. In particular the following models have generated a lot of interest; Menter's SST k-w model is good for predicting boundary layer separation while the Realisable k-e model appears to be good at predicting shear layers and vortical structures. For external aerodynamics, coupled boundary layer packages continue to provide an effective and considerably cheaper alternative option to turbulence models. Specialist packages can be obtained with this functionality [3].

With increased computer power has come the possibility of performing affordable unsteady calculations. This leads to even more methods of calculation such as unsteady RANS, LES and VLES; many of these are available in commercial products. Before performing an unsteady calculation, some thought should be given to how the results will be analysed. Can the software generate the information required or is some extra software needed? In addition to this, there are IT issues associated with how to manage the data generated from these calculations in a way which allows the user to get the full value from the simulation.

Recent work has demonstrated that for unsteady flow calculations, where the unsteadiness is driven by a moving surface, it may be possible to calculate the flow at a snapshot of the motion. This approach greatly reduces the cost of a simulation. Currently, this method is applicable to a limited range of problems and is only available in a few bespoke codes [6]. Tessella have been involved in developing this technology which has potential application to propellers, ship screws, wind turbines, flutter and store release calculations.

4.4 Visualization and Analysis of Results

Before performing any CFD calculation, you should think about the information needed and how it will be extracted. To display the results of a CFD calculation most systems come with a post-processing and visualization package. These are often based on commercial packages.

The main commercial visualization packages used in the UK are:

- *ENSIGHT*, CEI Inc, USA
- *FIELDVIEW*, Intelligent Light, USA

These packages can be used to view general 3D data but have many functions specifically for CFD.

A number of developments are taking place, which enhance the power of these post-processing tools. Several organizations have been developing methods to extract and visualize specific flow features. FIELDVIEW and ENSIGHT have recently released a limited capability to visualize shock waves and vortical features based on a publicly available library released by MIT. Tessella has been actively involved in this field and as part of this effort has helped develop tools to efficiently process large data-sets [7].

Many organizations provide more tailored visualization solutions that are based on existing packages, such as Visual3, AVS and TECPLOT. Particularly if there are specialist visualization or analysis requirements, these packages can be set up to give specific functionality not provided by the commercial packages. Tessella has considerable experience in this field.

The final visualization option is an in-house solution, which is tailored to the exact needs of the application.

5 HOW TO BE ‘SMART’ ABOUT USING CFD

Smart solutions use technology, particularly IT, to improve the way organizations work so that they can deliver better products at reduced cost and in reduced timescales. CFD uses the power of IT to deliver impressive results, however, turning those results into something that will deliver benefit to an organization may require smart solutions. For instance, a CFD calculation of an aircraft does not design a new aeroplane.

Organizations using CFD are increasingly finding that they can make much better use of their CFD technology by using smart solutions. The aim is to build systems for specific applications that encapsulate the knowledge of CFD experts. For instance design tools can be written which automatically make use of CFD and thus allow designers to benefit from CFD without having to become CFD experts.

A simple example of using smart solutions would be to build a GUI for a collection of FORTRAN programs. The GUI would provide an integrated, user-friendly environment coupled with an interactive help system. Such environments can make CFD software easier to use and more accessible to non-experts.

Smart solutions can achieve much more than this however. CFD management systems can automate the entire CFD process from the initial CAD through to the analysis. They can even be used to automatically explore a design space. In addition to a user-friendly front end to that design space, a database and other data management facilities can be added to track data through entire projects.

CFD-management tools can use a variety of CFD methods to provide a cost effective route to exploring a large number of design concepts. For example, a large number of cheap calculations with low accuracy can be merged with a small number of more accurate calculations to provide a wide-ranging and accurate coverage of the design space.

Building smart systems that manage the use of CFD and other design tools under one interface can lead to a powerful system that radically increases productivity. However, these systems require a wide range of IT and software expertise together with the relevant technical expertise. Tessella is uniquely placed in having staff with all of the necessary skills to develop these systems.

6 WHERE TO LOOK FOR MORE INFORMATION

6.1 CFD on the Web

Most organizations involved in CFD development, including bespoke software, have web sites related to their use of CFD, e.g.

- ☐ www.ara.co.uk - examples of some of the developments in CFD software for the aerospace industry
- ☐ www2.eng.cam.ac.uk/~mea/fluid/cfdlab/cfdlab.htm – academic group producing specialist commercial software
- ☐ www.CFD-online.com - a very comprehensive CFD resource. It includes:
    - a list of commercial CFD development organizations (…/Resources/homes.html#Company)
    - a jobs list – indicates which organizations are active and growing
    - links to other introductions to CFD (if you don't like this one)
    - more advanced information on CFD

7 SUMMARY

CFD is a rapidly developing tool, which when coupled with the right systems has the potential to significantly impact and add value to the processes within organizations dealing with moving fluids. The immense array of options available implies that expert advice should be sought from a technical organization with CFD experience. If you are interested in exploring any of the ideas discussed in this article or if you are considering whether a smart IT or software solution could enhance your use of CFD then Tessella can advise you and assist with any development needs you may have.

Bibliography

[1] www.CFD-online.com, Oct. 2001

[2] Aircraft Research Association, Manton Lane, Bedford, www.ara.co.uk, Sept. 2001
The research department of ARA specializes in developing CFD technology for the aerospace (and related) industry and has pioneered a number of innovative techniques, including multi-block and marching grids.

[3] raphael.mit.edu/visual3/visual3.html, Jan. 2005

[4] Shaw, JA, Peace, AJ, 'Simulating three dimensional aeronautical flows on mixed block-structured / semi-structured / unstructured grids', Proc. 21st Congress of ICASE, Melbourne, Australia, 1998.

[5] Leatham, M, Stokes, S, Shaw, JA, Cooper, J, Appa, J, Blaylock, TA, 'Automatic mesh generation for rapid response Navier-Stokes calculations', AIAA 2000-2247, 2000.

[6] Hunt, DL, Childs, M, Maina, M, 'QUACC, a novel method for predicting unsteady flows- including propellers and store release', The Aeronautical Journal, August 2001.

[7] www2.eng.cam.ac.uk/~mea/fluid/cfdlab/cfdlab.htm, Sept. 2001
This group has pioneered the use of unstructured grids for viscous flows through its code NEWT, which is widely used in turbine and Formula 1 applications.

Tessella - providing innovative software solutions to scientific, technical and engineering problems

Tessella specializes in the application of innovative software solutions to scientific, technical and engineering problems. Our services cover software design and development, IT consultancy, infrastructure support and project management.

Other Technical Supplements published by Tessella include:

- ☐ Active Server Pages
- ☐ Archiving of Electronic Information
- ☐ Asset Management & Monitoring
- ☐ Automated GUI Testing
- ☐ Bayesian Statistics
- ☐ Beowulf Clusters
- ☐ Beyond LIMS
- ☐ C++
- ☐ Choosing and Using a LIMS
- ☐ COM
- ☐ Computational Fluid Dynamics
- ☐ Computer Image Processing
- ☐ Decision Support Systems
- ☐ Development for the Mobile Platform
- ☐ Digital Preservation Practical Experiences
- ☐ e-GIF
- ☐ Electronic Data Capture
- ☐ Electronic Lab Notebooks
- ☐ Evolutionary Computing
- ☐ Excel
- ☐ Extending the Life of Software
- ☐ FDA 21 CFR Part 11
- ☐ Formulation
- ☐ FORTRAN 90
- ☐ Grid Computing
- ☐ High Throughput Experimentation
- ☐ High Throughput Screening
- ☐ Instrumentation
- ☐ Integrated Lab Systems
- ☐ J2EE
- ☐ Java
- ☐ Linux
- ☐ Microsoft .NET
- ☐ n-tier Architecture
- ☐ Object Oriented Programming
- ☐ Open Source and Free Software
- ☐ Pocket PC
- ☐ Portable GUI Development
- ☐ Real Time Systems
- ☐ Regression Testing
- ☐ Security and the Internet
- ☐ Simulation
- ☐ Soft Computing
- ☐ Software Development Cycle
- ☐ Software Documentation
- ☐ Software Portability
- ☐ Software Re-engineering
- ☐ Software Specification
- ☐ SQL
- ☐ UNIX Inter-Process Comms
- ☐ UNIX System Performance
- ☐ Web Services
- ☐ Web Scripting
- ☐ Windows 2000 Services
- ☐ Workflow Systems
- ☐ XML
- ☐ X Windows

Certificate No. FM 22778

INVESTOR IN PEOPLE

Tessella Support Services plc
3 Vineyard Chambers, Abingdon, Oxon, OX14 3PX, England
Tel: (+44) (0) 1235 555511 Fax: (+44) (0) 1235 553301
info@tessella.com www.tessella.com