Colour in Visualisation for Computational Fluid Dynamics

David Kinnear[1], Mark Atherton[3], Michael Collins[2], Jason Dokhan[2] & Tassos Karayiannis[2]

[1] *Fluent Europe, Sheffield, S9 1XU*
[2] *School of Engineering & Design, Brunel University UB8 3PH*
[3] *Department of Engineering Systems, London South Bank University, SE1 0AA*

Abstract

Colour is used in computational fluid dynamic (CFD) simulations in two key ways. First it is used to visualise the geometry and allow the engineers to be confident that the model constructed is a good representation of the engineering situation. Once an analysis has been completed, colour is used in post-processing the data from the simulations to illustrate the complex fluid mechanic phenomena under investigation. This paper describes these two uses of colour and provides some examples to illustrate the key visualisation approaches used in CFD.

1. Overview of CFD simulation

It should be appreciated that the relatively recent development of commercially biased CFD codes, in fact represents the culmination of theoretical and experimental research in fluids for over a century. The basic rationale is that there are two major classes of substance behaviour in Nature, namely that for a) solids and b) fluids (comprising liquids and gases). Whereas in the presence of a force field, solids suffer *deformation,* fluids *deform continuously*.

These responses may be expressed as descriptive equations for the momentum principle:

for a solid, stress = function [strain]

while for a fluid, stress = function [rate of strain].

In the case of fluids the above expression of conservation of momentum, combined with a similar expression for conservation of mass, leads to the Navier-Stokes equations, defined for some 125 years. They form a set of highly complex non-linear partial differential equations, untreatable theoretically except for a small number of 'classical' solutions for fairly simple cases. However, numerical simulation treatments have developed in parallel with that of digital computers since the Second World War, and dramatically so since about, say, 1990.

Simulation through Computational Fluid Dynamics (CFD) enables the dynamics of flow to be studied by building a model that mimics the circumstances needing investigation. CFD allows the prediction the fluid dynamics of a system of interest. It allows the visualisation not only of flow velocities, pressure distribution and turbulence, but also transfer of heat, phase changes, physical movement induced by flow and areas of particle and droplet transport.

CFD has found numerous applications across many industries. These applications vary widely but common to all is the visualization of the models and simulation results. The intelligent use of colour within the visualization techniques employed proves invaluable. This paper discusses this use of colour to aid the visualisation of models and results in a CFD simulation. This use of colour is not unique to CFD, but is common to many forms of engineering analysis and simulation.

2. The necessity for use of colour

A typical CFD simulation might involve complex geometry represented by many millions of grid or mesh points. Visualisation of this geometry allows the engineer 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. An engineer can use this data to determine important engineering parameters such as drag coefficients, heat transfer rate and maximum temperature depending on the purpose of the calculations. However, if we want to understand why we need to understand what is happening in a qualitative manner. 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 that a single graphic can replace a thousand words.

3. Rendering to illustrate complex shapes

One way colour is used is to visualise the geometry of the model. A typical model is constructed based on the geometry of the engineering situation. The surface of the object is made up from points (or nodes in CFD terminology) and these nodes are connected together by edges that form a surface face mesh. The points and edges of the mesh can be viewed directly whereas the intricate detail of the surface is better represented by shading the surface mesh (see Figure 1). Geometry rendering using shading allows a 2-D graphical image to be perceived by the eye as a 3-D object. Colouring can further enhance representation of complex shapes. This is illustrated in Figure 1, where the surface of a mixing impeller is represented in these different ways.

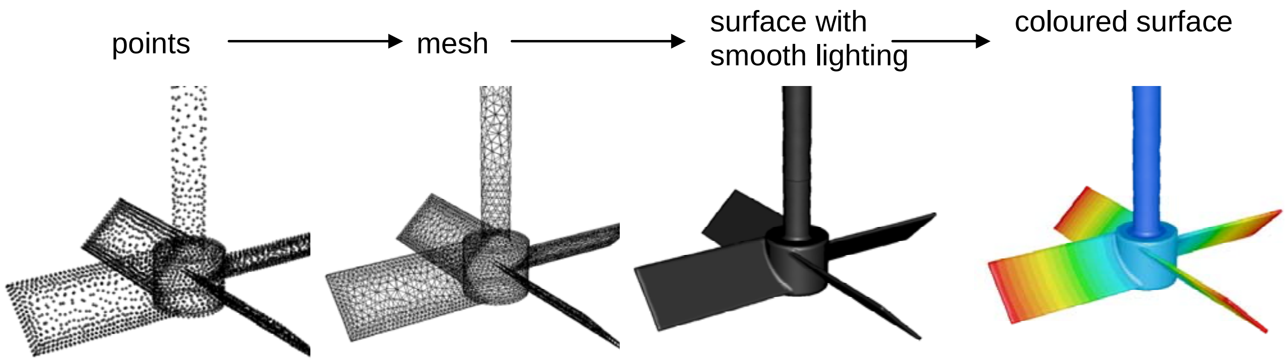
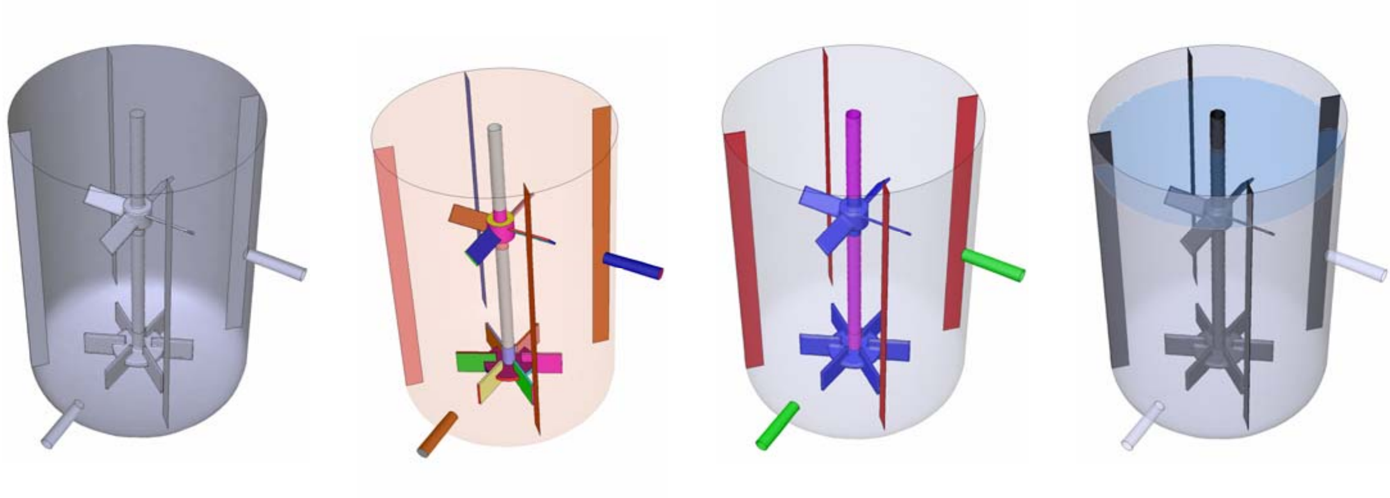


Fig. 1. Rendering to illustrate the shape of an impeller used in a mixing vessel

4. Colouring of items to identify similar objects among a large number of objects

Where the geometry of interest consists of a multitude of objects, grouping them and colouring the groups allows the viewers to quickly understand what they are looking at. Figure 2 illustrates an example of a model of a mixing tank. Here, the surfaces of the objects are coloured in different ways. Colouring in grey is often used just to illustrate the geometry of the objects when the flow is being visualised but it fails to illustrate the different objects in a model. Colouring objects individually lets the viewer see the different objects. Colouring items of the same type in the same colour is perhaps the best way to illustrate a model for checking purposes. In Figure 2c the impellers are coloured in blue, red illustrates the baffles, purple represents the shaft and green represents the inlet and outlet pipes. In CFD models, items are commonly grouped together so that the same boundary conditions can be applied and colouring in the same way like this makes it easier to check what boundary conditions are applied to each zone of the model. Figure 2d is the more realistic visualisation that it preferable when presenting the results of the simulations. There is a general move to making visualisation of CFD models and results more realistic with both texture and colours chosen to be the same as the real objects.



(a) Colour all items the same

(b) Colour all faces a different colour

(c) Colour items of the same type

(d) Colour items as they would appear naturally

Fig. 2. Colouring to illustrate a multi-object system

5. To illustrate the very large quantities of numerical data

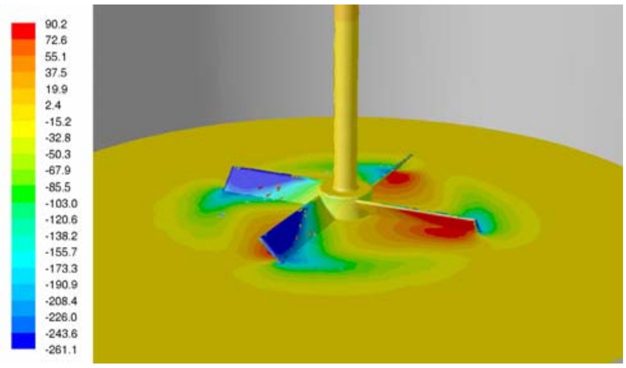
A critical step in a CFD analysis is the efficient use of the computed data and this can be undertaken in a variety of ways. Once geometry is confirmed and simulations undertaken the main use of colour is then the presentation of the data from the simulations. The engineer will normally want some numerical data such as a drag coefficient or pressure force that allow one design to be compared with another. The graphical post processing however can be used to identify and illustrate why. For example, the separation points or recirculation that generate the pressure drop or drag effects might be of interest and graphical post processing is valuable in allowing the engineer to identify key regions where the geometry can be modified to improve the design.

For the simplest of 3-D turbulent flows, the fundamental variables are the three velocity components, the pressure and at least two parameters that represent the turbulence. There are therefore at least 6 parameters stored at millions of data points in most simulations. For unsteady flows, these data exist at each point in time and visualisation in time and space.

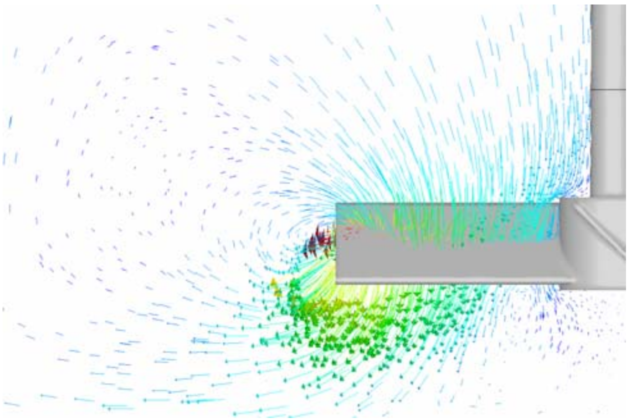
In CFD we use a range of post-processing objects to illustrate the flow as illustrated in Figure 3. These include:

- contours
- vectors
- iso-surfaces and iso-volumes
- pathlines and particle tracks

These are used by a CFD engineer to illustrate the key features of the flow. The virtual world of the CFD model relies on these four key objects, but the results would be naked without the colour that goes with them.

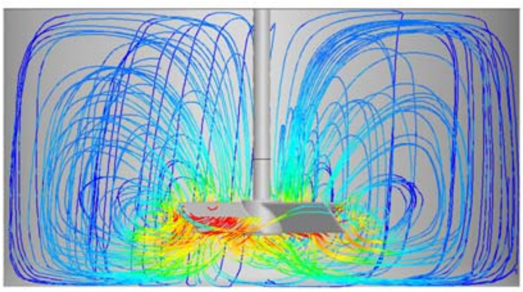


(a) Contours are used to illustrate the magnitude of a variable on the surfaces on which it is plotted. Here we see the high and low pressure regions in front and behind the impeller blades in a mixing tank. Red indicates the highest pressure and blue indicates the lowest pressure and a linear variation in colour is used to represent the range in between.



(b) Velocity vectors are used to illustrate the flow direction and magnitude at each point in the mesh. The arrow indicates the flow direction and the length and colour indicate the magnitude.

(c) Iso-surfaces are used to identify regions within which a variable is in a specified range. In this example, the regions of low pressure behind the impeller blades have been identified and plotted. Colour, shading and lighting allow the region to be visualised.



(d) Path lines illustrate the paths the individual fluid particles travel within the flow. These path lines indicate the formation of a general recirculation flow pattern within the tank. The colour of the lines indicates the velocity magnitude.

Fig. 3. Colour is used in contour plots, velocity vectors, iso-surfaces and path-lines to illustrate the magnitude variation of solution variables.

The value of colour is that it works with the key CFD post-processing objects to allow the engineer to really understand the flow of interest. Visualisation in a variety of ways allows the engineer to understand the flow paths, recirculations, and other complex flow features that result in the parameters of interest.
Once the flow is clearly understood, the analyst needs to communicate this to managers, designers and other interested parties. Here a few well chosen colourful graphical illustrations is all that is needed to share the knowledge of the complex simulations and gain maximum value from the data obtained.

6. Further illustrations of the use of colour in CFD

6.1 Vortex in a mixing vessel

Figure 4 shows the results of a multiphase CFD simulation of an air vortex in a mixing vessel. In this example a swirling flow is generated in a vessel using the rotation of a radial blade impeller. This rotation is seen in the pathlines seen on the base of the vessel. Air is drawn down at the axis of the impeller and some air gets held up behind the impeller blades as seen by the iso-surface that indicates the surface separating the air and the water. In addition, a recirculating toroidal vortex is setup as illustrated by the velocity vectors. Turbulence is also generated and this is strongest just downstream from the impeller blades. Mixing in vessels like these is an important area in CFD applications and the use of colour is invaluable in illustrating the reasons for good and bad performance of mixing equipment for specific processes. Further information may be found in reference [1] which discusses the wide range of CFD simulations that can be done to investigate mixing process. Also reference [2] provides a comprehensive guide to visualisation of computational fluid mixing.

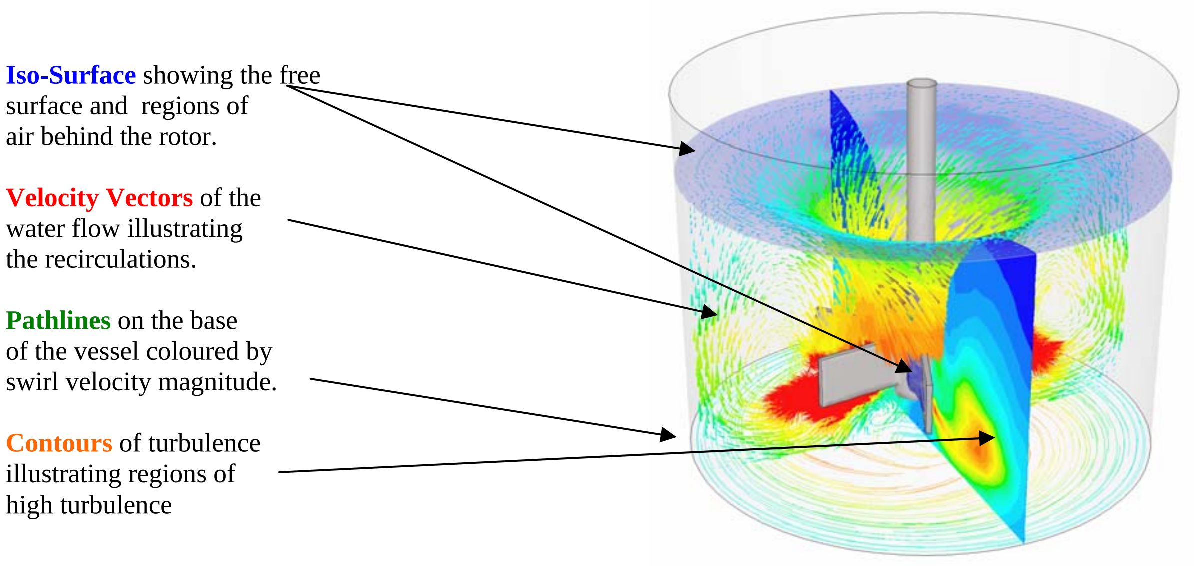


Fig. 4. Draw down of an air vortex in a mixing tank

6.2 Design of flow through blood vessels and vascular stents

Through a series of investigations we are specifically looking at issues of design in vascular surgery and arterial devices. The vessel shown in Figure 5 represents a portion of the iliac artery that naturally has a helical path for physiological purposes. Whilst arteries are helical *in vivo*, a straight artery would promote a narrower range of Wall Shear Stress (WSS) values whereas clinicians associate areas of particularly low or high

WSS as regions prone to diseases such as atherosclerosis [3]. Therefore a CFD model can help with surgical planning in identifying the best trajectory and assessing the likely impact of geometrical changes due to a graft by virtue of the WSS distribution. In this case only the lightest blue and lightest green areas represent values of WSS that are not associated with susceptibility to disease, which are very difficult to discern in greyscale.

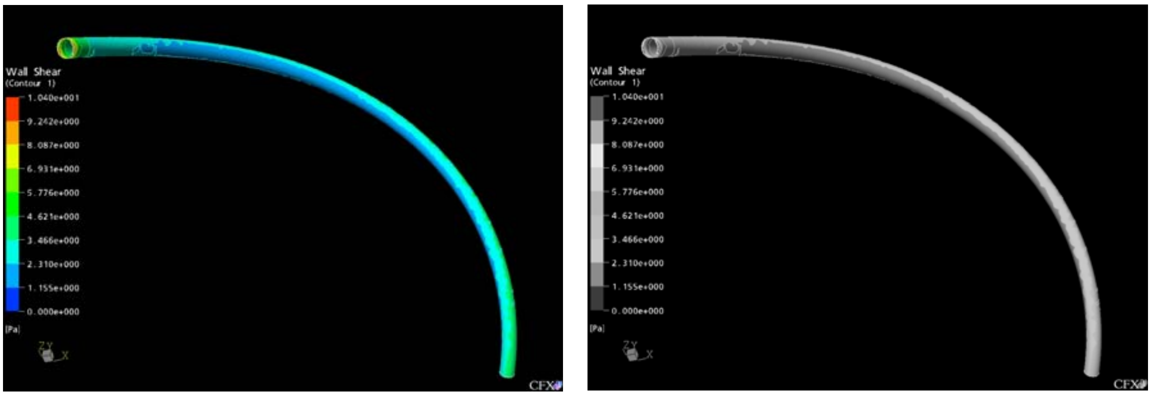


Fig. 5. Wall Shear Stress distribution over a portion of the iliac artery (in colour and greyscale)

Stents mechanically support diseased arteries that would otherwise restrict flow to vital organs. However, in some cases the effect of the stent on blood flow, and in turn the artery wall, can stimulate further disease. Improved stent designs can be investigated using CFD to model the flow characteristics of blood as it passes through the stent [4].

Figure 6 shows the velocity streamlines of the fluid as it passes over the stent wires. It is important to identify areas of recirculation and stagnation, as these are closely associated with atherosclerosis. In colour the velocity values are more readily discerned whereas in greyscale velocity values are virtually indistinguishable.

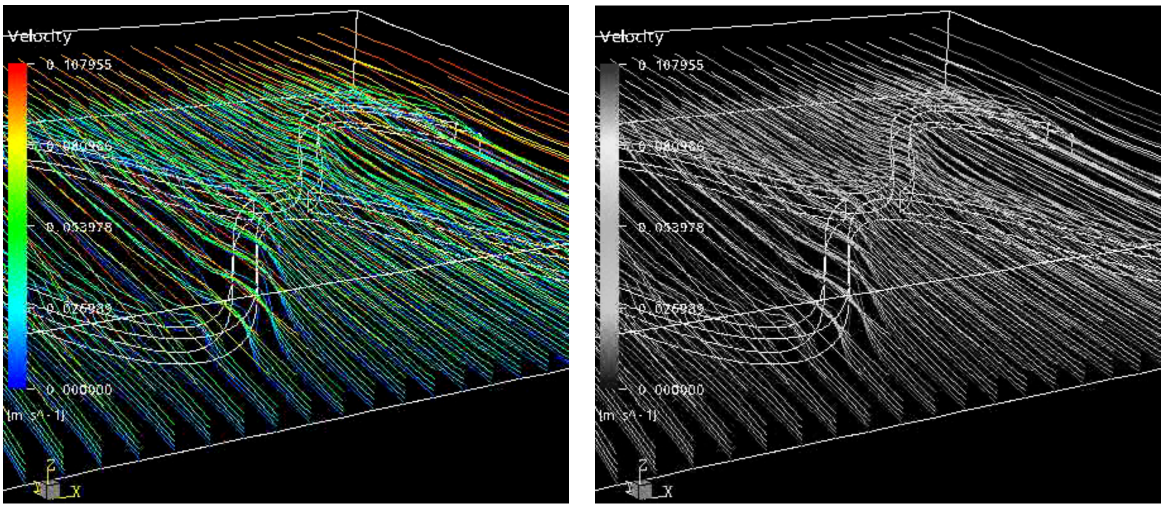


Fig. 6. Velocity streamlines over stent wires in colour and greyscale

The general aim is to seek designs that minimise disturbed flow, flow separation, vortices, high oscillatory shear and, as previously stated, excessively low or high average Wall Shear Stress values. If colour is so important in steady flow then it is even more so for time varying flows. The multidisciplinary nature of modelling medical flows means that the CFD results are often explained to non-experts and colour graphics are very effective in such cases.

7. Conclusions

Several geometrical objects are used in CFD to illustrate flow patterns and flow variables obtained from computational analysis. Critical to illustration of the flow features is the colour that is used to maximise the data conveyed by a single image.

Colour in CFD is invaluable for:

- Understanding the complex flow phenomena that exist in real engineering situations.
- Providing convincing illustration of flow phenomena.
- Minimising the time and effort needed to describe the flow of interest.
- Reporting the results of simulation in an easy to understand format.

Acknowledgements
Extended information on the mixing vessel work can be obtained from Fluent.

References
[1] Marshall E.M., and Bakker A., Computational Fluid Mixing, Pub. Fluent Inc, 2002.
[2] Bakker, A., The Colorful Fluid Mixing Gallery, http://www.bakker.org/
[3] Garasic, J.M., Edelman, E.R., Squire, J.C., Seifert P., Williams, M.S. & Rogers, C., Stent and artery geometry determine intimal thickening independent of arterial injury. Circulation, Vol 101, pp 812-818, 2000.[4] Atherton, M.A., Tesch, K. & Collins, M.W., Effects of stents under asymmetric inflow conditions, Biorheology 39, pp501-506, 2002.