Computational Modelling of Unsteady Aircraft Aerodynamics Flows

Bryan E. Richards, Ken J. Badcock, George Barakos, Mark Woodgate, Jason Henderson, Daniel Feszty, Mark Allan, Ryan Menzies

CFD Group, Dept of Aerospace Engineering, University of Glasgow
b.richards@aero.gla.ac.uk

ABSTRACT: This paper reviews simulations using Computational Fluid Dynamics (CFD) of unsteady flow applications relevant to aircraft aerodynamics. Key elements of the CFD methodology are described which include a description of parallel multiblock (PMB), the in-house unsteady viscous flow code and the powerful cluster computing environment that has been constructed to tackle demanding problems successfully. The flow cases described include transonic flow over a cavity, the pulsating flow over a spiked blunt body, vortical flow over a manoeuvring delta wing, surge wave in an engine intake duct and dynamic stall over an aerofoil. The paper demonstrates the powerful nature of CFD in helping to understand unsteady flow processes.

INTRODUCTION
In the most prominent areas of fluid dynamics research in aerospace being carried out today (e.g. flutter, buffet, rotorcraft flows, gas turbines), the study of the unsteady behaviour of fluid flows plays an important if not dominating role. For fixed wing aircraft or spacecraft, unsteadiness is often undesirable, such as in the cases of cavity flows and spiked blunt body flows to be described later. In some cases unsteady flow is induced such as in manoeuvring vehicles and a case of flow over a delta wing is described later. In other cases unsteady flow is endemic such as helicopter and propulsion flows. Research in unsteady flows is concerned with understanding the processes either to reduce or eliminate unsteadiness or to exploit it.

Much of the past work on unsteady flows has been accomplished using experimental methods. There are many powerful experimental techniques for the study of fluid flows but achieving high spatial and temporal resolution of all quantities of a flow region is prohibitive, but necessary for its understanding. Surface pressure measurements require individual fast response sensors with high frequency data processing equipment and their placing and sheer numbers can make it difficult and expensive. Measurements in the flow field are even more difficult since it is usually impossible to attain the level of miniaturisation required to avoid affecting the flow. Pressure visualisation techniques can help with the spatial resolution but suffer from lack of response and also can affect the flow. The most sophisticated measurements are associated with hot wire thermometry and laser Doppler velocimetry leading to PIV and some interesting results from the latter are now becoming available.

With the availability of advanced CFD techniques and low cost major computing facilities we have potentially powerful tools for studying flow unsteadiness in aircraft design. At a level of Reynolds averaged Navier-Stokes (RANS) with turbulence modelling, CFD is capable of implementing design problems on complete aircraft in unsteady conditions. The implication for a reliable design solution is that the physics of the phenomena has been modelled adequately and that the resulting equations have been solved numerically accurately. This is not straightforward and the confidence of a designer needs to be boosted usually by means of some wind tunnel simulation of the problem in the process. Thus the wind tunnel approach to design has not in anyway been superceded.

An important ingredient in a designer’s repertoire towards an efficient design is an understanding of flow processes. This understanding leads to intuitions that be brought into play in this process. With increasing complexity of design to achieve even more efficient configurations, these intuitions and the connected understanding of the flows become more desirable. This paper describes the use of a powerful CFD system validated against experimental results to provide increased understanding of some complex unsteady flows.

METHODOLOGY
Parallel Multiblock

The parallel multiblock (PMB) code developed in-house is based on implicit methods for compressible flow solvers [1]. The objective has been to provide a high quality flow solver for the examination of real problems of scientific or engineering interest. The PMB development has drawn on the expertise and experience in the CFD group at the University of Glasgow in the areas of compressible flow solvers, implicit methods, transonic and high speed applications and parallel computing.

There are presently two versions of the code, the two-dimensional (and axisymmetric) flow solver PMB2D and fully three-dimensional flow solver PMB3D Both these include the following features: Euler, Navier-Stokes or RANS equations; cell-centred, finite volume method; Osher’s scheme and MUCL3 variable interpolation for convective fluxes; structured, multi-block grids; Newton-type implicit time stepping; approximate Jacobian matrices for improved convergence and reduced operation count and storage. The linear system is preconditioned using a Block Incomplete Lower-Upper factorisation, and solved using a Generalised Conjugate Gradient Method. It can be used for steady or unsteady flow; unsteady flow employs an implicit dual-time method. Parallel implementation is through PVM or MPI using coarse-grain domain decomposition.
The flow solvers have been applied to a very wide range of flow conditions and geometries. For PMB2D applications include: inviscid, laminar, turbulent aerofoil sections; pitching and ramping aerofoils; inviscid, laminar, turbulent foerebodies; multi-element aerofoils; turbulent base flows; supersonic compression ramps; nozzle flow; highly underexpanded jets; unsteady cavity flows; steady and unsteady aerospikes; and film cooling. For PMB3D they include: inviscid and turbulent wings; inviscid, laminar, turbulent ogive cylinders at incidence; pitching delta wings; complex 3-D aircraft geometries; cavity flows; surge waves in intake ducts

Parallel computing using PC clusters

Presently, any major computing carried out in aerodynamics is done using some level of parallelisation and a popular way of achieving this is using distributed memory architecture. This not only gains benefit from carrying out the task simultaneously on many processors but also enables a large amount of memory required to define problems of geometric complexity. Parallelism is achieved in CFD codes by domain decomposition with each processor calculating the solution on a partition of the whole grid. Then values need to be communicated to each processor corresponding to information at the boundaries of grid partitions. This message passing is implemented using a high level library such as MPI (message passing interface). The coding of partitioning and message passing does not constitute a major problem if the basic flow algorithm is well conceived. This is all built into the PMB code, solving the Reynolds averaged Navier-Stokes (RANS) equations with turbulence modelled, providing ease of use to the user.

Small academic research groups can have difficulty getting access to national computing facilities. Generally resource on these facilities is awarded to a group of collaborators who use common software on fundamental problems of physical complexity that have huge run times e.g. study of turbulence, ocean currents, meteorology etc. RANS calculations are mostly directed towards industrial problems that are large, mainly because of their geometrical complexity, but the speed derived from fast computers is directed towards a fast turnaround time needed for rapid design decisions. This along with the significant time in generating a grid to make changes of a geometric nature to the problem means irregular access to a facility that appears to be incompatible with the major users. Thus CFD users solving problems modelled using the RANS equations are forced to look for other solutions than using national facilities, such as a Beowulf facility.

The cost of even a powerful Beowulf facility is low enough that it is within the grasp of a small research group. Also when building such a facility the codes to be used on it can define its configuration. A small focussed group will have a minimum of compute intensive codes to implement on the facility, so that it can be configured simply. The CFD Group uses only the PMB code. This could also be said of teams within industry or small companies targeted at specialist areas in which the Beowulf would be an ideal facility. In other words small groups do not need the flexibility of a mainframe, and the purchase of one would not be economic. Also they would be free of restrictions imposed of corporate facilities.

One of the economies associated with the use of a Beowulf facility is the use of Linux as the operating system. This has been shown to provide a very robust operating system. It is a public domain software but which is now accepted as being reliable and also that commercial software is configured to run on it. For these reasons Linux and the Beowulf concept is becoming more widely acceptable.

Previous papers [1], [2] described the 16 node Pentium 200 cluster which was installed at the University of Glasgow in 1997. The attractions of such a facility are well documented and revolve around price and performance. The system has now been upgraded with the development of a 32-node cluster [3]. This new cluster was built for £40,000. The cluster’s internal connectivity is provided by a Cisco catalyst 3548 fast Ethernet switch which provides 48 x 100BaseTX full duplex ports and two proprietary expansion slots one of which is populated with a 1000BaseSX module giving full duplex gigabit Ethernet connectivity to the cluster’s main file server. The free expansion port can be utilised to chain additional Catalyst switches allowing the cluster to be scaled beyond its present configuration. The cluster’s main file server is a dual 700MHz Pentium III box with 512Mb of RAM based around the ServerWorks III-LE chipset and is currently running Solaris 7 x86. This is accessed via NFS auto-mounters on 7 different operating systems by client machines on the departmental LAN. The chipset features a 64-bit 66MHz PCI bus with a 200Gb file system used for data generated by processes on back end nodes. The back end nodes are made up of 32 x 750MHz AMD Athlon (Thunderbird) uni-processor machines each with 768MHz of 100MHz DRAM. All the nodes have 100Mbps full duplex 3Com 3C905 network interface cards. The Athlon based machines are running Mandrake 7.2 with a newer 2.4.0 kernel primarily for its NFSv3 enhancements. The only software used on the back end nodes is the Glasgow flow solver. The LAM MPI implementation is used for the message passing.

UNSTEADY TRANSONIC FLOW IN AN OPEN CAVITY

The study of open cavities, i.e. those of small length to depth ratios (L/D<7), in walls in high speed flows is presently the target of intensive studies because of the current need to launch stores internally from an aircraft. The present approach of stores carried externally causes a large radar cross section which is not acceptable for future stealthy aircraft. For certain configurations the acoustic environment within a cavity can cause damage to components within the store. Also launching the store through the shear layer and unsteady environment can cause flight instabilities. Thus the interest lies in understanding cavity flows, predicting them and also ideally controlling them so that the acoustic environment is improved.

The flow has been found to be complex with an oscillating shear layer and pressure waves interacting. There were two main explanations of the oscillation derived from experiment. The first suggested that the unsteadiness is due to vortices shed from the front lip of the cavity. When these are reflected from the downstream wall of the cavity a pressure wave propagates back upstream which causes the shear layer to separate, resulting in more vortex shedding. However the model does not explain how the pressure waves are generated at the downstream wall or how the shear layer is separated at the front tip. A second explanation emphasises the influence of pressure in the cavity. These waves drive the motion of the shear layer which in turn causes mass to be added and expelled from the cavity, causing the pressure
wave generation. The study aimed to explore these two descriptions. The study was made on a cavity of \(L/D = 5\) to 7 at various Mach numbers \([4], [5]\) but a representative case of \(M = 0.85\) with a unit Reynolds number of \(6.786 \times 10^6\) per m is chosen to illustrate the results. Calculations were made on a grid of 20301 cells. Time steps were chosen in order to capture the highest frequency oscillations expected and the calculation continued until the flow has settled and a representative spectrum of the unsteadiness was achieved. The success in modelling the flow is demonstrated in the agreement of the frequencies of discrete tones compared with the experiment and the formula of Rossiter \([6]\), accepted as a reliable indicator and the sound pressure level (SPL) obtained from analysis of the computations compared with experiment (see Fig 1).

![Fig 1. Variation of SPL along cavity floor for \(M = 0.85\) and \(L/D = 5\)](image)

As a result of the success of the prediction of the experimental data, then it was appropriate to use the numerical data of high resolution in space and time to attempt to understand the flow processes. The results were illustrated with pressure and vorticity field contours and streamlines. The flow is shown to involve the interaction of vortices in the cavity, which play a major role in the feedback mechanism. There are usually two vortices present in the cavity, though one is shown to dominate at any one time (see Fig 2). From a time that only one elongated vortex is seen, a second vortex is formed by splitting off from the former. This new vortex is strengthened by vorticity generated at the upstream lip of the cavity and later by the merging with the remnant of the preceding vortex as it progresses down the cavity. As it moves towards the aft wall it moves upwards resulting in the splitting of the vortex with some part escaping downstream and the remainder becoming the remnant, mentioned previously, in the aft cavity corner. This happens since the motion of the vortex eventually causes the shear layer to detach from the aft wall. The pressure field is affected by the low pressure felt in the region around the vortex core that travels down the cavity leaving regions of higher pressure between them. The pressure near the aft wall is subject to the highest pressure fluctuations as the shear layer fluctuates with the wall subjected in rotation to the stagnation of the freestream and the influence of the vortex and its escape downstream. These high level fluctuations are then propagated upstream both within the cavity and in the outer flow.

These upstream travelling waves then appear have an influence on the frequency with which the vortices are dispatched from the fore wall of the cavity. The way that the pressure and suction waves travelling upstream and downstream respectively interacting with each other goes a long way to explaining the discrete tones sensed and the particular SPL spectra that are measured at different positions on the cavity wall.

These observations derived from the numerical experiments have added understanding to high speed cavity flow problems.

**PULSATING FLOW OVER A SPIKED BLUNT BODY**

The family of spiked blunt bodies in high-speed flow has appeared in many areas of aerodynamics studies in recent years. The configuration had been selected to operate as a steady flow, such as a means for reducing drag of a blunt body, but at certain conditions serious instabilities have occurred. Applications also include axisymmetric propulsion unit intakes with centre bodies which have demonstrated a “buzz” type of unsteadiness, concave nose shapes formed by turbulent ablation on re-entry vehicles and use of concave nose shapes for planetary re-entry to create high drag have all shown instabilities. It is important to understand these phenomena and ways of alleviating them. Most studies to date have been experimental generating little field data, thus understanding has been incomplete. In this paper the more dramatic type of unsteadiness, called pulsation is illustrated.
An axisymmetric solution of the Navier-Stokes equations was used to investigate the problem of pulsation at zero incidence using PMB [7], [8] and [9] to investigate the experiments carried out by Kenworthy [10]. A view of the geometry is shown in Fig 3. A grid of 40,000 cells was used. A spike to diameter ratio (l/D) of 1.0 in a freestream of \( M = 2.21 \) with Reynolds number based on D of 0.12 million is illustrated in this paper although a greater variety of cases including hypersonic flows was studied. Turbulence effects were not taken into consideration in the calculations since for pulsation the driving mechanism was thought to be inviscid in nature as verified in this study. A verification study was carried out to determine the grid and time step size that would provide accuracy.

The results from the calculations were first validated against experiment. The pressure traces were compared with measured values as shown in Fig. 4. A portion of the experimental trace is included showing excellent agreement for the dominating frequency and good agreement with amplitude. Some selected graphics from Schlieren pictures from the experiment and density contours from the calculations also demonstrate both the nature of the flow and the good agreement between them. These validations against experiment justify using the results to examine the detailed mechanisms of the flow. During the collapse of the expanded foreshock, a vortical region is generated within the triple shock system when the foreshock meets the aftershock (Fig 5b,c). This region feeds the separation zone with gas, originating from the freestream, for only about a quarter of the time period of pulsation. When the growing and collapsing bow waves merge, the pressure at the afterbody face reaches a maximum (see point B-C in Fig. 4 corresponding to Fig. 5 b,c).

Also the pressure of the vortical region interacts with the merged shock leaves a residual of the vortical region associated with a strong pressure gradient in the upstream direction. This forces gas to be reversed rapidly into the separation region, from which it re-enters the supersonic stream behind the oblique shock via a shear layer. However this re-circulated flow is confined to the lower layers of the supersonic region by the flow coming from the freestream and

hence will again be entrained by the vortex. This means a repeated recirculation of the flow and also that the mass of the gas trapped in the separation zone will be fairly constant. Thus the original vortex and its residual pressure field have two different roles: the former one fills the separation zone with a certain amount of gas whereas the latter one ensures that this gas does not escape.

The combination of these three factors: a nearly constant mass of gas in the separation zone, a recirculation driven by the residual pressure field and the high pressure accumulated at the cylinder face will drive the inflation/expansion process. The constant mass of gas increases its volume (the volume of the separation zone) as the high pressure near the cylinder is relieved.

This leads to a lateral expansion of the foreshock and ultimately to its transformation to a bow wave (Fig. 5d-f). The high pressure arising at the bow wave will lift off the shear layer and allow the flow to penetrate below the separation region. However, it does not possess enough energy to stop the recirculated flow, so after they collide it turns back in a direction opposite to the freestream flow to enter the supersonic region behind the oblique portion of the foreshock. As the shear layer is lifted off, more streamlines become involved in this process. Therefore, the size of the high pressure subsonic zone behind the expanded foreshock grows until it overwhelms the recirculation zones resistance. Then the whole shock system collapses again.

This described driving mechanism represents a new explanation of the pulsation phenomenon. It has been enabled because CFD has offered an important advantage of providing insight into all regions of the flow field. The analysis of as many as four physical features (pressure, Mach number, instantaneous streamlines and velocity vectors) have been used [8] to provide a richness of information not presently imaginable using experiments.

It had been recognised that in early investigations the pulsation mode happens due to some sort of mass influx from the freestream into the separated region e.g. [11] or that a
supersonic jet directed towards the axis arising from the
Edney type IV interaction [12], [13]. These concepts also
assumed that the influx occurred during the whole of the
inflation process. The numerical analysis demonstrated that
although the phenomenon happens due to mass influx from
the freestream into the separated region neither of the latter
cases are correct.

DELTA WING FLOW

Delta wings are regularly used for military aircraft because of
their favourable aerodynamic characteristics in supersonic
manoeuvring flight. At high incidence the majority of the lift
comes from leading edge vortices although at extreme
incidence this contribution decreases due to vortex
breakdown moving onto the wing. During rapid flight
manoeuvres such as pitching, rolling or yawing then vortex
breakdown becomes dependent on the history of the
manoeuvre and so the aerodynamic loading behaviour
becomes quite non-linear. These flow phenomena need
understanding and CFD simulation can play a role. There is
also a need to develop the understanding of vortices and their
dynamics alongside wind tunnel tests, but it is known that

tunnel walls affect the data. CFD can simulate free flight and
wind tunnel constrained flow independently.

PMB3D has been used to investigate tunnel wall influences
on steady and unsteady flows [14]. Euler simulations have
been conducted on a 65° delta wing in wind tunnels of
varying cross section, in static and pitching conditions. Such
Euler simulations can adequately model the wall influences
on vortex breakdown due to the inviscid nature of the
phenomenon.

When a delta wing is placed inside a wind tunnel the leading
edge vortices induce an upward component of velocity as the
circulating flow impinges on the sidewall. This has the effect
of increasing the effective incidence along the leading edge
of the wing (considering the method of images, a tunnel wall
can be modelled with a vortex of equal strength but opposite
sense of the leading edge vortex, thus an upward component
of velocity is induced by the tunnel side wall), lifting the
vortex core higher above the wing, and incrementing the local
static pressure due to blockage effects. The pressure
distribution on the tunnel walls of three different wind tunnels
(with the same wing, incidence, and flow conditions) can be
seen in Fig. 6. The square tunnel has a S/W = S/H = 0.42,
the 3x2 tunnel has a S/W=0.42 and S/H = 0.63, and the 2x3
tunnel has a S/W=0.63 and S/H = 0.421. The methodology
used in this study is to create one multiblock grid in such a
way that by extracting outer blocks, tunnels of different cross
section can be studied, whilst keeping the grid around the
wing constant.

CFD simulations have shown that in all cases, vortex
breakdown is closer to the apex of the wing when the wing is
confined by tunnel walls. By comparing solutions from
different tunnel cross sections, it has been concluded that roof

\footnote{The model span (S) to tunnel height (H) or width (W) ratios
indicate the 3x2 tunnel and square tunnel have the same side
wall location, but the roof and floor are nearer the wing in the
3x2 tunnel. For the 2x3 tunnel, the side wall is closer than in
the square tunnel, but the roof and floor distances from the
wing are equal.}
and floor proximity have little influence on vortex breakdown location, with the side wall location being the most dominant factor. Thus for a given tunnel cross section the model should be orientated in such a way that the side walls are as far away from the wing as possible. Tunnel blockage was also found to have little effect on vortex breakdown.

For sinusoidal pitching motion, CFD simulations have highlighted considerable differences in breakdown location when the wing is on the downstroke of its pitching cycle compared to those on the upstroke (see Fig. 7). Again close proximity of the side walls causes the largest variation in results. It can be seen that for sinusoidal pitching motion, hysteresis loops form in the breakdown location, and subsequently in load and moment curves. Clearly as the leading edge vortex recovers from completely broken down at high incidence, the recovery is delayed by the presence of the side walls. On the upstroke where the wing has started from low incidence, the breakdown locations are in better agreement.

Since Euler simulations cannot model the secondary separation induced by the suction peak from the large primary vortices, RANS simulations are required to model accurately the viscous effects on leading edge vortices. An example of the complicated flow structure is shown in Fig. 8. To date, high Reynolds number RANS simulations have confirmed the conclusions drawn in the Euler study for the steady cases. This indicates how remarkably well Euler calculations can model the vortex dominated flow of delta wings.

SURGE WAVE IN AN ENGINE INTAKE

Engine surge is a complicated phenomenon that tends to occur at the compressor face of an aircraft intake but the causes of surge can be wide ranging, especially when considering the engine systems as a whole (nozzle, turbines, combustion chamber, and compression systems) as each individual component can induce surge. The production of an engine surge is usually the result of some or all of the compressor blades stalling. This stalling can be attributed to many causes dependent on operating conditions, the more common being naturally occurring transients in the flow, cowl lip separation due to hard aircraft maneuvers, and high engine face total pressure distortion.

When the compressor disc stalls this can have the effect of acting like a solid wall in this unsteady flow. The abrupt complete or partial blockage of the flow is referred to as an engine surge. This engine surge can create a strong shock wave, which can propagate up the aircraft intake. This propagating wave is sometimes referred to as a hammershock. The strength of the shock can be significant - at times as much as twice the steady pressure (even though the wave is transient), hence it is not unheard of for such waves to cause structural damage within the duct. Indeed the design of the Tornado took account of hammershock pressures for the aircraft ducts, ramps and linkage systems. In summary we can say that surge is manifested by large
scale oscillatory flow instability which can be violent, often with pulsating reversal of flow involving the entire unit. Compressor surge can also produce high acoustic situations in the form of violent bangs - a series of surges which is collectively known as cyclic surge. A single surge is known as a pop surge.

By modelling surge propagation under different conditions it is possible to obtain pressure-time histories for intake ducts. This information can be used in conjunction with structural modelling packages to determine loads inflicted on the duct structure. If necessary, re-design or attenuation measures can be taken and further iterations carried out until what is left is an intake that is structurally sound and efficiently supplies the engine face with a minimally distorted flow. Other consequences of surge can be far reaching when considering the case of twin side-by-side intakes as are very often found in military aircraft. It has been suggested that surge propagation in one intake can induce flow distortion in the adjacent intake that is sufficient to induce a further surge. This is clearly a highly undesirable situation. Other implications arise when considering intakes with splitter plates etc.

Various surge signatures have been applied at the engine face plane of a diffusing s-shaped aircraft intake and the consequent propagation of the surge wave through the duct has been modelled using PMB3D. There has been no attempt to model the production of an engine surge since the main interest for this work is solely the unsteady propagation. This type of problem has only been studied to a limited degree both experimentally and computationally. There has been a wide variety of surge signatures applied at the engine face to model surge. This work will detail the application of a signature of the type shown in Fig. 9. The main characteristics of such signatures is a rapid rise to a peak over-pressure. The time scale shown in Fig. 9 is non-dimensionalised for use with PMB3D (more information on the non-dimensionalisation can be found in [15]) but the rise time equates to around 10 milli-seconds.

Comprehensive validation of intake duct flows has been done for steady problems [15]. Validation of the unsteady surge problem is limited due to the limited data available. Focus was restricted to an examination of the 1D shock tube problem and the examination of surge in a uniform straight pipe for which very limited computational data was available. Unfortunately there was no experimental or previous computational data available for surge in an s-shaped intake. However successful validation for the straight duct and shock tube problems coupled with grid independent and time-step independent solutions for the surge case described here were encouraging.

Figs. 10 and 11 shows the symmetry plane and surface wall over-pressures respectively at four time intervals, following the application of the surge signature of Fig. 9, as the surge front propagates from the engine face towards the duct throat. Normal non-dimensional operating pressure at the engine face is 13.15 for this case and so all pressures are scaled by this value. The peak pressure applied at the engine face is therefore 26.3 and is applied at a non-dimensional time of 0.71. Figure 10(a) and 11(a) show the surge wave forming at the engine face. Normal operating conditions are still present upstream of this. Lower pressures associated with the acceleration of the flow around the cowl into the intake and acceleration around the lower side first bend are visible. This acceleration around the first bend lower side has major implications to the progression of the surge front as we shall see as separation is also induced. By the time the peak surge pressure has been applied, the surge front has already propagated to a location between the two intake bends (Figs. 10(b)-(c), 11(b)-(c). The waves are coalescing into a strong surge front. As the surge front reaches the first bend there are
peak over-pressures induced towards the upper wall that reach values of over twice the pressure applied at the engine face. These levels are only experienced on the upper regions of the wall due to the interaction of the shock front with the separated region. As the surge front continues towards the intake throat (Fig. 10(d), 11(d)) the peak pressures are a little lower but appear to have spread over a larger area. Finally the surge front exits the duct (not shown) and there is a period of choked flow and flow reversal behind the surge front until a degree of recovery is experienced.

DYNAMIC STALL

One of the major challenges in the computation of unsteady aerodynamic flows is the accurate prediction of the dynamic-stall (DS) phenomenon. This appears in high-alpha manoeuvres and is caused by the development of an energetic vortical structure known as the dynamic-stall vortex (DSV). Accurate prediction and, possibly, control of dynamic stall can enhance the performance in various engineering applications. For example, the manoeuvrability of fighter aircrafts could be enhanced if the unsteady airloads generated by DS are utilised in a controlled manner. Effective stall control of the retreating blade of a helicopter rotor could also increase the maximum flight speed by reducing rotor vibrations and power requirements. Similarly, by controlling DS the maximum speed of windmills or turbine rotors can be increased resulting, subsequently, in the production of more electrical energy and reduction of rotor vibration. The DS can be studied by considering the pitching motion of an aerofoil beyond its static-stall incidence-angle. There are several phenomena associated with the pitching motion of the aerofoil, the most important being the generation of intense vorticity in the suction surface near the leading edge (see also Figure 12(a)). The formation of the DSV is accompanied by additional vortices emerging both from the leading and trailing edge. During the flow evolution, large variations of the aerodynamic loads on the aerofoil’s surface occur. Reviews of past experimental works can be found in the papers of Telionis [16] and McCroskey et al. [17] [18], while a comprehensive experimental study was more recently published by Piziali [21] From the computational point of view, the study of high Reynolds number flows depends strongly on the turbulence model used. Moreover, computations of flows with moving boundaries are demanding in terms of computer resources and, therefore, numerical issues in connection with the time integration of the Navier-Stokes equations also require careful consideration. Past research in relation to steady and unsteady turbulent flow simulations in the context of URANS, has shown that the accuracy of numerical predictions is significantly affected by the accuracy of the turbulence model employed. At present, non-linear eddy viscosity models appear to offer realistic representations of the turbulent flow field taking into account Reynolds stress anisotropy at a cost significantly less than second moment closures. In this work non-linear models of the $k$-$\varepsilon$ family (Craft et al. [19]) have been employed. There are several phenomena associated with the pitching motion of an aerofoil, the most important being the generation of intense vorticity over the suction surface near the leading edge. As the pitching motion goes on, the DSV is formed and, subsequently, detaches from the body and convects along the suction surface. The above is accompanied by large variations of the lift, drag and pitching moment coefficients. The DS evolves either with the generation of weaker vortices, if the aerofoil remains above its static angle of incidence (ramping case), or terminates if the aerofoil returns to an angle sufficiently small to allow re-attachment of the flow (oscillating case).

Fig. 12. Onset of the DSV in the leading edge region of an aerofoil at subsonic (a) and transonic (b) conditions.

The amplitude of oscillations and the mean incidence during the motion of the profile play a dominant role in the development of the dynamic stall. Fig. 14a presents the lift loop obtained using a variety of turbulence models from simple one-equation closures to a full Reynolds stress model. Since for this case the amplitude of the oscillation is small and the mean incidence well below the static stall angle a simple hysteresis of the loads is obtained. The flow remains attached in both experimental and CFD results. Moving to higher oscillation amplitudes, flow separation becomes dominant and large variations of the aerodynamic loads are expected. This is shown in Fig. 14b where CFD results are compared against Piziali’s measurements. The sharp loss of lift as the aerofoil reached the peak incidence is well predicted by the CFD, however, only the non-linear eddy-viscosity models are able to get good quantitative agreement with the experiments. Most of the differences are
CONCLUSIONS

Computational Fluid Dynamics (CFD) enhanced by advanced computing resources is reaching a maturity such that it can be exploited to improve understanding of complex flow processes. In the past explanations of some complex flows has been based on sparse information usually from experiments combined with analytical approaches and intuitions. A carefully applied CFD solution will provide accurate high resolution spatial and temporal information of all physical parameters within the calculated flow field. The approach used then in the various examples demonstrated in this paper is to validate first the code against the original experimental data and then to use selected visualizations of this high resolution data to help further the flow understanding. The cases include transonic flow over a cavity, the pulsating flow over a spiked blunt body, vortical flow over a manoeuvring delta wing, surge wave in an engine intake duct and dynamic stall over an aerofoil. Demonstrated the use of the in-house unsteady viscous flow code, PMB, and a powerful cluster-computing environment made this approach work successfully.

REFERENCES