Paper 6.3

Overview of CFD Modelling of Orifice Plate Flometers

Neil Barton, Michael Reader-Harris and Jeff Gibson
NEL

OVERVIEW OF CFD MODELLING OF ORIFICE-PLATE FLOWMETERS

Neil Barton
Jeff Gibson
Michael Reader-Harris
National Engineering Laboratory

1   INTRODUCTION

Computational Fluid Dynamics (CFD) is a powerful tool for solving problems associated with orifice-plate flowmeters. It provides a useful complement to experimental data, particularly when applying information derived from tests to field conditions and in circumstances where a high degree of control is required which may be difficult to achieve in conventional tests. However, as with any simulation-derived data, the results of CFD simulations should be analysed appropriately.
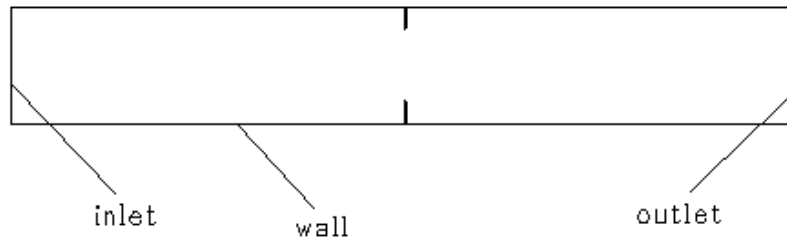
CFD has been used at NEL over the last 20 years to analyse various aspects of orifice-plate performance. It has been used in development of the ISO 5167 standards [1, 2] and in assessment of installation errors, orifice-plate contamination effects and environmental effects on numerous metering stations (e.g. [3, 4, 5 & 6]).

This paper provides an overview of CFD simulation of orifice-plates for flow-measurement Engineers. It introduces basic concepts of CFD modelling, and compares simulation results from various projects undertaken at NEL with test and field data to provide a feel for the accuracy of the CFD method. Finally, the application of CFD simulation results to industrial flow measurement problems is discussed.
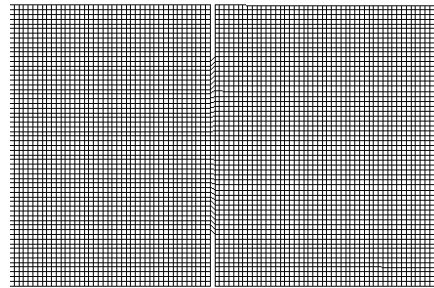
2   CFD MODELLING METHODS

This section gives a brief overview of the CFD calculation process. For more information, Versteeg & Malalasekara [7] provide an excellent basic introduction to the subject.
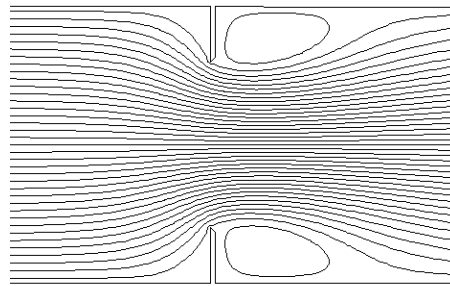
Figure 1 illustrates the process of setting up a CFD simulation. The first step entails defining the shape of the boundaries of the component to be modelled in two or three dimensions (in this case for an orifice-plate). The space occupied by gas or liquid is broken down into a mesh of small cells. Boundary conditions, such as inlet flow velocity and outlet pressure, are defined and the fluid density and viscosity are set. This information is used by an iterative numerical solver program to compute flow variables such as pressure and velocity for each cell. The results of a converged solution can then be viewed in terms of flow vectors, streamlines and coloured contours of the computed variables. Point values of variables can also be extracted from the solution data.

a) Definition of geometry and boundary conditions

b) Meshing

c) Visualisation and processing of results

Figure 1.   Illustration of the CFD modelling process

In the case of a simulation of an orifice-plate operating in non-ideal conditions, point pressure values at the tapping locations are used in the standard orifice-plate equation to define the flow rate measured by the orifice-plate:

$$Q_{measured} = C\varepsilon \frac{\pi d^2}{4} \frac{\sqrt{2\rho\Delta P}}{\sqrt{1-\beta^4}}$$ Equation 1

In which:

C   is a discharge coefficient calculated based on a CFD simulation of the orifice-plate in ideal flow conditions
d   is the orifice diameter
ρ   is the fluid density
ΔP   is the pressure difference between upstream and downstream tappings
β   is the orifice-plate diameter ratio
ε   is the expansibility coefficient, set as unity for incompressible flows

Flow measurement error is then defined as follows:

$$\% \, Error = 100 . \frac{(Q_{measured} - Q_{actual})}{Q_{actual}} \%$$

Equation 2

In which:

Q_{actual} is the mass flow rate defined as a simulation boundary condition

The roots of CFD methods can be traced back to Leonhard Euler's work on inviscid flow in the 1750's and the development of the Navier-Stokes equations between 1822 and 1845. However, the use of computers to solve numerical representation of turbulent flow only really became commonplace in academia in the 1970's. The first commercially available CFD codes became available in the 1980's and, by the 1990's, CFD was regularly used to solve problems in a wide range of industries particularly in the automotive and aerospace sectors, but also in the oil & gas, chemicals, medical, defence and power sectors.

These days, nearly all industrial CFD is done using multi-purpose, commercially available codes such as Fluent, CFX, Star-CD and Phoenics. At NEL, Fluent software is used to model flow-measurement problems and other multiphase and heat transfer problems in the oil, gas and process sectors. Modern CFD software typically comprises a CAD-type interface which allows definition of complex three-dimensional components, a solver and a graphics package which will allow sophisticated flow visualisation. The most versatile codes will allow calculation of single-phase flows, multiphase flows, compressible flows, heat transfer, transient flows, deforming and moving components and chemically reactive flows. In the early days of CFD all work was done on mainframe machines, but increasing computer power has made CFD simulation on desktop workstations and PCs viable today.

Although the usability and speed of CFD software have increased immensely since the early days, its predictions are still potentially inaccurate if a simulation is poorly defined. The accuracy of the solution depends on the cell resolution in particular locations and the alignment of cells with the flow. If few cells are used, then sharp changes in the flow (e.g. at the edge of a jet) will not be well captured.

Significant inaccuracies can also be associated with using an inappropriate turbulence model. A vast range of turbulence models have developed, although the most popular for general CFD work is the k-ε model, which was developed in the early 1970's. Other popular turbulence models include the RNG k-ε model, the realisable k-ε model (both of which are variants of the original k-ε model), the k-ω model and various types of Reynolds Stress Model (RSM). All of these models have advantages and disadvantages. For example, it is known that the k-ε model can be inaccurate in some swirling flows but, as it is well understood and usually converges easily, it is a good choice for many general applications. It should be noted that, as yet, no turbulence model has been developed that is highly accurate or stable in all circumstances.

In practice, it is very difficult to separate inaccuracies due to poor mesh definition and inaccuracies associated with limitations in the turbulence model. Moreover, the number of variables associated with a typical simulation makes it almost impossible to define the uncertainty of an individual CFD calculation. Therefore the most practical method of judging the validity of a CFD prediction is to compare simulation results against good test data to gain a feel for the likely discrepancy between the two. The following section makes such comparisons for a range of different orifice-plate simulations.

3 EXAMPLES: CFD SIMULATIONS OF ORICE PLATES

3.1 Comparisons with ISO 5167

Most orifice-plate studies aim to predict the flow measurement error associated with some non-ideal operational condition. To achieve this CFD simulations are usually run of the orifice-plate in both ideal and non-ideal conditions as described in the previous section. The advantage of this approach is that it effectively eliminates most of the inaccuracy associated with the simulation method.

As a consequence of this method, a large number of simulations have been run of orifice-plates in ideal flow conditions over the years at NEL. Figure 2 compares predicted discharge coefficients with values given in ISO 5167. Depending on the particular problem being modelled, the orifice-plate can be represented in two-dimensional axisymmetric or three-dimensional co-ordinates. In the two-dimensional case, many more cells can be used increasing the resolution of the model and hence improving its accuracy. Consequently two-dimensional models have always tended to produce a better agreement with ISO 5167. However, more recently, increased computing power has facilitated a similar degree of accuracy in three-dimensional models. Consequently, we would expect most modern simulations of orifice-plates in good flow conditions to agree with ISO 5167 to within 1%.
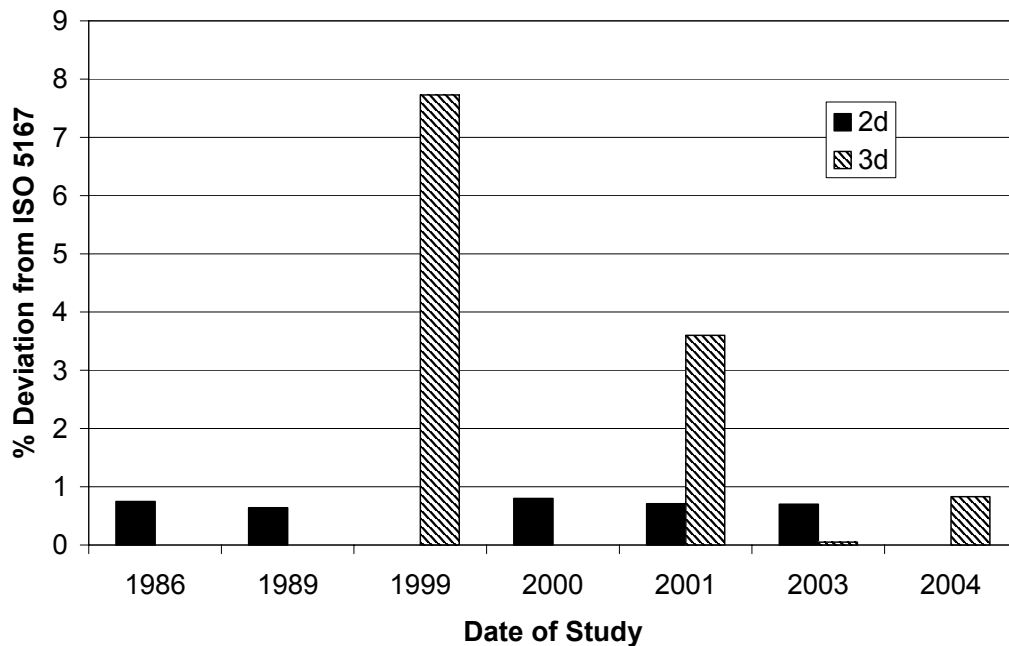

Figure 2.   Predictions of Discharge coefficient from a range of two- and three-dimensional studies

3.2   Reversed Orifice-Plates

A number of projects have been run at NEL which have used CFD simulations and test data to assess the consequences of installing an orifice-plate backwards. The metering error produced by a reversed orifice-plate depends on the ratio of the width of the bevel to the orifice diameter (b/d). Good agreement has been achieved between CFD predictions and test data [8] and field measurements in these studies (Figures 3 & 4).
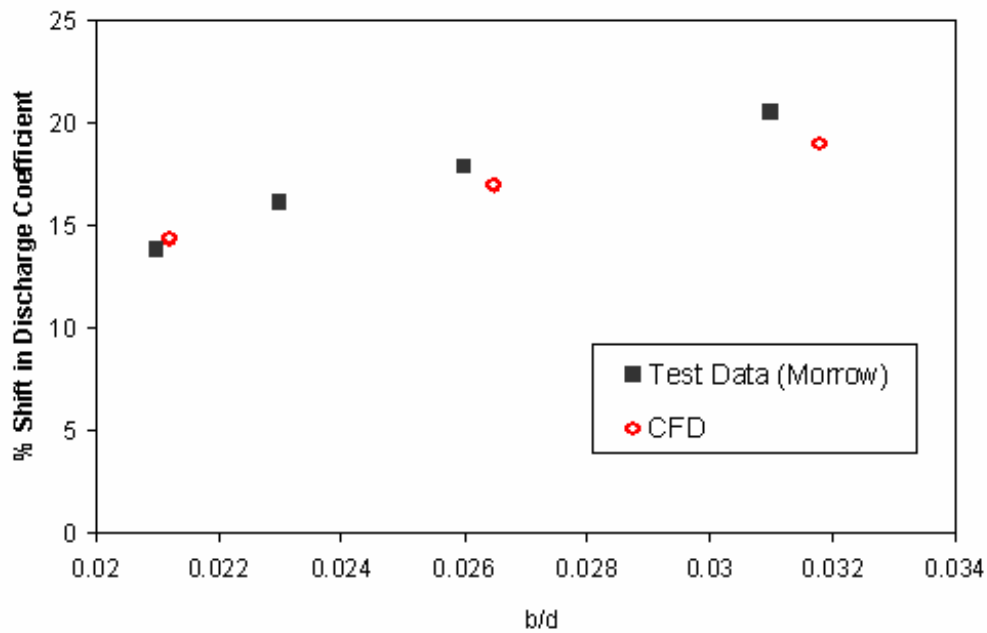
Figure 3.   Effect of reversing an orifice-plate - comparison with test data
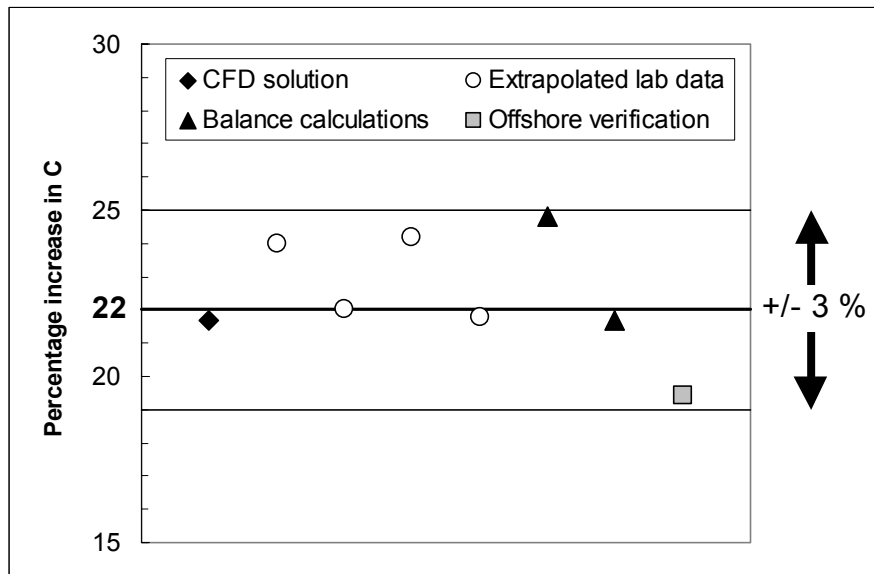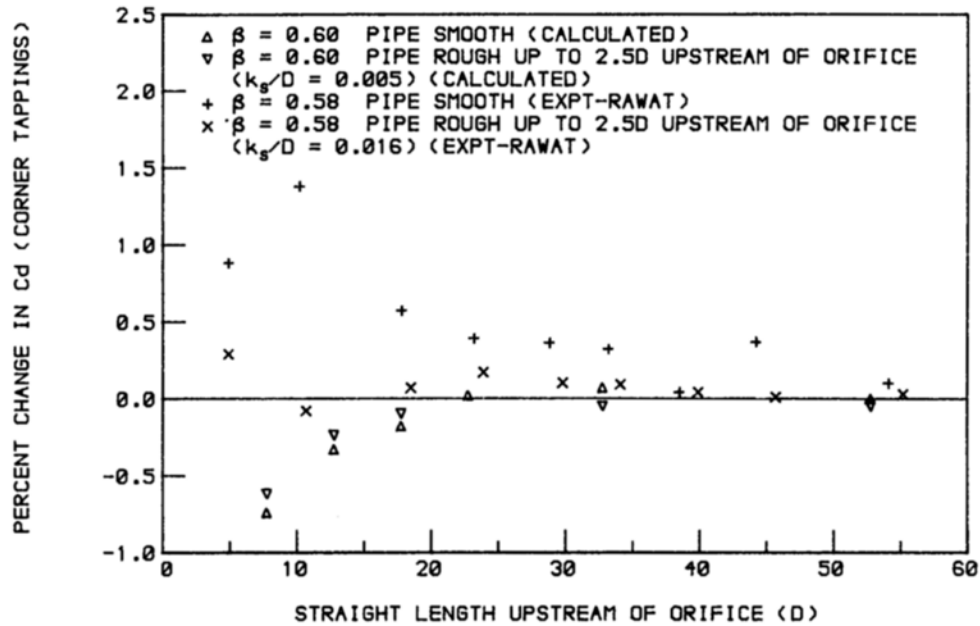

Figure 4.   A comparison of different predictions and measurements of the effect of reversing an orifice-plate in an offshore metering skid [3]
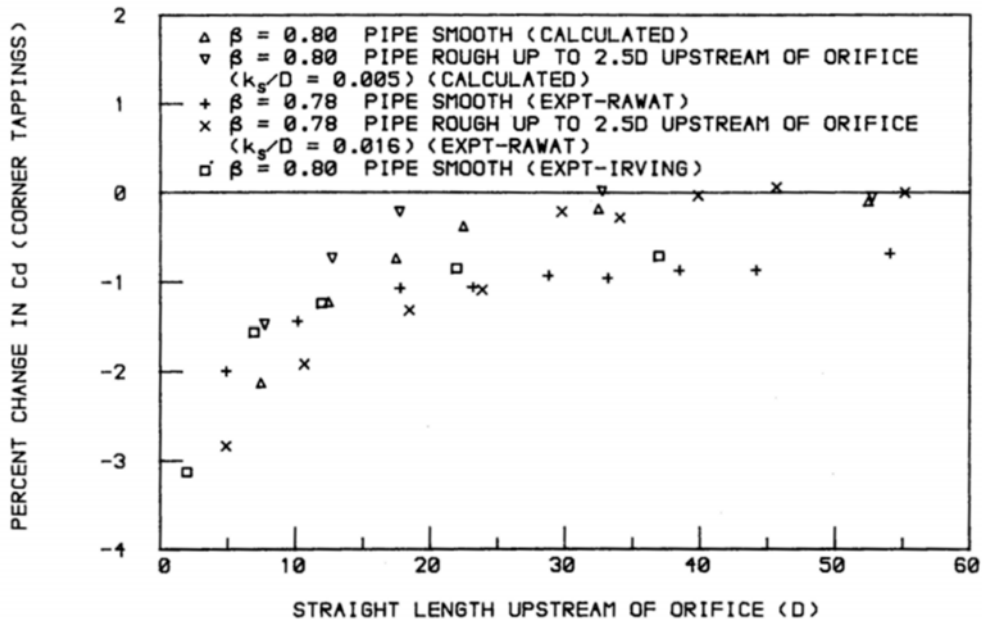

3.3   Errors Caused by Upstream Pipework Fittings

It is well known that if an orifice-plate flowmeter is installed too close to an upstream elbow, or other pipework fixture, its flow-measurement uncertainty will increase. Although there is a large amount of experimental test data on installation errors in orifice-plates, the large number of combinations of orifice-plate diameter ratio and upstream pipework configuration make assessing the installation error of a particular metering station difficult. CFD simulations provide useful information when estimating installation errors as they can simulate any combination of upstream fittings and therefore closely represent actual operational conditions.

CFD methods were first used at NEL to study orifice-plate installation effects in the 1980's [1]. This early work used a dedicated FORTRAN program to model orifice-plates in two dimensions with turbulence effects represented using the k-ε model. The effect of an upstream double bend was modelled by defining varying amounts of swirl at the flowmeter inlet, depending on the distance between the meter and the downstream elbow. The CFD results agreed well with test data for a plate with a diameter ratio of 0.8, but not as well for lower diameter ratios. The discrepancy at low diameter ratios was probably primarily caused by the fact that velocity profile distortion effects could not be adequately accounted for in the two-dimensional simulation.

a) Diameter ratio = 0.6

b) Diameter ratio = 0.8

Figure 5.   Comparisons of the predicted and measured shift in discharge coefficient caused by an out-of-plane double elbow [1]

The use of three-dimensional CFD models to predict orifice-plate installation errors was assessed at NEL in the late 1990's and three-dimensional models have been used in a number of industrial flow-measurement studies since then (e.g. [4, 5 & 6]). Figures 6, 7 and 8 compare three-dimensional CFD error predictions against test data from various sources. All of these cases use the realisable k-ε model to illustrate the typical degree of agreement (and discrepancy) that can be expected using a particular combination of mesh and turbulence model.

Figure 6 shows a good agreement between CFD simulations and tests of orifice-plates downstream of single elbows. Agreement is not quite as good immediately downstream of out-of-plane double bends, but predictions are within about 1% beyond 15D (Figure 7). The CFD simulation correctly predicts the change from negative to positive errors as the diameter ratio reduces. The effect of a 19-tube tube bundle downstream of an elbow is well predicted (Figure 8).
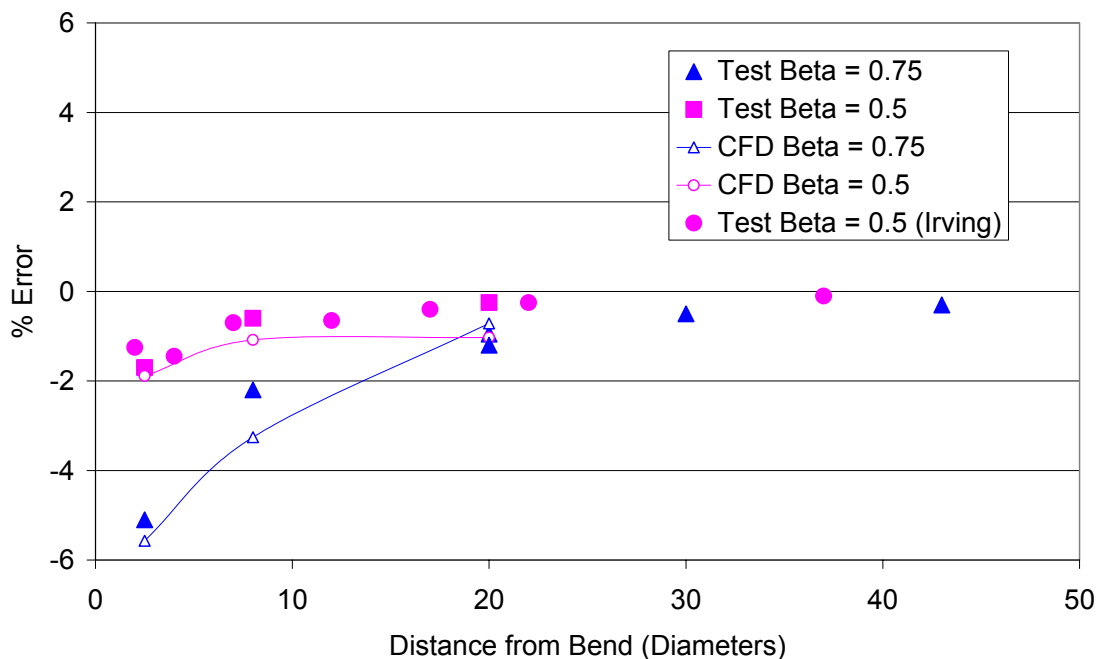

Figure 6.   Predicted and measured error of orifice-plates downstream of a single elbow
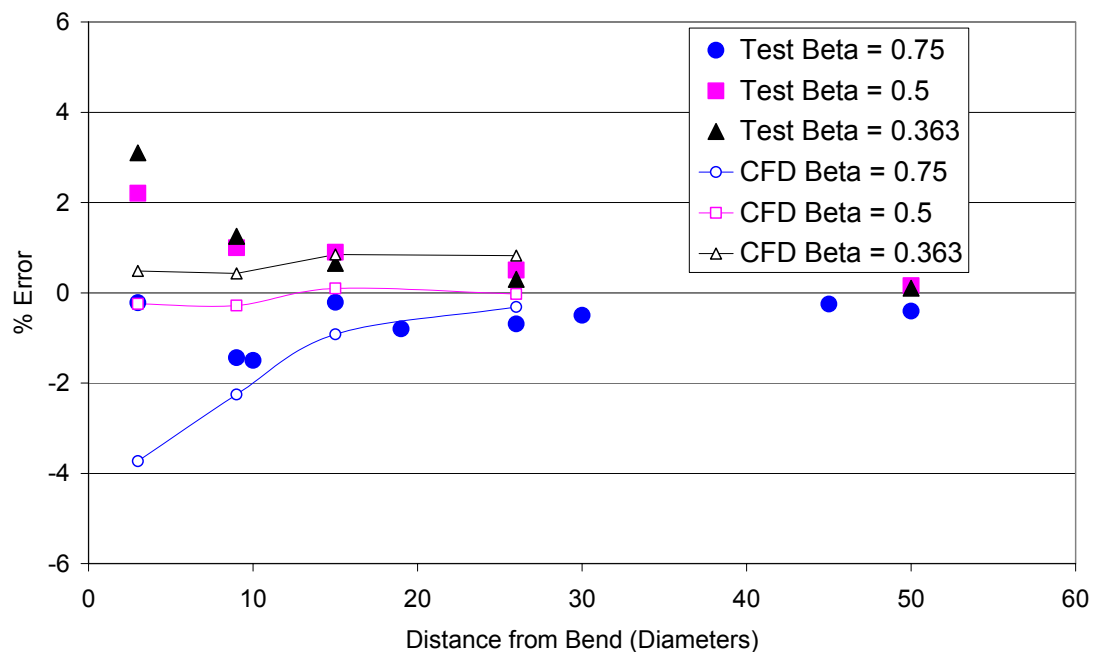(Test data taken from Mattingly & Yeh [9] and Irving [10])

Figure 7.   Predicted and measured error of orifice-plates downstream of an out-of-plane double elbow (Test data taken from Mattingly & Yeh [9])
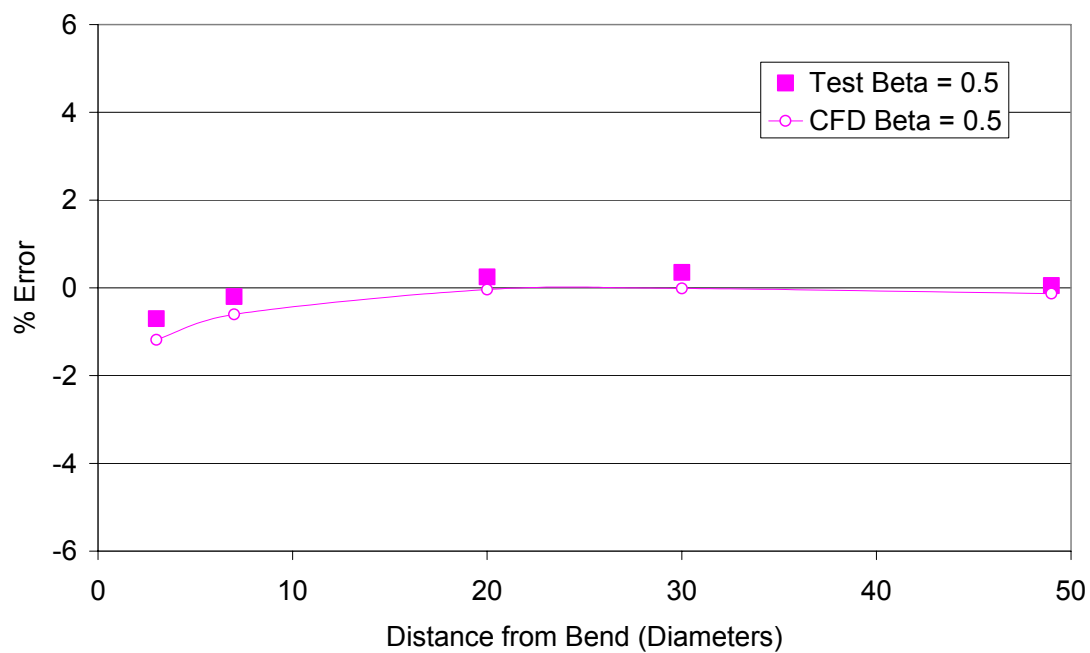

Figure 8.   Predicted and measured error of orifice-plates downstream of a single elbow bend with tube bundle (Test data taken from Mattingly & Yeh [9])

Simulations have also been run of a more complex header installation, as shown in Figure 9. This installation generates very high swirl in the metering tube. Figure 10 shows that the CFD simulations correctly predict a positive installation error and the trend of the error. They also show the difference between the top and bottom tappings at high diameter ratios. However, they under-predict the magnitude of the error at low diameter ratios. This result is consistent with the under-prediction of error in swirling flow downstream of double bends (Figure 7) and is believed to be primarily caused by limitations of the turbulence model in high curvature flows. In this case, alternative turbulence models, which account for high strain-rate effects, would be expected to give better agreement with the test data.
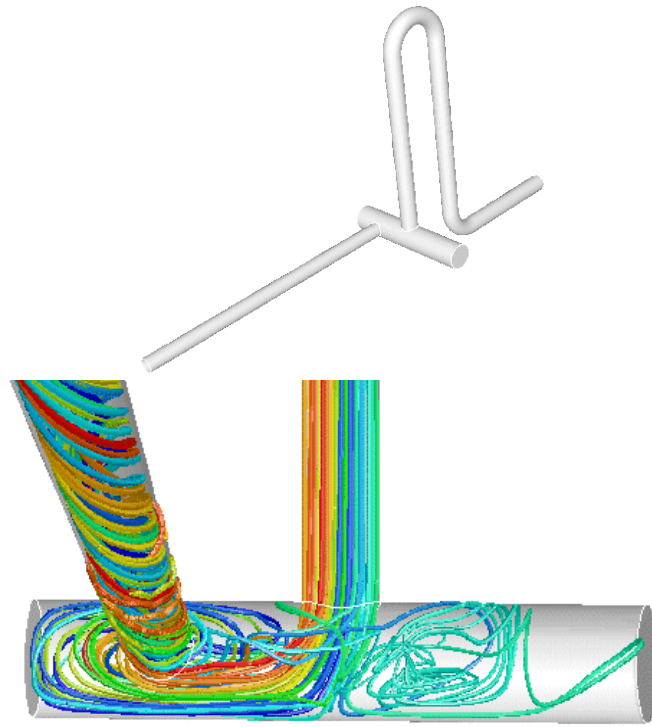

Figure 9.   Geometry and predicted flow pattern in the header installation
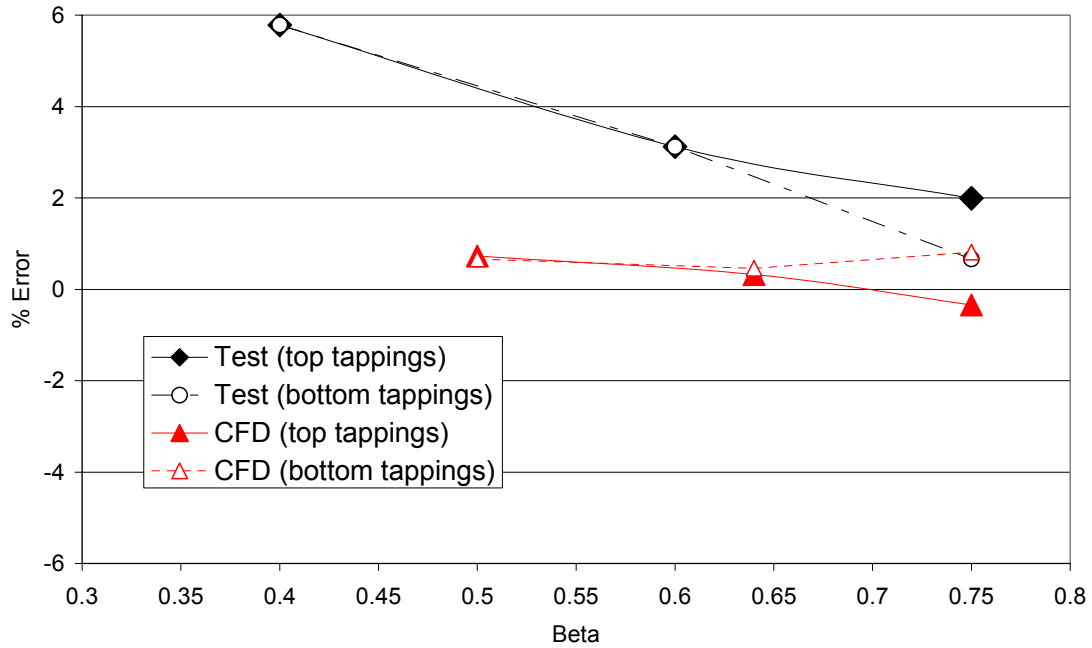
Figure 10.   Measured and predicted flow measurement error for the header installation

3.4   Roughness Effects and Orifice-plate Contamination

One of the primary difficulties in the study of roughness and contamination effects is the control and measurement of roughness under test conditions. Consequently, test data on roughness effects tends to be quite variable. One of the major advantages of CFD in this context is that well-defined roughness values can be specified, which can help a great deal when interpreting test results.

Two approaches are generally used in CFD simulations to model wall roughness effects, as illustrated in Figure 11. In the first, the "wall function", which determines flow behaviour immediately next to the wall, is adjusted to increase wall drag and, hence, adjust the shape of the boundary layer. This method requires definition of a roughness height and a shape factor. The shape factor for sand-type roughness (i.e. small, uniformly distributed, close-pack spheres) is well defined, but the shape factor for other types of roughness (e.g. radial channels or ribs) is not as well defined.
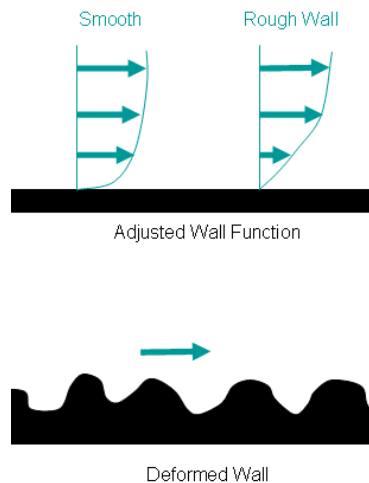

Figure 11.   Alternative representations of wall roughness

An alternative approach is to assume a smooth surface with deformed walls to represent the build-up of material on a surface. The practicality of this method will depend on the scale of the roughness and the amount of computing power available. The two methods can also be combined to produce a roughened and deformed surface. Consideration also needs to be made as to whether it is appropriate effectively to add to the wall thickness: for example to represent a surface encrusted in material, or to remove wall thickness if a surface is corroded or eroded.

Reader-Harris [1] used CFD methods to study the effects of rough-walled pipework upstream of orifice-plates. In this case wall functions were adjusted to model roughness effects. The simulation results compared reasonably well with a fairly scattered set of test data, gathered from a number of sources (Figure 12). The understanding of flow behaviour gained by using CFD in this case helped to explain previously confusing test data on orifice-plates and ultimately lead to the development of a method for estimating the flow measurement error caused by rough upstream pipes [11], see Figure 13 [12] .

More importantly, clarity about the dependence of the discharge coefficient on pipe roughness led to the replacement of the single Reynolds number term in the discharge coefficient equation by a Reynolds number term that is in fact a velocity-profile term (based on the known effect of pipe roughness) and a throat Reynolds number term [13].
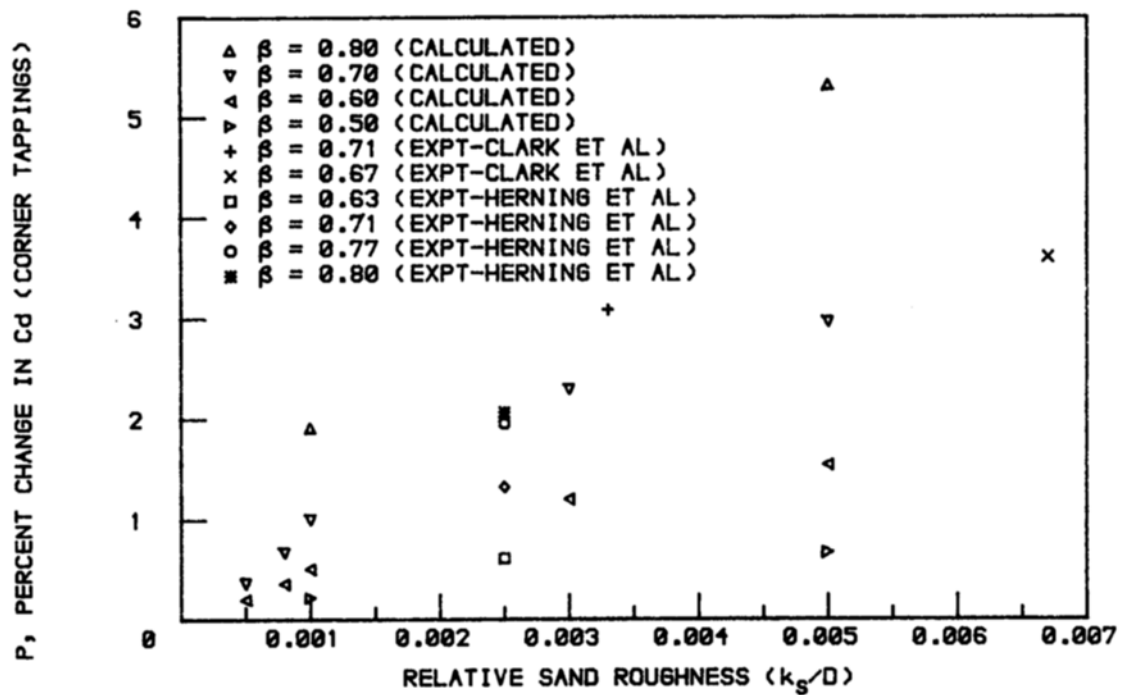

Figure 12.   Predicted and measured shift in discharge coefficient of orifice-plate installed downstream of rough pipes
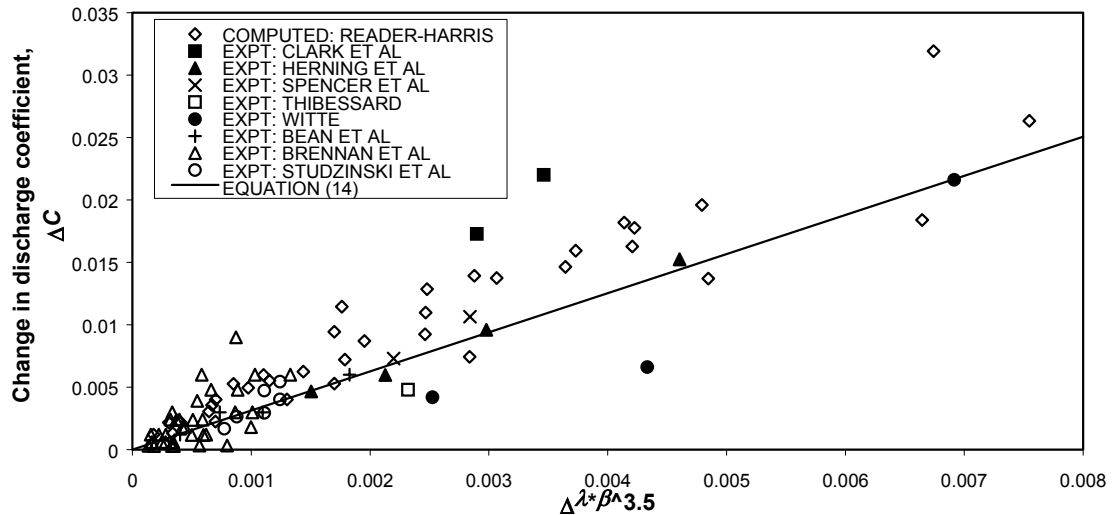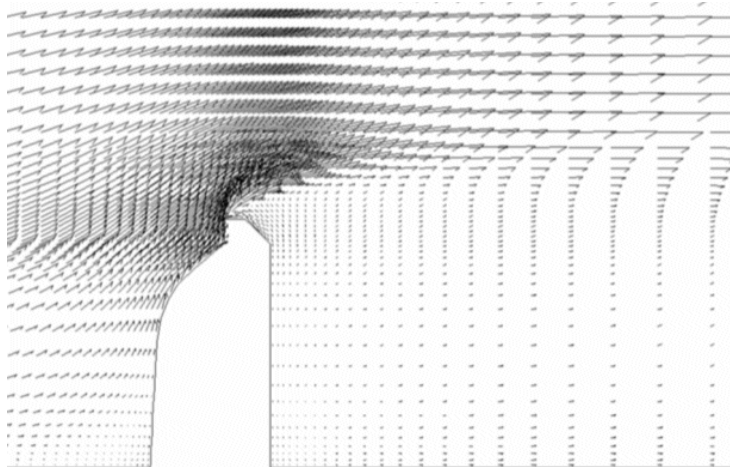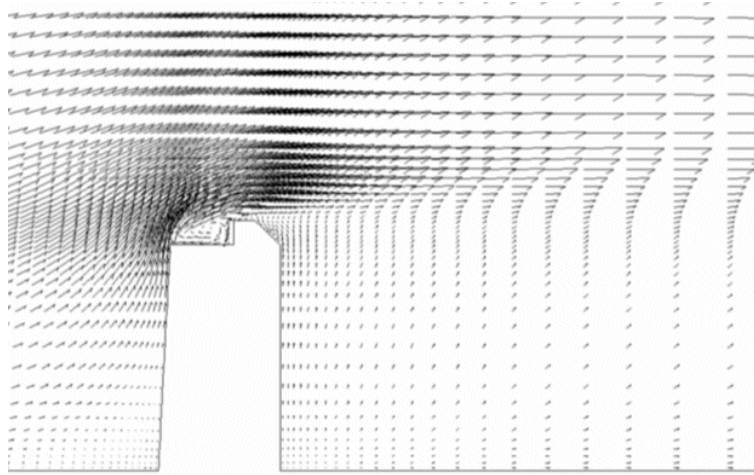
Figure 13.   The effect of rough pipe on the orifice-plate discharge coefficient [12]

The effects of orifice-plate contamination have also been modelled using both deformed walls and adjusted wall functions to represent the build up of material on the upstream face. In this study it was found that the flow measurement error depended heavily on the distribution and thickness of the contaminant layer.   Figure 14 shows how the distribution of a thick contaminant layer affects the flow behaviour around the edge of an orifice-plate. Although no direct validation with test data has been performed, predicted errors were of a similar magnitude to those predicted using methods outlined in BS ISO/TR 12767:1998 [14].

a) Rounded distribution profile

b) Sharp-edged distribution profile

Figure 14.   Predicted flow through an orifice plate with a thick layer of
contaminant on the upstream face

4.   CONCLUSIONS

This paper has presented the results of a series of CFD simulations modelling different aspects of orifice-plate performance in order to give a feel for the degree of accuracy that can be expected from CFD simulations and to illustrate ways in which CFD data can be used.

The quality of CFD results depends, primarily, on the mesh and the turbulence model used. A good match has been achieved between CFD predictions and test data in the case of reversed orifice-plates. CFD methods can also give reasonable predictions of orifice-plate installation errors provided that an appropriate turbulence model is used if highly swirling flow occurs. The influence of rough upstream pipework is well represented in CFD simulations. The study of roughness effects depends on the way in which roughness is defined within the simulation. CFD has proved useful in the study of rough pipes upstream of orifice-plates and contamination of plate surfaces with dirt or grease.   In the former case, the CFD results agree reasonably well with fairly scattered test results. In the latter case, basic agreement with information in ISO standards has been established.

CFD is a powerful tool for investigating flow-measurement problems in industrial conditions. In particular it permits the study of flowmeter response at operational conditions, with full control over the parameters being varied. It also provides a detailed understanding of the flow behaviour that leads to mis-measurements.   However, care has to be taken when defining the boundary conditions, the mesh structure and the turbulence model to ensure that the simulation results closely represent the phenomenon being studied. The results of CFD simulations are most useful when the source of errors is well understood and when predictions can be validated against test or field data.   As computing power increases and as more validation work is performed CFD is likely to be increasingly used to assess and to understand orifice-plate metering applications.

5.   REFERENCES

[1]   READER-HARRIS, M.J. Computation of flow through orifice-plates downstream of rough pipework. Proceedings of International Conference on Flow Measurement in the mid 80's, National Engineering Laboratory, 9-12 June 1986.

[2]   BARTON, N.A. & STEWART, C.D. Use of CFD techniques to improve ISO 5167-1. NEL Report 337/99, 1999.

[3]   BROWN, G.J., READER-HARRIS, M.J., GIBSON, J. & STOBIE, G.J.. Correction of the readings of an orifice-plate installed in reverse orientation. 18th North Sea Flow Measurement Workshop. Gleneagles, Scotland, 2000.

[4]   WILCOX, P. & BARTON, N.A.. Using computational fluid dynamics to investigate the flow through an offshore gas metering station. 19th North Sea Flow Measurement Workshop. Kristiansand, Norway, 2001.

[5]   BARTON, N.A. Modelling the effect of the weather on an unlagged flowmetering system. Flowtidings DTI Flow Programme Newsletter, Issue 40, Winter 2004. http://www.flowprogramme.co.uk/Publications/flowtidings.asp

[6]   WOOD, I.. Replacement of an orifice sales gas metering system with ultrasonic meters. Flow Measurement 2001 International Conference, Peebles, Scotland, 7 – 10 May 2001.

[7]   VERSTEEG, H.K. & MALALASEKERA, W.. An introduction to computational fluid dynamics. The finite volume method. Longman Scientific & Technical, 1995. ISBN 0-582-21884-5

[8]   MORROW, T. B. Private Communication. San Antonio, Texas: GRI MRF, February 2000.

[9]   MATTINGLY, G. E., and & YEH, T. T. Summary report of NIST's industry-government consortium research program on flowmeter installation effects with emphasis on the research period February - December 1990:TEE, used as an elbow configuration. NISTIR 4753. Gaithersburg, USA: National Institute of Standards and Technology, 1992.

[10]   IRVING, S. J. Effect of system layout on the discharge coefficients of orifice-plates. Part 3: Experimental investigation into the effects of bend combinations. BHRA Report RR1462. Cranfield: BHRA, 1978.

[11]   READER-HARRIS, M.J.. Pipe roughness and Reynolds number limits for the orifice-plate discharge coefficient equation. Proceedings of 2nd International Symposium on Fluid Measurement, Calgary, Canada, PP 29-43, June 1990.

[12]   READER-HARRIS, M. J. The revision of ISO 5167 – an update. In Proc. of 16th North Sea Flow Measurement Workshop, Gleneagles, pp 22, October 1998. East Kilbride, Glasgow: National Engineering Laboratory.

[13]   READER-HARRIS, M. J., and SATTARY, J. A. The orifice plate discharge coefficient equation. *Flow Measurement and Instrumentation*, Vol. 1, pp 67-76, January 1990.

[14]   INTERNATION ORGANIZATION FOR STANDARDIZATION. Measurement of fluid flow by means of pressure-differential devices – Guidelines to the effect of departure from the specifications and operating conditions given in ISO 5167-1. ISO/TR12767: 1998.