22 January 2019

Computational Fluid Dynamics (CFD) and Aircraft Design

Professor Gary Page
Aeronautical & Automotive Engineering Dept
Loughborough University

Having a local speaker made his introduction easier than many, and was an opportunity to reflect on his roles at the university from 1992 to the current day. His involvement in teaching and in aircraft and aero-engine related research with colleagues has been a considerable and significant contribution over a wide range of practical and theoretical programmes. Throughout that period analytical techniques based on well-established concepts have changed largely through extension of their application by using modern digital technology under the title 'computational fluid dynamics' (CFD). The speaker has been central to many projects that have applied these new techniques, and he stressed that his introduction to this still relatively new 'science' would be a sojourn rather than a speech, and that was to be a promise upheld - a notable absence of equations, a wide range of examples, and plenty for the eye and mind to consider.

His introduction covered principles of fluid mechanics using schematic diagrams that reminded everyone that the properties of fluids within a defined volume can change. E.g.: if the temperature changes, so too can the density of the fluid and perhaps too the pressure: to mention just three properties. He showed the side elevation of a simple fluid flow, divided into what could be viewed as 'tiles,' and stressed that these represented the side-view of a three-dimensional matrix of cubes. These are the building blocks used in computational fluid dynamics (CFD) analysis. Conservation of energy principles require the assumption that, although the properties of individual cubes can change, energy cannot be lost or gained – and this constrains the way a fluid reacts: all re-balancing according to the physics of the fluid. The majority of work addressed in the presentation concerned air, and between the building blocks in the CFD matrix there can be 'imbalances' – these represent changes of properties – such as mass, temperature, flow velocity, etc.

His schematic of a 3x3 matrix was explained to become a 3x3x3 = 27 cubes in a typical three-dimensional CFD model, but the matrix dimension needs to be very small, to attain realism outcomes. Because of this CFD applications have increased in complexity alongside the pace of computing power improvements in recent decades.

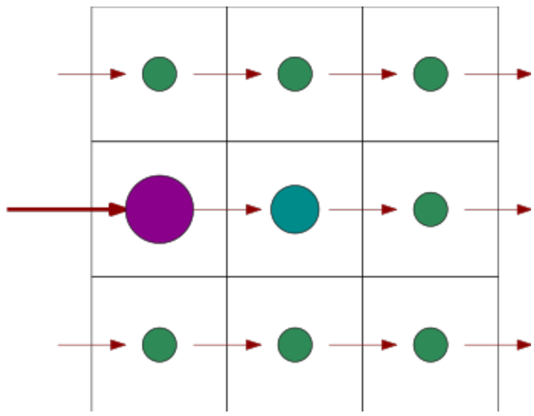

Figure 1: This diagram of a CFD matrix illustrates the principle explained in the lecture, whereby properties within each defined volume can change. e.g.: velocity, pressure or temperature. In accordance of fluid mechanic principles, the parameter properties propagate throughout adjacent matrix elements.

As analytical techniques gathered pace it has been possible to conduct more refined analysis. The presentation included example applications that addressed several aspects of design, and highlighted where CFD offered analysis of air flow situations that were not easy to study using instrumented wind-tunnel models.

These included modelling of convection and diffusion situations, e.g.: the diffusion of hot and cold streams within aero-engines and their exhaust, and the interaction with structural components, such as trailing-edge flaps. Where performance is compromised a CFD assessment provides an insight for engineers to narrow their choices, and promotes changes being examined with greater confidence. It cannot be claimed to solve the problem – that is still left to the designer.

On referring to flow turbulence the accent was widened to address all fluid mechanic applications where understanding the management of flow phenomenon is vital to efficiency. Reference was made to

- transport (aircraft, ships, cars and trains)
- energy (nuclear powerplants, wind turbines and energy cells)
- medical (flow in the human cardiac system), and
- environmental (large-scale meteorological 'systems' – hurricanes, etc.) plus in the latter category the forecasting of flood risks and the influence of prospective climate change parameters.

One example of an aeronautical application was a CFD-created plot of pressure distribution – illustrated with colour variants that related to air pressure on the upper surface of a high sub-sonic aircraft wing during high-altitude cruise (shown at Figure2).

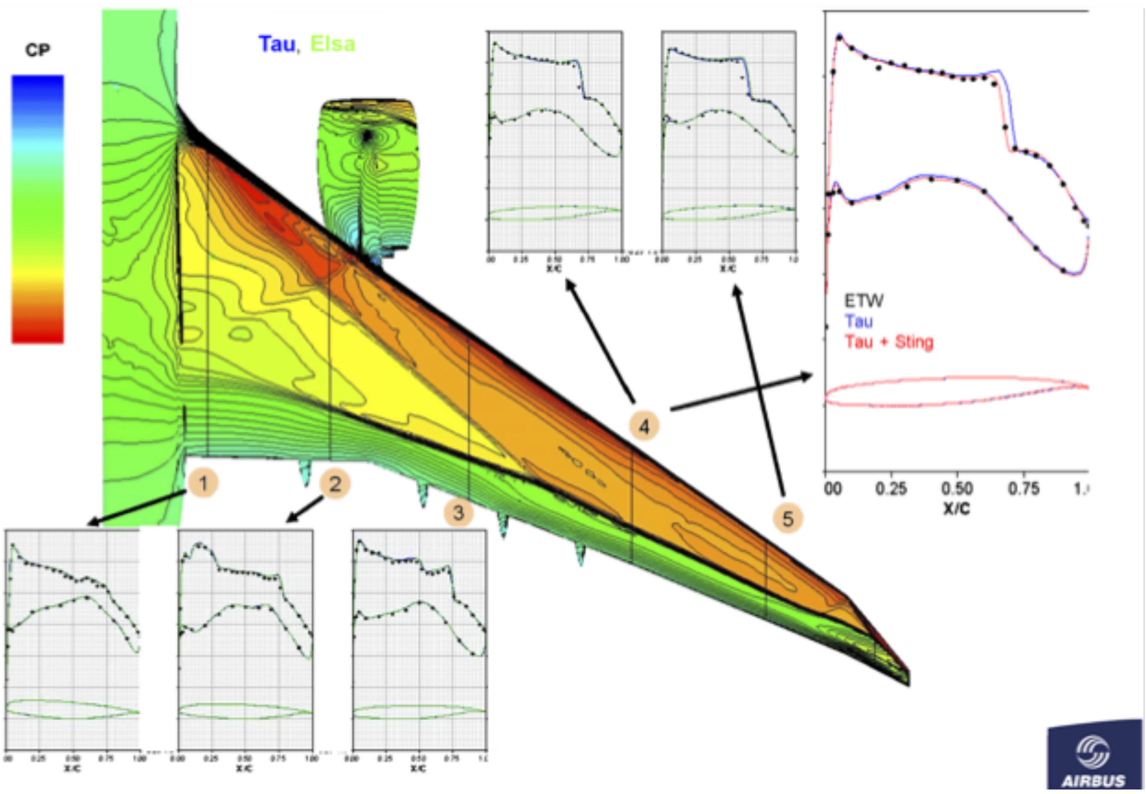

Figure 2: An industrial view on numerical simulation for aircraft aerodynamic design (source: Adel Abbas-Bayoumi, Klaus Becker Journal of Mathematics in Industry, (2011))

This illustrated:

- the influence of underwing-slung engine pylon interference (on the wing upper surface and near to the leading edge)
- and overall flow effects. An example was a line along which the air properties change suddenly at a shock wave along the majority of the wing span (it appears as a thick black line on figure 2 (next page), some distance inboard of the wing trailing edge).

There was no mention of the actual CFD grid, but this was undoubtedly a high-resolution model of a flight-case – set to representative wing-loading, altitude and speed conditions. It would have required a degree of computing power that has only become available in the modern day. The relevance of the data from such analyses was stressed to have many applications too, relevant to overall flight dynamics and flight control parameters.

The final example shown was a very specific application. This was a full 3-D model of the airflow around a Harrier, in hover close to the ground and with all four nozzles ejecting the exhausts vertically downwards. This was modelling of a Harrier with a hot 'plenum chamber' configuration with hot front exhaust: it was relevant to supersonic VTOL research that did not result in a flown design.

A slow-time animation, created by showing numerous closely-spaced cross-sections at frequent stations along the fuselage length, provided a vivid example of how air from each nozzle expanded outwards close to ground level, and that the portion nearest the aircraft centreline was trapped between the two exhaust plumes and formed a 'reverse plume' that impinged the underside of the aircraft . Belly strakes (already fitted on the Harrier – but apparently enlarged in the model) trapped the upward flowing air by forcing it to flow rearward. A retractable 'dam,' also already in use, prevented exhaust gases flowing forward, where it could have entered the engine intakes.

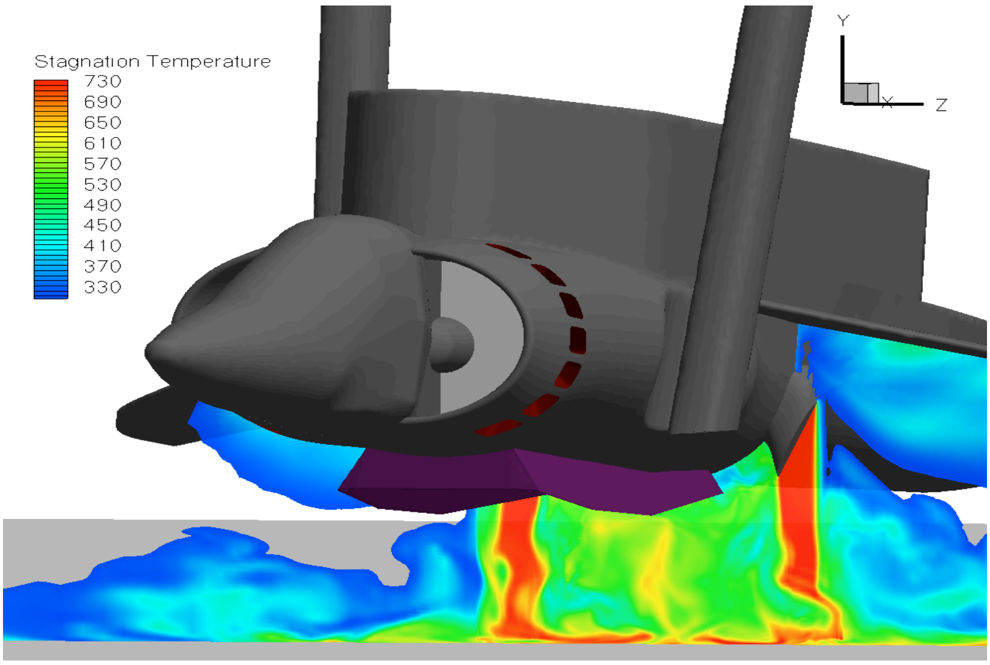

Figure 3: This is an extract from a CFD animation of a Harrier in a hover close to the ground. It is a snapshot that shows a single cross-section of air temperature across the CFD matrix field. Cross-sections ahead and behind this snapshot were shown in the animation, and illustrated activity fore and aft in the volume of air from almost nose to tail of the aircraft was visible. (The purple areas are strakes modelled on the aircraft belly).

This was a fine example of what mammoth-scale CFD modelling is beginning to achieve. We saw the flow-visualisation as if it was an entity, but it was really a series of sections that had resulted in a three-dimensional CFD matrix of enormous proportions. For those who had to use empirical rules and combine that with the knowledge in their heads only a decade or so ago, to see this kind of visualisation is a fine example of the rate of progress this analytical capability delivers. It does not replace the designer's ability to think-through design solutions, but it is able to provide information sooner than building and testing models alone, and as such will facilitate a reduced timescale, and expression of extra confidence, in the design outcomes.

An underlying theme in the whole presentation was that the problems faced by modern-day aircraft designers are nowadays more often examinable at early configuration stages. This could give more freedom to designers. They may be able to examine more ambitious aircraft configurations, also the information provided assists in bolstering confidence and reducing risk, attributes that have the potential also to reduce the time needed to conduct flight trials.

It was a presentation that updated many of those present on what progress is being made. Some student project presentations to the Branch in recent years have alluded to elementary CFD analysis work already reaching student level, and it is clear that computation techniques overall will keep the evolution of such applications improving in the future.

The session attendance was about 140 people. They were well pleased to hear such an information-packed and well-paced presentation and expressed delight at answers to their questions and a strong appreciation through their applause.

Presentation notes by Mike Hirst