Cambridge University Press & Assessment
978-1-107-42883-6 — Advanced Computational Fluid and Aerodynamics
Paul G. Tucker
Excerpt
More Information

CHAPTER 1

The Need and Methods for Studying Aerodynamics

The age of chivalry has gone. That of . . . calculator's has succeeded and the glory of Europe is extinguished forever.

E. Burke (1972)

1.1 Introduction

According to Slotnick et al. (2014), in 2025, air transport will contribute 1.5 billion tonnes of CO_2 emissions annually. Of course, there are many other fluid dynamic engineering systems and processes that contribute to environmentally damaging emissions. Hence, it is critical to study aerodynamics with the view to creating more energy-efficient systems.

Broadly speaking, the design engineer has the following three options for informing decisions regarding aerodynamics:

- Measurements
- Analytical solutions
- Computer simulations

1.2 Computational Fluid Dynamics

Since its inception around the 1960s, computer based simulation has revolutionised fluid mechanics. The key area of computer simulation in fluid mechanics is called Computational Fluid Dynamics (CFD). It is generally taken as involving the solution of the full equations governing conservation of mass, momentum and energy in a fluid, with generally little simplification to these equations. Currently, mostly it is still best for a company to take advantage of all three of the above techniques. The development of CFD has been most strongly driven by the aerospace industry. CFD's name has sometimes been partly tarnished, with suggestions that the acronym should be standing for 'Cheats Frauds and Deceivers' or 'Colourful Fluid Dynamics'. As with any experimental method, the use of CFD often requires extreme skill. CFD codes cannot be run by engineers (focused on extremely complex multidisciplinary design tasks) lacking some specialist training and without clear guidance. The modelling of turbulence has limited the accuracy of CFD, rendering the need for comparisons with test data (validation). The uncertainty in modelling turbulence has made CFD almost a postdictive method. However, with the power of modern computers, eddy-resolving techniques (the nearly exact flow equations solved to high accuracy) are beginning to emerge. Hence, we are beginning to move into a predictive era of CFD. In the future, the way in which CFD is used is likely to change substantially.

Generally with CFD simulations, the fluid flow field is divided into numerous small cells. Processes for carrying this out are described in Chapter 3. The flow-governing equations, described in Chapter 2, are solved locally for each of these cells. The solution process ensures mass, momentum and energy conservation between the cells. The geometry of the system under study can be taken from Computer-Aided Design (CAD) information (see Chapter 7), and generally geometry changes, relative to experiments, can be easily made. Key values of CFD are:

- It is an effective means for the rapid evaluation of what-if design scenarios.
- Geometry changes can be relatively easy.
- Safe for dangerous experiments.
- Can be performed at full-scale conditions (Reynolds and Mach numbers).
- Can be linked to formal design optimization procedures.
- Sensitivity analysis can be performed.
- Computer flow visualization and analysis techniques allow relatively easy analysis of flows to provide a deeper flow physics understanding.

CFD is not necessarily purely used for what we traditionally think of as design, but also sees much use in suggesting solutions to existing deficient designs through analysing them. The use of CFD, as with other modelling tools, should increase understanding of the design problem by the rigorous requirements imposed by mathematical modelling. In addition, the use of computational modelling tools can enable radical new designs (which may otherwise be disregarded due to their departure from established designs) to be assessed relatively inexpensively without the cost of manufacture and testing of a new design. Unlike most stress analysis used for solids, CFD involves solving non-linear coupled equations. Hence, CFD requires considerable rigour to be applied correctly. Next, to further contrast their merits, the traditional approaches to studying fluid flows was be described in more detail. First experimental methods will be considered, followed by analytical analysis and then CFD.

1.3 Experimental Methods

When considering experimental methods and aerodynamics, one frequently thinks of wind tunnels. With these one would aim for geometric similarity (i.e. ensuring the model is exactly to scale) and also, through matching the Reynolds number, hopefully some degree of dynamic similarity, that is, $Re_M = Re_A$, $\rho_M U_M L_M / \mu_M = \rho_A U_A L_A / \mu_A$. The implication of this, assuming a wind tunnel is being used to study the performance of a model (the 'M' subscripts identifying this) for an actual system (subscript A) that works in air (hence the fluid properties will essentially cancel), is that for a 10th-scale model, the wind tunnel fluid velocity must be 10 times that for the actual system. Hence, if we wished to study the aerodynamics of a car going at 100 kilometers per hour (kph), with a 10th-scale model, the wind tunnel air speed would need to be 1,000 kph. This speed, however, is very close to the speed of sound, and local fluid accelerations and decelerations would produce shock waves. Also,

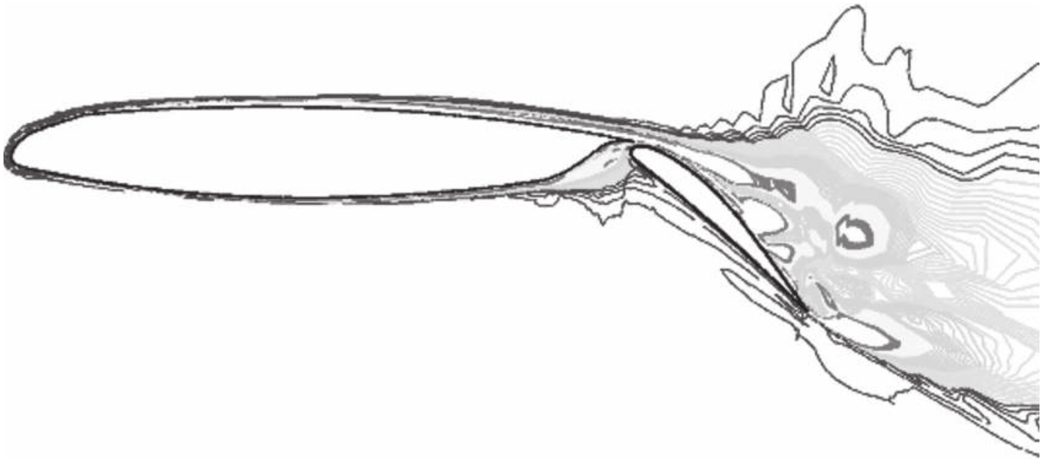

Figure 1.1. Vorticity contours for high lift configuration (from Tucker (2006) – published with kind permission of Wiley).

even if this did not occur, the air would undergo significant compression. Hence, the wind tunnel results would be inaccurate. In fact, for most model tests it is virtually impossible to get a model Reynolds number as high as that for the actual system. As well as the Mach number problem, there is also the issue of having a sufficiently large wind tunnel. Wind tunnels are very expensive to maintain and run and require highly expert staff. There are cheaper ways of achieving higher Reynolds numbers, but these have limitations and so are not discussed here.

As well as high running costs, the costs of manufacturing a scale model, in a large industrial facility, can be high. For example, a half-wing model costs around £300,000. The models need to be made from special steels. For example, the load on the mount connecting the model to the tunnel needs to withstand a force of around 25,000 N. Accuracy needs to be high, with $\pm$15–50 μm model accuracy. High-accuracy machining and testing facilities are required in addition to the actual wind tunnel and instrumentation facilities. Of course, the model needs to be instrumented with at least numerous pressure taps and these taps need to be connected to data loggers without connecting tubes impeding the air flow. The presence of an engine on a wing and the propulsive thrust from this can also dramatically influence the wind tunnel results. Hence, it is wise to include an engine simulator. These cost about £50,000 each. Thus, a wind tunnel test programme for a particular model can cost in excess of £1 million. In this context it is not difficult to see why CFD is viewed as relatively inexpensive.

With this high cost one might expect high accuracy. However, this is not always the case. For high-speed flow configurations and also high-lift configurations the wind tunnel blockage effect can significantly disrupt results. Figure 1.1 shows vorticity contours for a CFD prediction of the flow over a high-lift configuration. Around a $\pm$20% variation in predicted lift can be found when using different established CFD turbulence modelling practices. However, around a 10% variation can also be found due to the wind tunnel wall influence. This figure is ascertained through numerical

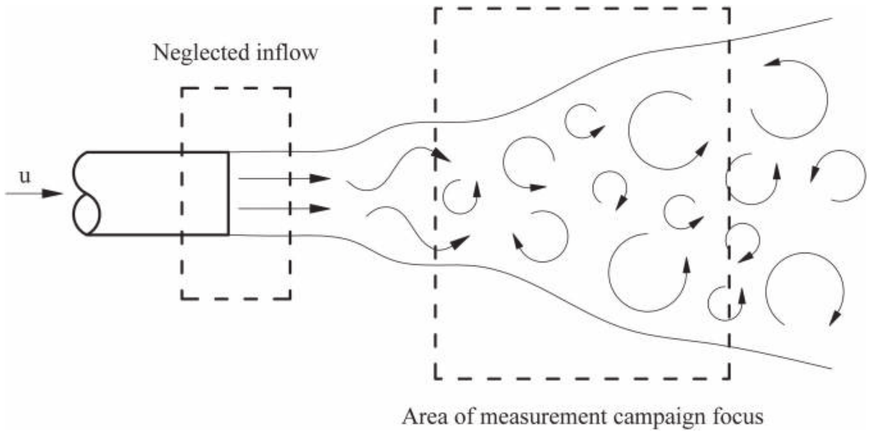

Figure 1.2. Lack of boundary condition definition for measurements.

studies. Hence, experiments do not always yield high-accuracy data. Heat transfer can be especially difficult to measure accurately. When using scaled models to study the dynamics of ships and other free-surface flows, the Froude number must also be matched. It is very rare that one can match both the Froude and Reynolds numbers.

Typically in a wind tunnel, global lift and drag will be measured along with local surface pressures. Also, to gain an idea of general flow patterns and separation points, tufts or a film of oil can be placed on surfaces. However, typically with experiments the level of data points and detail is limited relative to CFD.

A summary of the abilities of experimental methods might be that they can give accurate results at a limited number of points in space, using equipment that is often expensive to purchase and requires skill to calibrate and use successfully. Whilst time variation can be measured, the techniques often disturb the flow. A key problem with experiments is also that generally there is always some uncertainty regarding boundary conditions. This can be wind tunnel wall influences (as noted above) or can be inflow, to name but a few uncertainties. Figure 1.2 illustrates a typical problem with many experimental campaigns, but this time relating to, say, a propulsive jet. Typically the objective would be to study, say, the dynamics of the flow downstream the nozzle exit. However, this leads to the situation where most data sets for jets neglect to measure the inflow boundary conditions such as the boundary layer thickness at the inflow and the turbulence levels across the inflow. This can be a critical problem for the assessment of the CFD and indeed even with regards to the value of the measurements themselves.

Another, interesting aspect with jets is that they can be surprisingly sensitive to their surroundings, and this can greatly complicate comparisons with CFD. Essentially this is again a boundary condition issue. There are further complications with measurements, this being the aspect of repeatability. Shock locations, for example, can vary greatly between wind tunnel runs. Hence, modern measurements tend to assess repeatability. Clearly this aspect relates to uniqueness of flow solutions, and therefore a predictive CFD method should suffer from the same problem, namely a

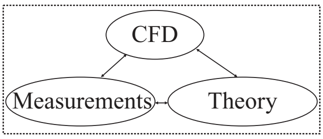

Figure 1.3. The fluid dynamics triad.

lack of repeatability as the differing runs find the different solutions, and replicate any possible physical modulations between solutions. This behaviour is certainty true for real systems.

Independent of all of this, if a number of flow situations have to be considered – say, for a number of physical geometries – experimental methods need new physical geometries to be created and then tested. This of course increases the time and expense of testing nearly linearly. Notably, however, rapid prototyping techniques are emerging, and this raises some interesting questions of how CFD and measurements might work together in the future. If high surface finish is not needed, components can be manufactured cheaply and relatively quickly using three-dimensional printers. These can even produce metallic components. This can be achieved for a range of metals (for example, stainless steel, aluminium and titanium), through the process of laser sintering. Under some circumstances, three-dimensional printing measurements might be quicker than CFD. Another interesting new area is where the CFD simulations and measurements are performed concurrently in various linked ways.

1.4 Analytical Solutions

Analytical solutions are important for gaining ballpark figures for design calculations. Generally analytical solutions essentially consist of exact solutions to simplified flow governing equations with simplified boundary conditions. Analytical solutions can be useful for verifying the soundness of CFD programs.

Notably, analytical equations can find themselves embedded in CFD programs or processes. For example, they can be embedded as models for variable distributions at the elemental level. Examples of these can be found in both temporal (see Patankar and Baliga 1978) and spatial schemes (see Patankar 1980) or for post-processing. For the latter this can be for predicting far-field sound where analytical solutions are used to connect near-field sound level data from the CFD to the far field (see Williams and Hawkings 1969). He and Oldfield (2011) use a one-dimensional analytical conduction of heat solution to connect the solid and fluid domains when performing conjugate simulations. The extent of analytical solutions is vast, and they are not outlined here, but the use of some for code assessment purposes will be found within this text. Critically, both analytical and numerical models seek to accurately solve flow-governing equations. Hence, the flow-governing equations in their wide range of forms are discussed in the next chapter.

As noted, traditionally, CFD is best used in conjunction with measurements and theory/analytical solutions in the triad shown in Figure 1.3. However, as noted

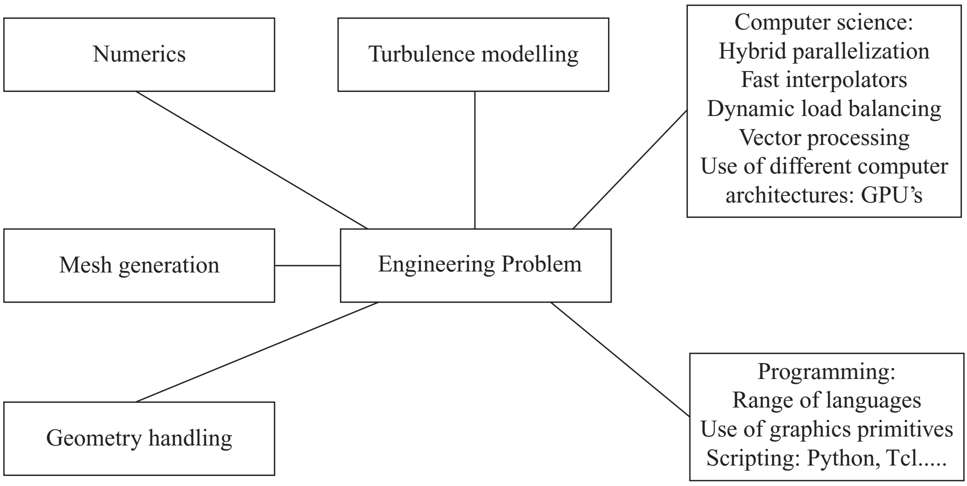

Figure 1.4. Aspects that need to be considered when bringing CFD to an engineering problem.

earlier, as simulation becomes increasingly predictive, the expectation is that the need for input from measurements is reduced.

1.5 CFD in Aerospace

Aircraft manufacturers use CFD for a wide range of aspects of design. These include flutter, fuel systems, sting corrections, cabin ventilation, cockpit ventilation, avionics cooling, ice prediction, thrust reverser design, nacelle design, auxiliary power unit inlet/outlet design, aerodynamic loads data, belly fairing design, tail design, wing design, fuselage design, power plant integration, spoiler control surfaces and flow control design such as vortex generators. Aeroengine manufacturers also use CFD for a similarly wide range of activities including even oil fire modelling.

Figure 1.4 shows aspects that need to be considered when bringing CFD to an engineering problem. As can be seen, these are wide ranging and go well beyond just the consideration of numerical methods. As outlined in Chapter 2, care needs to be exercised when choosing the most appropriate form of governing equation. This equation then needs to be integrated over grid cells. The aspect of mesh/grid generation in discussed in Chapter 3. Numerical schemes formulated around a particular grid are discussed in Chapter 4. Turbulence modelling is dealt with in Chapter 5. Geometry handling is discussed in Chapter 6. The right-hand boxes in Figure 1.4 are largely associated with computer science. This is an area of ever-increasing importance that needs to be considered as part of CFD provision. Chapter 7 discusses pre- and post-processing. Further to what is outlined in Figure 1.4, CFD is becoming an increasingly multi-physics, multi-scale, coupled system endeavour. For example, as noted by Spalart and Bogue (2003), for aircraft, the ultimate aim would be simulations of the coupled, aircraft, propulsive unit, and pilot interactions. The aeroengine manufacturers have similar pressing needs for increasingly coupled simulations.

This is especially so considering that future aircraft will need increasing integration between the airframe and the engine, ultimately with the engine blended into the aircraft. Hence, Chapter 6 looks at advanced simulation techniques. It explores the ingredients needed for many current complex engineering calculations and some future requirements. Notably, multi-scale simulations are of some importance, since the most effective future flow control techniques probably need to be effective in the very inner parts of boundary layers. This gives length scale disparities of many orders of magnitude for high-speed aerodynamic flows. Finally, Chapter 8 looks at what CFD might look like in the next 10 years and beyond.

At the end of most chapters some problems are suggested. Some of these direct the interested reader to the writing of some elementary flow solvers. One is a compressible Euler solver and the other one uses techniques related to incompressible flows. The former utilizes a more general grid structure, whilst the latter has strict cylindrical/Cartesian coordinates. The codes can be validated using analytical solutions, some of which are given in the problems section of Chapter 2. The governing equations in Chapter 2 inform the reader about the equations needed for these codes, and the questions direct them towards the task of initializing a compressible flow field and setting some boundary conditions. The meshing chapter gives the opportunity to write a very simple mesh generator for the above. The numerical methods chapter gives ideas for approaches to generating the systems of equations arsing from integrating the governing equations around the meshes and advancing them, where appropriate, through time. Chapter 6 suggests some more complex potential uses for this solver and further tasks. The pre- and post-processing chapter allows further opportunities to explore the output from the above programs. Appendices B & C provide some information to help with the code development tasks.

REFERENCES

HE, L. & OLDFIELD, M. 2011. Unsteady conjugate heat transfer modeling. *Journal of Turbomachinery*, 133, 031022.

PATANKAR, S. 1980. *Numerical heat transfer and fluid flow*, CRC Press.

PATANKAR, S. & BALIGA, B. 1978. A new finite-difference scheme for parabolic differential equations. *Numerical Heat Transfer*, 1, 27–37.

SLOTNICK, J. P., KHODADOUST, A., ALONSO, J. J., DARMOFAL, D. L., GROPP, W. D., LURIE, E. A., MAVRIPLIS, D. J. & VENKATAKRISHNAN, V. 2014. Enabling the environmentally clean air transportation of the future: a vision of computational fluid dynamics in 2030. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 372, 20130317.

SPALART, P. & BOGUE, D. 2003. The role of CFD in aerodynamics, off-design. *Aeronautical Journal*, 107, 323–329.

TUCKER, P. G. 2006. Turbulence modelling of problem aerospace flows. *International Journal for Numerical Methods in Fluids*, 51, 261–283.

WILLIAMS, J. F. & HAWKINGS, D. L. 1969. Sound generation by turbulence and surfaces in arbitrary motion. *Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences*, 264, 321–342.

CHAPTER 2

Governing Equations

2.1 Introduction

Care must be taken regarding whether the chosen flow-governing equations are applicable. For most flows the Navier-Stokes equations are adequate. However, for example, Microelectromechanical systems (MEMS), which have potential for flow control, have a characteristic system length scale, L, range such that 1 mm $>$ $L > 1$ micron. This extreme range of scales creates physical modelling problems. The continuum assumption no longer fully holds and the no-slip condition at solid walls is no longer valid. Satellites in the exosphere are another system where the continuum assumption does not hold. The dimensionless parameter used to tell if the continuum assumption still holds is the Knudsen number (Kn). This is the ratio of the mean free molecular path, $\tilde{L}$ and L. Hence,

$$Kn = \frac{\tilde{L}}{L} \tag{2.1}$$

Generally the fluid can be assumed a continuum for $Kn < 0.1$. However, even then the no-slip surface boundary condition needs to be modified. For $Kn = 0$ the Navier-Stokes equations reduce to the Euler (see Section 2.2.2.1).

It can also be important to consider the Damköhler number:

$$Da = \frac{Chemical\ Reaction\ Rate}{Convective\ time\ scale} \tag{2.2}$$

For finite Damköhler numbers, it can be necessary to consider finite rate chemistry in the Navier-Stokes equations (see Section 2.2.2). Also, for non-Newtonian fluids, altered stress-strain relationships are required and so attention needs to be paid to such matters as well as the formulation of the Navier-Stokes equations themselves. For dynamically complex flows, the precise formulation of the Navier-Stokes equations used, when discretized, can have a strong impact on solutions.

Another aspect is the mathematical nature of the flow-governing equations. The equations presented below can be classified as elliptic, parabolic or hyperbolic. This classification is helpful in pointing to the most appropriate numerical scheme necessary to solve the governing equation. Even though, as mathematically defined, governing equations can have a single identifier, typically, in a physical sense equations can have a mixed nature. For example, an equation might be parabolic in time and elliptic or hyperbolic in space. The parabolic temporal nature reflects the fact

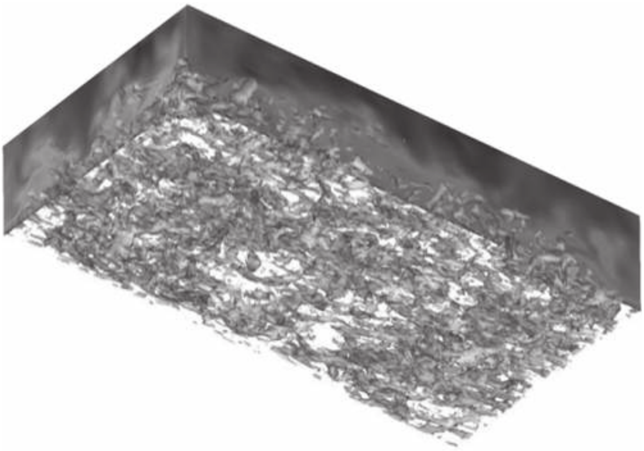

Figure 2.1. Isosurface of constant vorticity from numerical solution of the Navier-Stokes equations.

that future events have no impact on the past. Some equations can be formulated to accord with a physical reality so that this is also true spatially. The diffusive terms of the equations are elliptic in nature. This implies that the solution at a point is a function of information at all the surrounding boundary points. For an equation that is parabolic in space the downstream information is just dependent on upstream data. Hyperbolic equations are typically associated with high speed flows with shock waves. With these the regions of flow influence between points in a flow are more complex. Events in multiple distinct zones in a flow can disconnected in multiple directions.

Some texts refer to the combined set of the mass conservation (continuity), momentum and energy equations as the Navier-Stokes equations. Following, here generally the Navier-Stokes equations are taken to refer to just the full form of the momentum equation, including viscous-force terms. The full Navier-Stokes equations present a highly complete description of a fluid flow. In fact, nowadays, exact solutions of them are generally regarded as more accurate than measurements. As noted in Chapter 1, for the latter there are always experimental uncertainties with regards to, for example, boundary conditions and also the intrusive nature of the measurement probes. There are also more subtle issues as outlined in Chapter 1. Figure 2.1 shows instantaneous flow variable contours, from a solution of the Navier-Stokes equations, for the turbulent flow in a plane channel. The experimentally observed streak structures in the boundary layers are apparent in the solution. Unfortunately, as might be suggested from the Figure 2.1 flow complexity, analytical solutions of the Navier-Stokes equations are difficult, if not impossible, for all but the most basic, and generally steady, laminar flows. As will be shown, the Navier-Stokes equations are large non-linear equations and thus difficult to solve analytically. Hence, Figure 2.1 is a numerical solution of the Navier-Stokes and continuity equations. Figures 2.2 and 2.3 give numerical solutions for the Navier-Stokes and continuity equations for an airfoil at a high angle of attack, and the flow through

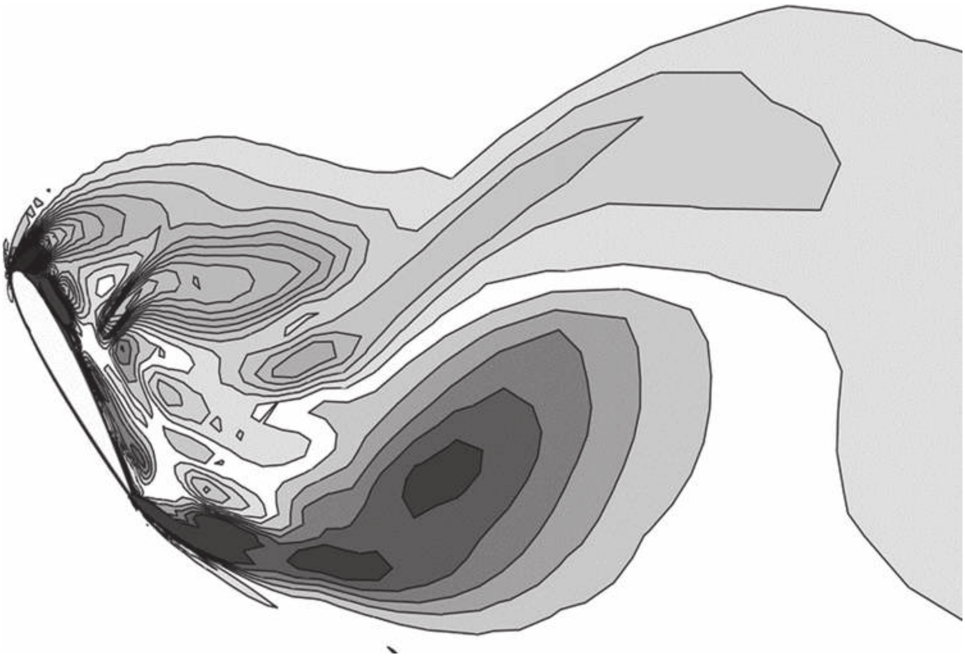

Figure 2.2. Vorticity contours from numerical solution of the Navier-Stokes equations for a NACA0012 wing at a high angle of attack. (from Tucker (2006) – published with kind permission of Wiley).

an aeroengine (bypass duct) mounted to a wing with an idealized flap. As can be seen, these are all rich in flow details.

Figure 2.4 is a flow chart showing the Navier-Stokes and related equations for increasing levels of simplification. In this chapter, these variants of the Navier-Stokes equations will be explored. First, a simple derivation of the Navier-Stokes

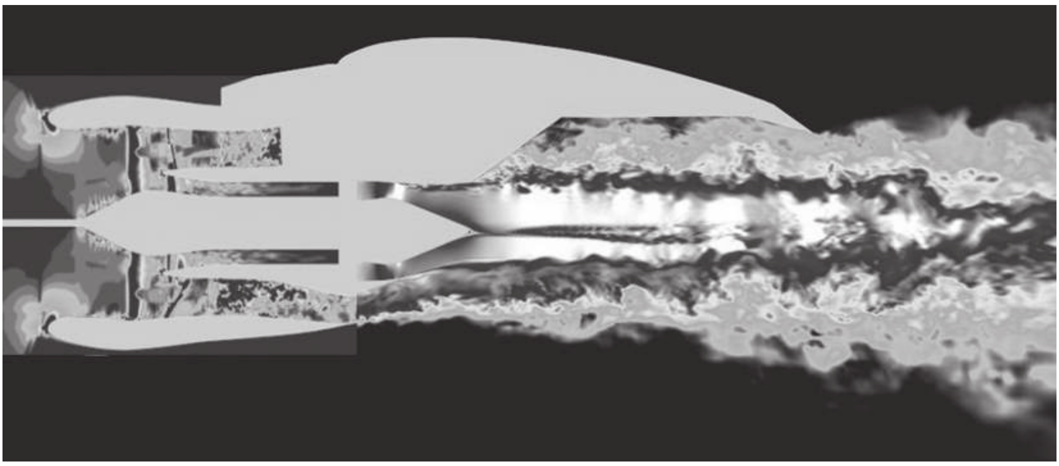

Figure 2.3. Instantaneous flow from solution of the Navier-Stokes equations for the flow though the bypass duct of a gas turbine aeroengine. (Adapted from Tyacke et al. (2015) – published with kind permission of Springer)