Progress toward CFD for full flight envelope

E. N. Tinoco, D. R. Bogue, T-J. Kao, N. J. Yu, P. Li and D. N. Ball
Enabling Technology and Research
Boeing Commercial Airplanes
Seattle, Washington, USA

ABSTRACT

The value of computational fluid dynamics, CFD, delivered to date has mainly been related to its application to high-speed cruise design. To increase its applicability CFD must apply to the full flight envelope frequently characterised by large regions of separated flows. These flows are encountered by transport aircraft at low speed with deployed high lift devices, at their structural design loads conditions, or subjected to in-flight upsets that expose them to speed and/or angle-of-attack conditions outside the envelope of normal flight conditions to name a few. Such flows can only be characterised by the Navier-Stokes equations. This paper will report the progress toward CFD for full flight envelope. The CFD methods in use at Boeing will be described. Examples presented will address high-lift, loads and stability and control concerns including Reynolds scaling from wind tunnel to flight, vortex generator simulation, spoiler and horizontal tail effectiveness. In general, results shown are in 'good enough' agreement with experimental data. Deficiencies and the need for further algorithm and process improvement are noted. The need for automation to enable the large scale use of CFD will also be discussed.

NOMENCLATURE

AGPS	aero grid and panelling system
Ames	NASA Ames 11ft Transonic Pressure Tunnel
b	wing span
BCA	Boeing Commercial Airplanes
BTWT	Boeing Transonic Wind Tunnel
C_l	sectional lift coefficient
C_L	total lift coefficient
CFD	computational fluid dynamics
CFD++	unstructured grid Navier-Stokes code
CFL3D	structured grid Navier-Stokes code
C_m	sectional pitching moment coefficient
C_M	total pitching moment coefficient
C_p	pressure coefficient
C_R	rolling moment
Eta	span fraction – $2y/b$
M	Mach number
n	load factor
NTF	National Transonic Facility
OVERFLOW	overset grid Navier-Stokes code
Re, Rc	Reynolds number
TLNS3D	structured grid Navier-Stokes code
VEAS	equivalent air speed
V_G	vortex generator
x/c	chord fraction
α	angle-of-attack
δ_{sp}	spoiler deflection angle

1.0 INTRODUCTION

The major impact of computational fluid dynamics, CFD, delivered to date at Boeing has mainly been related to its application to high speed cruise design[1-5]. Computational fluid dynamics will continue to see an ever-increasing role in the aircraft development process as long as it can provide value in the end users' eyes. It has improved the quality of aerodynamic design in the industrial environment, but has not yet had as much impact on the rest of the overall aeroplane

Paper No. 2962. Manuscript received 21 March 2005, revised version received 3 June 2005, accepted 22 June 2005.

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

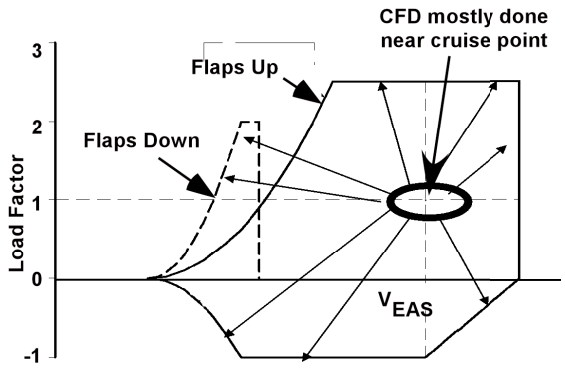


Figure 1. The flight *V-n* envelope.

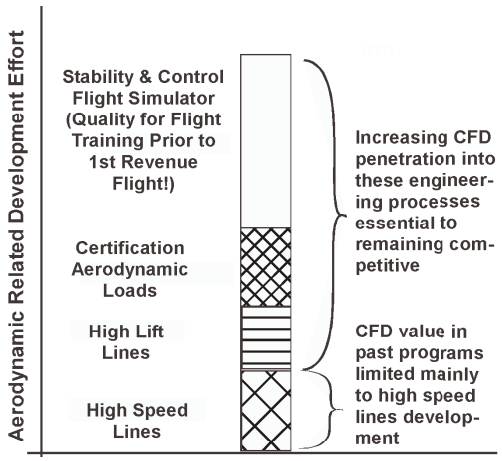


Figure 2. CFD role In aircraft development.

development process. CFD is becoming more interdisciplinary, helping provide closer ties between aerodynamics, structures, propulsion, and flight controls. This is the key to more concurrent engineering, in which various disciplines work together more in parallel rather than in the sequential manner as is typically today's practice. The savings due to reduced development flow time when is fully implemented will be enormous!

Most use of non-linear CFD has been near the high-speed 'cruise' point illustrated in the *V-n* diagram in Fig. 1. As illustrated in Fig. 2, the high-speed lines development is less than 25% of the total Aerodynamics related aeroplane development effort. To be able to expand the use CFD over the entire flight envelope considerable progress in algorithm and hardware technology is still necessary. Many flight conditions are CFD is only, at best, qualitatively representative for flows characterised by large regions of separated flows. Such conditions are encountered with manouevering combat aircraft, and with transports at low speed with deployed high lift devices, at their structural design loads conditions, or subjected to in-flight upsets that expose them to speed and/or angle-of-attack conditions outside the envelope of normal flight conditions to name a few. Such flows can only be characterised by the Navier-Stokes equations. Routine use of CFD based on the Navier-Stokes formulations to generate the large data bases needed for loads and stability and control application require orders of magnitude improvement in throughput (productivity) through improvements in geometry and grid generation, improved turbulence models, and improvement in algorithm and hardware performance.

The benefits of using CFD can be realised through qualitative flow evaluation, improving pre-flight predictions, Reynolds number scaling to full flight, resolving aerodynamic flight test issues, and reducing wind-tunnel testing. Additional impact can be realised by enabling multi-disciplinary collaboration providing closer ties between aerodynamics, structures, propulsion, and flight controls. However, improvements in CFD alone will not be enough. The process of aircraft development itself will change to take advantage of the new CFD capabilities. When fully implemented, the resulting savings due to reduced development flow time and reduced risk in certifying a new aeroplane will be enormous! This paper will report the progress toward CFD for full flight envelope at BCA. The paper will focus on the application of steady Navier-Stokes solutions to issues of interest to the loads and stability and control engineering disciplines.

2.0 CFD TOOLS AND METHODS

The purpose of CFD in industry is to produce a better product. To this end, CFD is used when the design engineers believe that its use will lead to that better product within the allotted budget of time, money, and effort. Producing this better product is a combination of improved performance, reduced design cycle time, reduced risk in meeting product objectives and cost targets. Product development engineers must be able to focus on engineering processes and have little time for manipulating CFD 'processes', i.e. codes must be very user oriented. To achieve this goal stable 'productionised' packaged software solutions have been developed to enable and promote consistent processes. These not only put CFD into the hands of the product development/product support engineers, but also allows the 'expert' user to get results more quickly with reduced variation. These integrated packaged software solutions combine the various CFD components to go from 'lofts to plots' in the time scale consistent with a fast paced engineering program.

These packages include: scripted packages for 'standard' configurations; geometry and grid/paneling generation components; flow solvers; and post-processing components for analysing the results. These are all placed under some form of software version control to maintain consistency. Computing resources must also be integrated to include high-end computing and graphics workstations. Mass data storage must be seamlessly available to store the vast amount of information that will be generated during the engineering application. These systems require dedicated computing system administration. The software control and computing system administration are necessary to free the engineers to focus their work on the engineering processes and not be consumed by the 'computing' process.

A key component of CFD and most engineering processes is geometry. CAD systems, such as CATIA, dominate most geometry engineering needs. However, these systems are designed for component design and definition and are not necessarily well suited for CFD use. A key component of many BCA CFD processes is AGPS – aero grid and panelling system[6]. AGPS is a geometry software tool implemented as a programming language with an interactive graphical user interface. It allows the engineer to create, manipulate, interrogate, or visualise geometry of any type. It features a journaling/replay capability to automate tasks such as surface grid generation and the creation of the complete input set-up to the CFD process.

Two CFD methods, in use in Boeing Commercial Airplanes (BCA), that feature significant automation for fast turn-around are ZEUS for structured grid Navier-Stokes analysis and, TRANAIR[7] for full potential plus boundary-layer analysis and design/optimisation, and an unstructured grid system under development using CFD++[8] for 2D and 3D high-lift Navier-Stokes analysis. A description of the ZEUS system follows.A limited number of solutions were also obtained with TRANAIR for comparison.

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

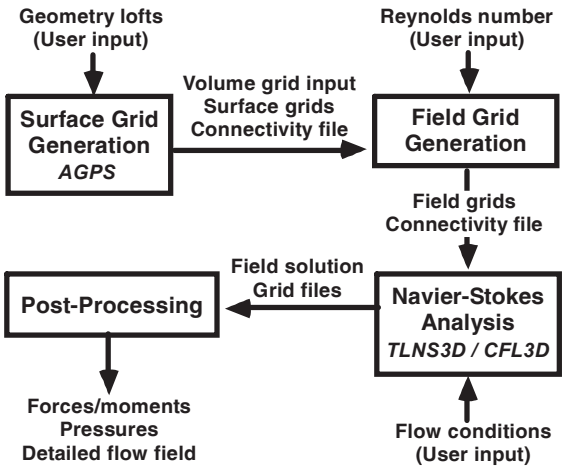


Figure 3. ZEUS Navier-Stokes analysis system.

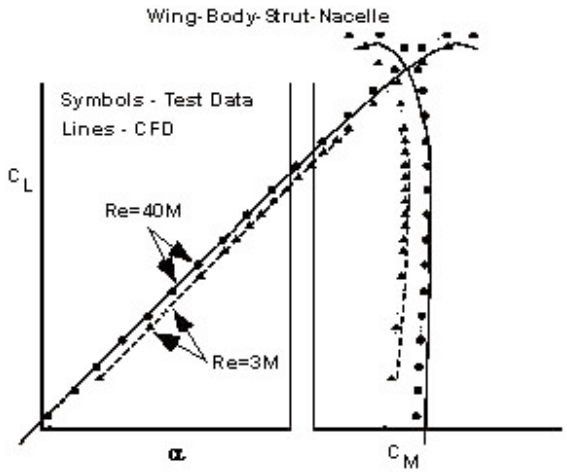


Figure 4. Reynolds number effect on longitudinal characteristics.

2.1 ZEUS

The ZEUS analysis system, illustrated in Fig. 3, is a series of UNIX shell scripts that coordinate surface grid generation, field grid generation, analysis, and post-processing for structured grid Navier-Stokes CFD codes such as CFL3D[9] and TLNS3D[10]. This system was developed with the goal of rapidly deploying CFD advancements to the user community. For a given type of configuration, a specific grid generation process is developed and packaged into the system. Boeing CFD developers have rewritten a 3D grid generation code through the use of an advancing front approach[11], so that a precise control on grid quality, such as grid spacing, stretching ratio, and grid orthogonality near configuration surfaces can be achieved. This is an important requirement for accurate resolution of viscous flow regions for all existing Navier-Stokes solvers. Two structured grid generation approaches are currently in use (i.e. the matched/patched multiblock grid approach and the overset or overlap grid approach). The former approach subdivides the flowfield into a number of topologically simple regions, such that in each region high quality grid can be generated. This is a rather time-consuming and tedious process for complex configuration analysis. However, once this 'blocking' process is done for one configuration, a similar configuration can be done easily through the use of script or command files.

To analyse a configuration using the ZEUS analysis system, users are only required to input geometry lofts and flow conditions, and the system takes care of the rest of the analysis processes. The output from the system includes automatic plots of sectional data, convergence history, configuration force and moment information, together with detailed flow field files for flow visualisation and diagnosis. Current configurations supported by ZEUS include isolated nacelles, wing/body, wing/body/pylon/nacelle with either twin or four-engine flow-through or powered nacelles, and wing/body/pylon/nacelle/horizontal-vertical tail. Options to support winglets and raked tips are also available. Configurations not included in the 'Standard', such as the addition of vortex generators, control surfaces, or spoilers, can still be analysed but will not benefit from the same degree of automation.

3.0 CFD APPLICATIONS

Unless otherwise stated the CFD results presented in this paper were obtained with either CFL3D or TLNS3D Navier-Stokes codes. Our experience has been that CFL3D is the more accurate code, while TLNS3D is more tolerant of grid quality. NASA experience with the two codes is reported in Ref. 12. Current strategy for structured Navier-Stokes codes is focusing on CFL3D. The turbulence models used were either the SA Spalart-Allmaras[13] or Menter's *k-w* SST[14] model. The SST model is the preferred model since it produces good results for highly separated flows, is relatively robust, and does not use an extraordinary amount of computer resources.

Wind-tunnel test data presented were obtained either at the BTWT, Boeing Transonic Wind Tunnel, or at the NTF, National Transonic Facility located at the NASA Langley Research Center. BTWT is an atmospheric wind tunnel with an 8×12ft test section. The NTF is a pressurised cryogenic facility capable of full scale Reynolds numbers. It features an $8{\cdot}2 \times 8{\cdot}2$ft test section.

3.1 Loads

Aerodynamic loads prediction is one of the more time consuming items in an aircraft development program. Aerodynamic loads are needed before the detail structural design of the airframe can proceed. Critical design loads are generally found at the extremes of the flight envelope. These loads can be subject to significant Reynolds number effects that the typical conventional wind tunnel may not catch. Cryogenic wind tunnels, which might have flight Reynolds number capability, are not necessarily a very practical option due to model strength limitations, the number of test conditions needed, cost, and flow time required to design, build, and test the wind tunnel model. Progress is being made in the ability to use CFD to provide Reynolds number corrections to conventional wind tunnel data, and to use CFD to determine preliminary loads prior to a wind tunnel entry.

Figure 4 shows a comparison of aeroplane lift and pitching moment characteristics at wind tunnel and flight Reynolds number. The CFD predictions capture the incremental differences due to Reynolds number and show reasonable agreement with the test data. Here the CFD results were shifted by a constant +0·3 degrees in angle-of-attack and +0·01 in pitching moment to better illustrated the incremental predictions with Reynolds number. Of greater interest to the Loads engineer is the distribution of these forces and moments across the wing. Figure 5 shows wing section lift and pitching moment coefficients as a function of angle-of-attack at several stations along the wing span of a twin-engine transport. CFD results are in good agreement with capture the incremental differences due to Reynolds number change.

Prior to the availability of wind tunnel data, CFD can provide wing load characteristics over a wide range of Mach numbers.

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

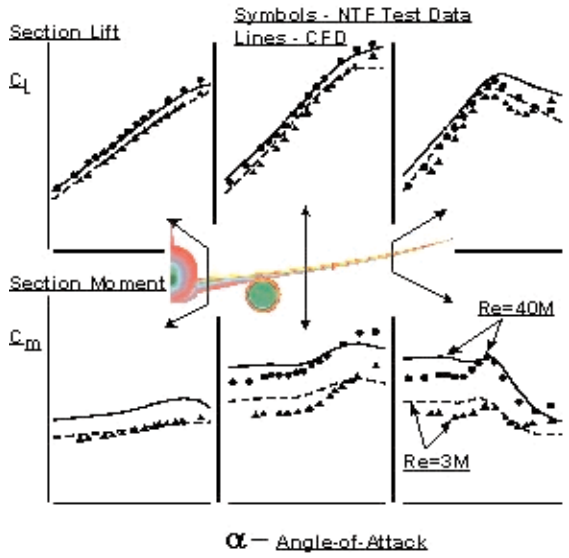


Figure 5. Section characteristics for a twin-engine transport.

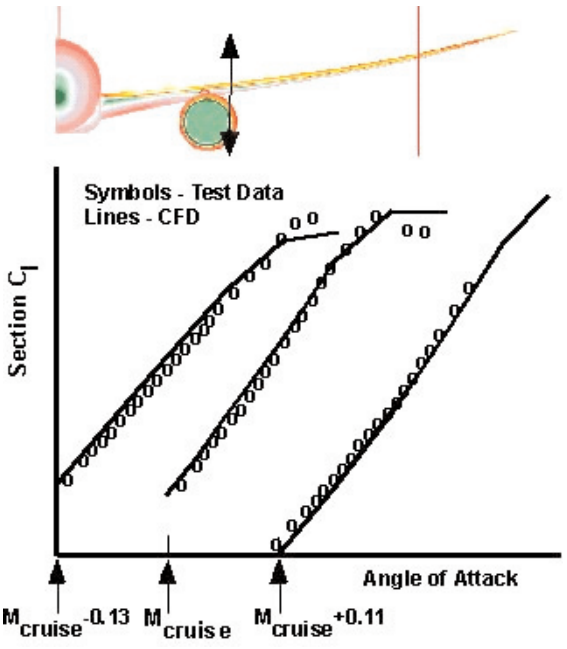


Figure 6. Section lift at several Mach numbers.

Figure 6 shows a comparison of CFD results with test data at a wing spanstation outboard of the engine nacelle. Section lift coefficients as a function of angle-of-attack are shown for three Mach numbers – cruise Mach, cruise Mach minus 0·13 Mach, and cruise Mach plus 0·11 Mach. The CFD results well capture the loading variation with Mach number. These data can be used in determining preliminary structural sizing and to support wing trade studies.

In the development of a large commercial transport the determination of 'final' aerodynamic flight loads demands a very high degree of accuracy. Too much conservatism in the loads prediction results in unacceptable excess structural weight. Under-predicting the flight loads may result in an uncertifiable aeroplane or certifying to a lower weight with significant performance penalties until the structure is improved. CFD is not yet able to deliver the required degree of accuracy and throughput necessary to replace the wind tunnel. The challenge today is how to best combine the use of CFD with wind tunnel testing to improve the prediction of aerodynamic flight loads while reducing the development cycle time.

3.2 Stability and control

Stability and control considerations can eliminate an aerodynamic design that otherwise meets its mission requirements if they compromise safety, lead to unacceptable handling characteristics, or are not certifiable. Traditionally, the only tools a stability and controls engineer has had for determining the pitch characteristics prior to wind-tunnel testing and for scaling the resulting data to flight conditions have been experience and common sense. Significant amounts of S&C wind-tunnel testing are necessary to obtain information on various combinations of control surfaces across the Mach, angle-of-attack, and sideslip ranges to simulate normal operations as well as failure conditions. The large data set obtained in wind-tunnel testing is then synthesized into a flight simulation database. This database must be shown to be in compliance with certification requirements if it is to be used as a 'level D' pilot training device.

As in the case of loads data, stability and control data in the wind tunnel is usually acquired after the aerodynamic design is finished, which adds to the development time. Stability and control issues of interest are far ranging. Analyses frequently require a high-resolution solution of the entire aircraft. Several solutions may then be required to compute a single characteristic such as Reynolds number scaling for a single combination of Mach number, angle-of-attack, and control deflection. Each Reynolds number will require a different volume grid to ensure the proper spacing in the boundary layer. Any errors in pre-flight values make post-flight modifications more difficult and time consuming. Potential cost benefits of an improved pre-flight database are expected to be at least one order of magnitude larger than potential reduction of wind tunnel testing.

3.3 Pitch characteristics

Wing vortex generators are used to improve the pitch characteristics of an aeroplane to achieve acceptable and certifiable handling characteristics without total reliance on electronic stability augmentation or flight envelope protection. Accurate CFD vortex generator analysis capability is particularly intriguing since it enables a dramatic change in the development process. Historically, vortex generator patterns are developed through expensive and time consuming wind tunnel and/or flight testing. A CFD based vortex generator analysis method enables a dramatic reduction in the development resources required by providing:

1. The ability to predict the effect of a VG pattern prior to wind tunnel or flight testing.
2. Increased insight into determining VG placement on a wing.
3. Evaluation of vortex generator Reynolds number effects in a controlled environment.

This technology would also allow improved collaboration between aerodynamics and loads organisations. Both Reynolds number trends and vortex generator increments could be utilised on sectional and integrated levels, furthering efforts towards a common database for loads and aerodynamics.

Computational simulations have been carried out to assess the ability of CFD to predict the effects of vortex generators on a twin engine transport configuration. Early studies by Yu[15] yielded encouraging results that have led to further refinements in the CFD process. All major relevant elements of the twin-engine aircraft

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

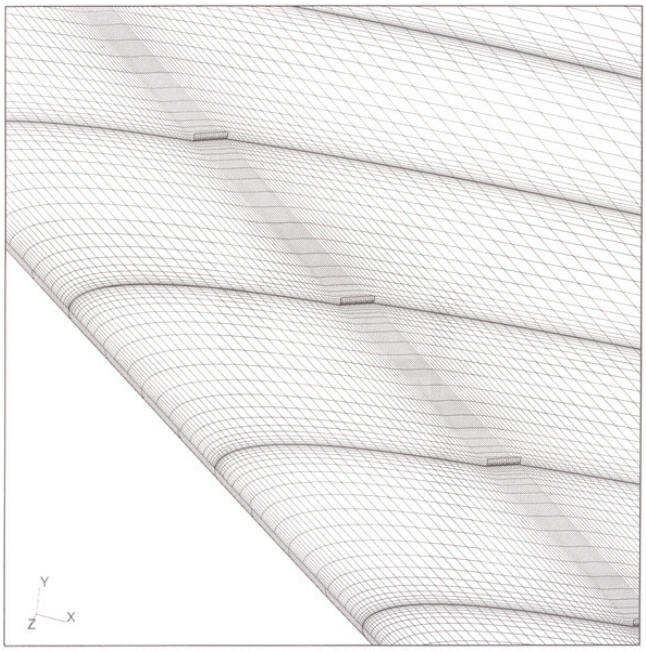


Figure 7. Surface grid near vortex generators.

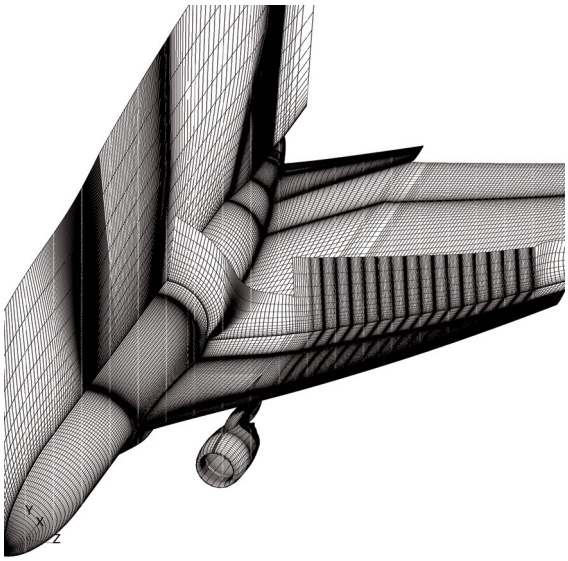


Figure 8. Multi-block grid for twin with vortex generators.

geometry were included in the study. The configuration included the wing/body, strut/nacelle, wing vortex generators, horizontal and vertical tails. Neither the nacelle chine nor the flap track fairings were modelled, as neither is believed to significantly affect high-speed pitch characteristics. The 1-*g* twist was used throughout this study, and no attempts were made to modify the twist as a function of angle-of-attack to account for wing aeroelastics.

In this application, vortex generators were modelled as small vertical plates mounted on the wing upper surface at specified locations and orientations. The size and height of the vortex generators are scaled with Reynolds number for wind tunnel or flight simulation.

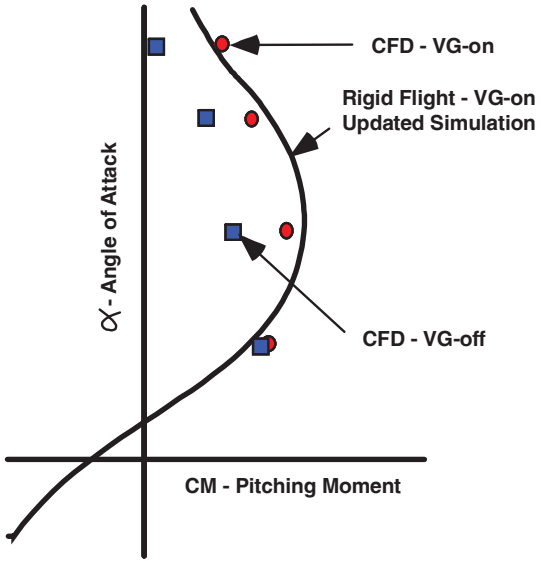


Figure 9. Effect of vortex generators on pitching moment.

They are treated as zero thickness, inviscid plates with the height of the order of local boundary layer thickness. To capture the effects of vortex generators computationally, fairly dense grids in streamwise and in spanwise directions are required as shown in Fig. 7. An illustration of the multi-block structured grids generated for this model is presented in Fig. 8.

The effects of vortex generators on aeroplane pitch characteristics are shown in Fig. 9. The results compare reasonably well with flight data with respect to predicting aeroplane pitch characteristics, even at relatively high angles-of-attack where the flow is massively separated. The 'rigid flight updated simulation' line represents the 'rigidised' flight test extracted pitching moment.

3.4 Lateral control characteristics

Lateral control characteristics are at least as important as longitudinal characteristics. The primary metrics used to evaluate lateral controls are the magnitude and linearity of the rolling moment. The impact of failing to predict the rolling moment accurately includes certification risks including basic roll rate requirements and aeroplane-pilot-coupling. These data are critical for an aeroplane with wing load alleviation. Experimental results on a generic transport indicate wing lateral controls exhibit significant Reynolds number effects[16]. Lateral controls typically consist of both ailerons and spoilers. Results from an initial study of applying Navier-Stokes CFD to look at Reynolds number effects on aileron controls were presented in Ref. 17.

Studies on the use of CFD for spoiler simulations are also ongoing. Spoiler characteristics were examined on a twin-engine transport whose CFD solution is illustrated in Fig. 10. The computational model included the full configuration with installed engines, horizontal and vertical tails, wing vortex generators, and deflected inboard and outboard spoilers on one side of the aircraft. Total pressure contours on two body station cuts aft of the wing are also shown in the figure. These clearly show the asymmetry of the resulting flow solution. A portion of the surface grid used to generate the solutions is shown in Fig. 11. The deflected spoilers are modelled as thin surfaces. The high grid density associated with each vortex generator is clearly visible. Over 65m volume grid cells were used in the solution.

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

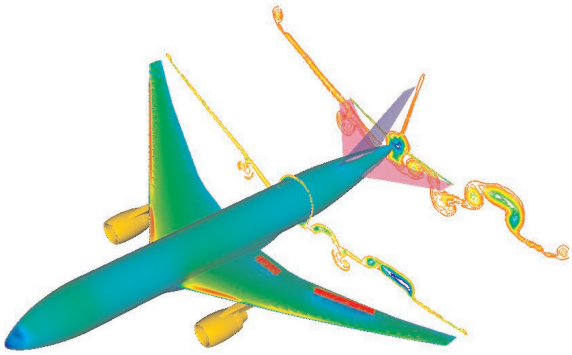


Figure 10. Navier-Stokes solution about a twin-engine configuration with deflected spoilers.

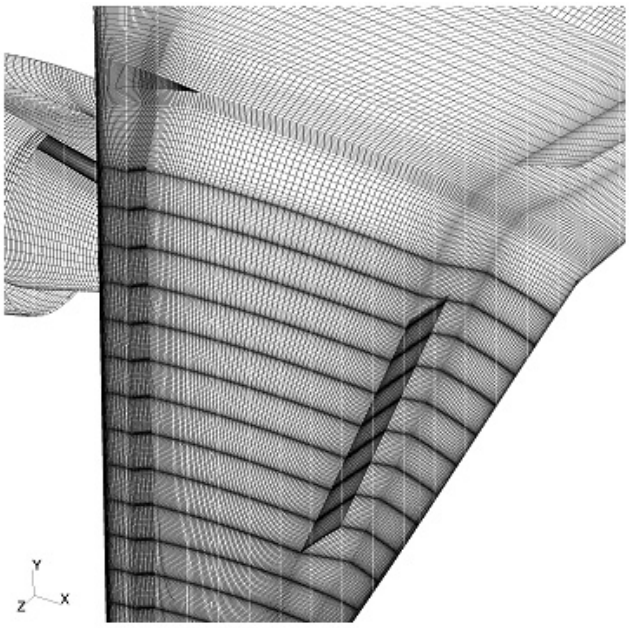


Figure 11. Surface grid with deflected spoiler.

Flow characteristics for a deflected spoiler case are shown in Fig. 12. Vortex generators and wing surface streamlines are illustrated. A total pressure slice through the flow field at mid chord shows vortices induced by the vortex generators, indicating that they are working well and properly modelled.

Pressure distributions are compared to test data at inboard and outboard locations for one of the solutions as shown in Fig. 13. Comparisons reveal the solutions are relatively accurate with good correlation on the wing surface aft of the spoiler. The outboard location reveals a spike in the CFD data caused by the presence of a vortex generator. The distribution and density of pressure ports in the experimental data was inadequate to resolve this detail.

The effect of Reynolds number on rolling moment due to spoiler deflection for this configuration is shown in Fig. 14. The results are for two Mach numbers and Reynolds numbers attainable on a full model in BTWT, the Ames 11ft Transonic Pressure Tunnel, and flight. Reynolds number effects are shown to increase at higher Mach number and increase roughly linearly with the log of Reynolds number.

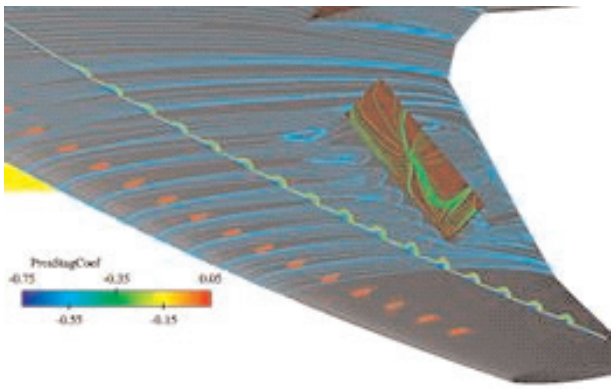


Figure 12. Flow detail around spoiler.

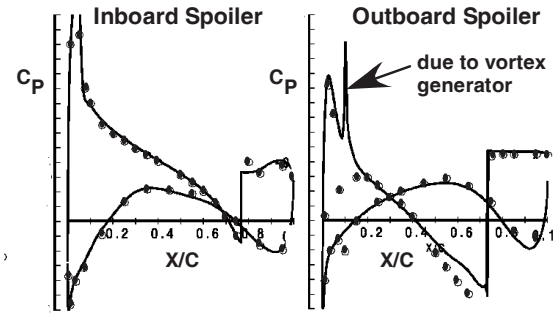


Figure 13. Pressure distribution with deflected spoiler.

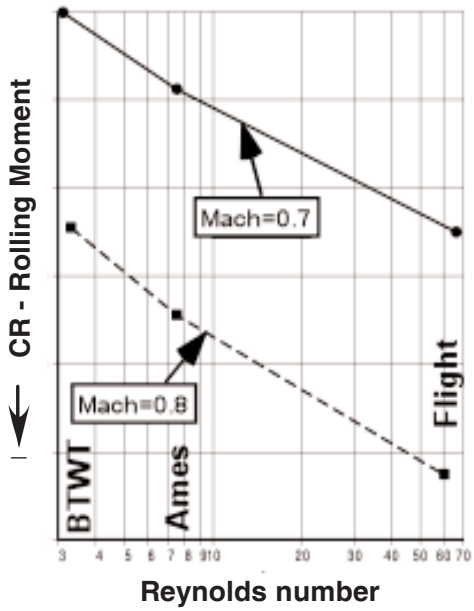


Figure 14. Reynolds number effect on rolling moment due to spoiler deflection.

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

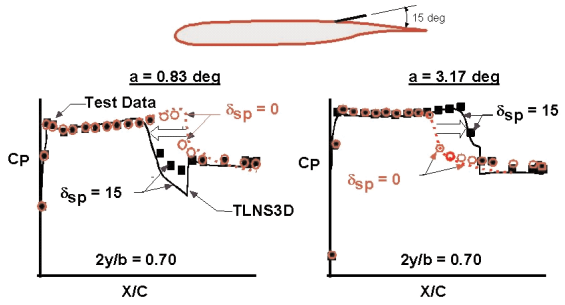


Figure 15. Effect of spoiler deflection on wing upper surface pressure distribution.

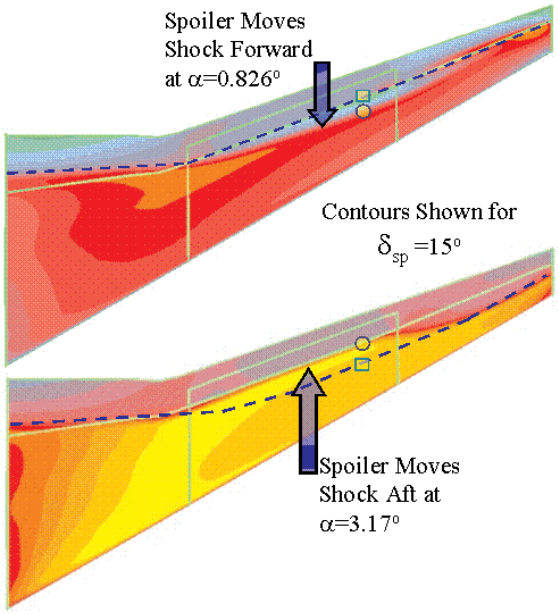


Figure 16. Spoiler effect on shock location.

3.5 Spoiler reversal

Finding surprises in handling characteristics demands the development of large wind tunnel based aerodynamic databases of thousands of variations of flight conditions and control settings. Even these are not always successful in discovering adverse effects prior to flight test. Current practice requires the anomaly to be discovered experimentally before CFD can be used to try to understand it. Discovery of such an anomaly on a research wing led to the initial CFD work in spoiler effects[18]. Wind tunnel data indicated that at a high Mach number beyond cruise Mach the effect of deflecting a wing spoiler on rolling moment would reverse as angle-of-attack was increased. A study ensued to see if CFD could simulate and explain this reversal. Figure 15 shows a pressure distribution comparison of experimental and CFD results. The data were taken at a high Mach number at the 70% span station on a transport wing with and without a deflected spoiler. The low angle-of-attack case shows the expected effect of decreasing lift due to the spoiler deflection. The high angle-of-attack case shows just the opposite, lift increasing with spoiler defection. The spoiler effectiveness has reversed! The culprit in this reversal is the action of the shock wave on the wing when the spoiler was deflected. At the lower angle-of-attack (and lower Mach numbers) deflecting the spoiler moves the shock forward decreasing the wing lift. At the higher angle-of-attack deflecting the spoiler actually caused the shock to move aft increasing the lift! This is further illustrated in Fig. 16.

CFD was able to match the experimental results in this case because the conditions for these phenomena had been previously identified. It is highly unlikely that these phenomena would have ever been found running a limited set of CFD solutions. Like the case for aerodynamic loads, the number of flight conditions that must be evaluated for determining preliminary stability and control, and handling characteristics demand at least two orders-of-magnitude reduction in CFD cost and flow time before it can be used effectively. The impact of CFD on stability and control prediction will be as an enabler to reducing the number of design/test cycles, and risk in an aeroplane development programme.

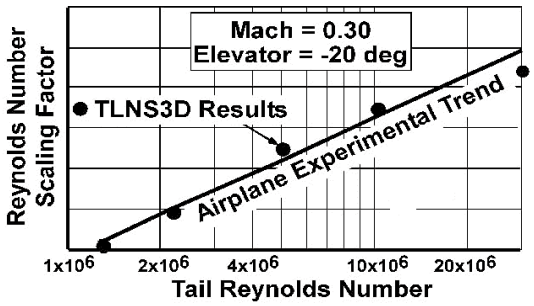


Figure 17. Elevator control effectiveness.

3.6 Longitudinal control

Determination of flight level effectiveness of the empennage for longitudinal control affects the optimal tail size, the flight characteristics and the resulting structural loads. Accurate prediction of the horizontal tail and elevator characteristics will result in improved performance, reduced risk, and are critical to ensuring compliance with certification requirements on a jet transport. Failure to quantify the empennage control characteristics results in limitations on the allowable center-of-gravity range, flight condition or general certification. The need for accurate control power characteristics is magnified by increased collaboration with other groups and adoption of stability augmentation systems. Previous analysis by Reichenbach and McMasters has shown empennage scale effects to be significant[19].

Reynolds number scaling effects on elevator control authority at a low Mach number are presented in Fig. 17. CFD results show good correlation with experimental trends and a general increase in control authority with increasing Reynolds number out to flight conditions. Figure 18 shows the primary cause of the control effectiveness increase is decreased flow separation aft of the hingeline as indicated by the streamlines propagating from the trailing edge.

Scale effects on high-speed stabiliser and elevator effectiveness were also investigated through NTF testing and computational analysis[17]. Results are shown near cruise Mach number and at two separate Reynolds numbers. Comparisons with elastic NTF data are made on a consistent aeroelastic basis. Data are shown for the change in pitching moment between two linear stabiliser deflection angles in Fig. 19. Correlation between CFD and NTF results is relatively good, with CFL3D generally over-predicting the change in pitching moment at each Reynolds number.

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

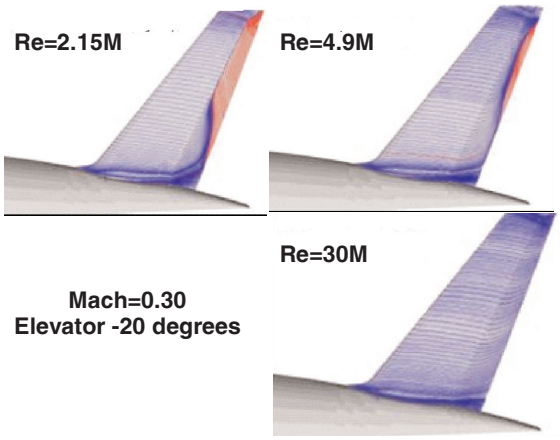


Figure 18. Reynolds number effects with deflected elevator.

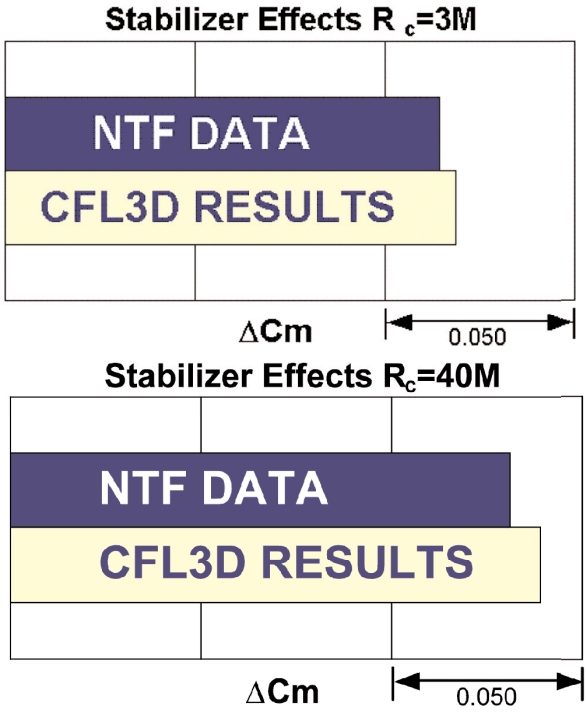


Figure 19. Effect of Reynolds number on stabiliser, near cruise Mach number.

Pressure distribution comparisons for two horizontal tail incidence angles at a transonic condition are shown in Fig. 20. These data were acquired on a different configuration tested at low Reynolds number at BTWT. The NTF model did not have tail pressures. CFD results show good agreement with the test data.

3.7 High lift

Unlike the routine use of Navier-Stokes CFD for a flaps-up configuration analysis near its cruise point and its emerging extension to full flight envelope for loads and stability and control concerns, extensive

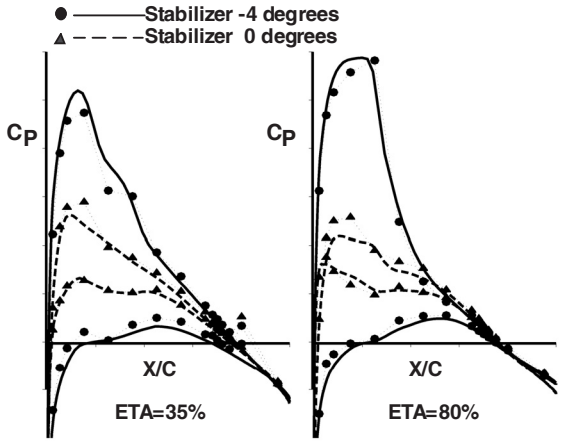


Figure 20. Horizontal tail pressure distribution.

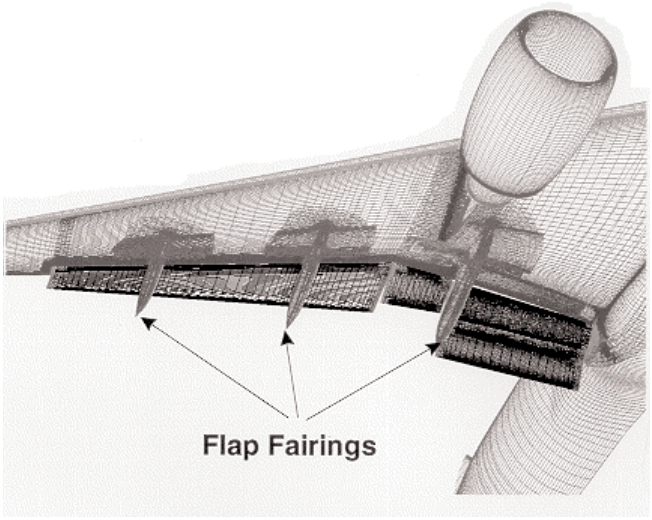


Figure 21. Overset grids.

use of Navier-Stokes CFD for low speed analysis of the high lift elements of an aircraft is far from routine. The low-speed/high-lift configuration undergoes significant 'tuning' during pre-certification flight testing – an expensive and time consuming process. On the Boeing 777 high lift elements were designed using a combination of linear inviscid three-dimensional codes and more complex viscous two-dimensional codes. Much of the actual configuration development was done in the wind tunnel. The impact of routine use Navier-Stokes CFD for the 'flaps-down' full flight envelope will be huge!

A number of very impressive Navier-Stokes solutions have been achieved to date on some very complex high-lift transport configurations demonstrating there is capability in this area. These studies have shown the importance of including all the detailed geometric features, i.e. slat and flap brackets, nacelle chines, etc. in modelling these high-lift configurations. Figures 21 and 22 show the overset grids and OVERFLOW solution of an aircraft with a complex high-lift system[20,21]. OVERFLOW excels for use on configurations with significant geometric complexity or features not captured by the automation in the block structured codes. Results agree well with experimental data for low to moderate angle-of-attacks. At high angle-of-attack, there are complex flow separations in the flap and slat gap regions, which could not be simulated adequately with the current one- or two-equation turbulence models. Improvements in

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

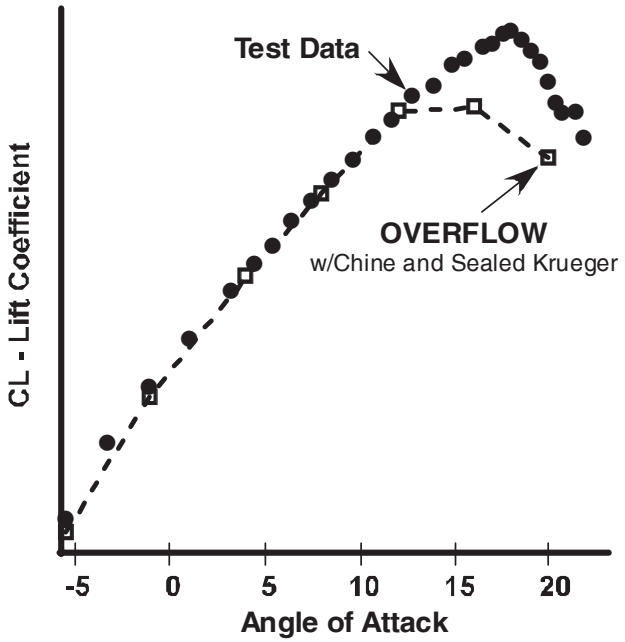


Figure 22. OVERFLOW solution on a high-lift configuration.

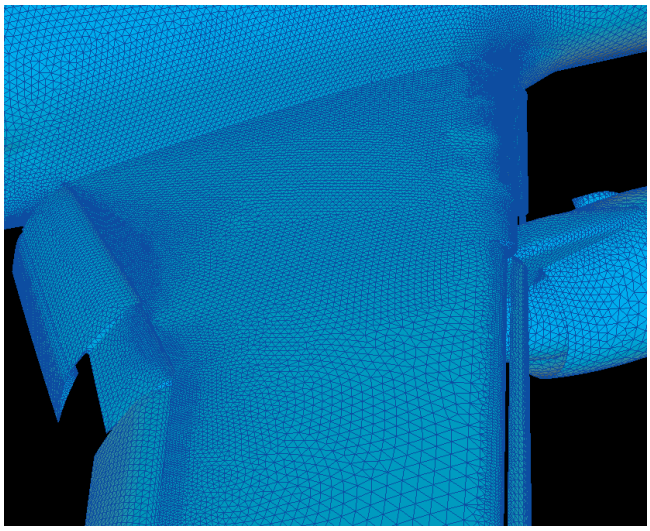


Figure 23. Unstructured grids.

turbulence models for separated flow simulation, as well as Navier-Stokes solver accuracy and robustness, are essential for a reliable prediction of aeroplane high-lift performance, as well as aeroplane pitch characteristics.

However as impressive as these solutions have been, much work is still needed in reducing the flow time to go from configuration definition to relevant plotted results by a product development engineer. The time scale to go from lofts to plots with the overset grid OVERFLOW system has been dramatically shortened in the past few years. However it still requires an 'expert' user and is still not fully compatible with a fast paced aircraft development programme.

The unstructured grid approach offers the promise of achieving the automation that will enable a product development engineer to go from 'lofts to plots' in one day. Development of this process with CFD++ as the solver is being actively pursued. A propriety geometry and grid generation process is being developed to automate use of the process for a class of configurations. An example of an unstructured grid for a twin-engine aircraft in landing condition is shown in Fig. 23. Total pressure isobars from the resulting solution, highlighting vortex and wake locations, are illustrated in Fig. 24. When this flow time reduction is achieved, extensive validation will still be needed, as well as changing the design process to take advantage of this new capability. When ready, greater use of CFD in the high lift design process will be an enabler to a significant reduction of the overall aircraft development cycle time.

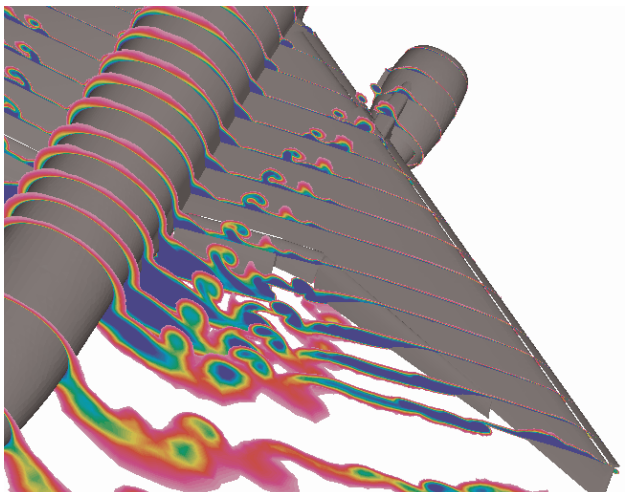


Figure 24. Total pressure contours from an unstructured solution at landing flap.

4.0 CONCLUDING REMARKS

Computational fluid dynamics based on the Navier-Stokes equations has demonstrated the ability to obtain solutions over the full flight envelope. Steady state solutions have included complete aeroplanes with control surface and spoiler deflections, vortex generators, and high-lift systems. In particular the use of CFD to predict Reynolds number trends has been presented. The required degree of accuracy demonstrated is not yet adequate in all cases. There is significant question whether any flow solution with significant flow separation can be considered and/or treated as steady. Continuing research and development of grid generation, discritisation and steady and unsteady solver technology, turbulence and transition modelling, etc. is addressing these deficiencies. Of equal or greater importance are the issues of usability, throughput, and robustness.

Validating CFD for acceptance by the product development engineers can be daunting. Higher levels of demonstrated accuracy are demanded of 'new CFD methods' than can be proven for existing empirical based processes. CFD is expected to duplicate the wind tunnel process yet the two are inherently different. Resolving these differences frequently requires increased attention to wind-tunnel model aeroelastics and more detailed tare and interference knowledge. The increased use of CFD in the development process is changing the amount and type of wind tunnel testing needed. The challenge is in determining the optimum complementary use of the two in advancing the aeroplane development process.

To be of use to the product development engineers these CFD solutions must be available over a wide range of Mach numbers. All relevant geometry must be included. The number of flight conditions and geometry variations that must be evaluated even for preliminary

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press

loads demands one or two orders-of-magnitude reduction in CFD cost and flow time, and corresponding improvements in robustness before it can be used effectively. CFD is showing that it is capable of yielding adequate results for many of the geometry/flight conditions of interest, however getting it into the formal product development engineering process still faces significant timeliness, technical, and cultural challenges. The impact of CFD in the greater aerodynamic development process will be as an enabler to more concurrent engineering reducing development cycle time, improving optimisation, and reducing the risk associated with new aircraft certification. Computational fluid dynamics will continue to see an ever-increasing role in the aircraft development process as long as it can provide value in the end users' eyes.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the contributions of M. Hong, J.E. Bussoletti, W. Rice, T.K. Matoi, L. Hedges and other members of Boeing Commercial Airplanes Enabling Technology and Development Group. Work presented here was supported by various independent sources of Boeing Commercial Airplanes.

REFERENCES

1. RUBBERT, P.E. and TINOCO, E.N. Impact of computational methods on aircraft design, August 1983, AIAA 83-2060.
2. BENGELINK, R.L. and RUBBERT, P.E. The impact of CFD on the airplane design process: today and tomorrow, *SAE Aerospace Engineering*, March 1992.
3. RUBBERT, P.E. CFD and the changing world of airplane design, September 1994, AIAA Wright Brothers Lecture, ICAS-94-0.2.
4. TINOCO, E.N. The changing role of computational fluid dynamics in aircraft development, June 1998, AIAA-98-2512.
5. JOHNSON, F.T., TINOCO E.N. and YU, N.J. Thirty years of development and application of CFD at Boeing Commercial Airplanes, Seattle, June 2003, AIAA-2003-3439.
6. CAPRON, W.K. and SMIT, K.L. Advanced aerodynamic applications of an interactive geometry and visualization system, January 1991, AIAA-91-0800.
7. YOUNG, D.P., MELVIN, R.G., BIETERMAN, M.B., JOHNSON, F.T., SAMANT, S.S. and BUSSOLETTI, J.E. A locally refined rectangular grid finite element method: application to computational fluid dynamics and computational physics, *J Comp Phys*, 1991, **92**, pp 1–66.
8. CHAKRAVARTHY, S., PALANISWAMY, S., GOLDBERG, U., PEROOMIAN, O., and SEKAR, B. A unified-grid approach for propulsion applications, 1998, AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Cleveland, July 1998.
9. BIEDRON, R.T. and RUMSEY, C.L. *CFL3D User's Manual* June 1998, NASA/TM-1998-208444.
10. VATSA, V.N., SANETRIK, M.D. and PARLETTE, E.B. Development of a flexible and efficient multigrid-based multiblock flow solver, 1993, AIAA paper 93-0699.
11. YU, N.J., SU, T.Y. and WILKINSON, W.M. Multiblock grid generation process for complex configuration analysis using Navier-Stokes codes, June 1995, AIAA-96-1995.
12. RUMSEY, C.L. and VATSA, V.N. A comparison of the predictive capabilities of several turbulence models using upwind and central-difference computer codes, January 1993, AIAA-93-0192.
13. SPALART, P.R. and ALLMARAS, S.R. A one-equation turbulence model for aerodynamic flows, 1992, AIAA paper 92-0439.
14. MENTER, F.R. Zonal two equation *k-w* turbulence models for aerodynamic flows, 1993, AIAA paper 93-2906.
15. YU, N.J., KAO, T.J. and BOGUE, D.R. Computational simulations of a commercial airplane configuration with vortex generators, 2000, Aerodynamics Conference, 2000, Royal Aeronautical Society, London, UK.
16. MINECK, R.E. Reynolds number effects on the performance of ailerons and spoilers, January 2001, AIAA Paper 2001-0908.
17. BOGUE, D.R.,TRAN, J.T., OM, D., RIVERS, S.M.B., PENDERGRAFT, O.C. and WAHLS, R.A. Scale effect investigation on the stability and control characteristics of an advanced twinjet configuration, June 2003, AIAA-2003-3401.
18. WILKINSON, W.W., LINES, T.R. and YU, N.J. Navier-Stokes calculations for massively separated flows, June 1996, AIAA-96-2383.
19. REICHENBACH, S.H. and MCMASTERS, J.H. A semiempirical interpolation technique for predicting full-scale flight characteristics, January 1987, AIAA Paper 87-0427.
20 JESPERSON, D.C., PULLIAM, T.H. and BUNING, P.G. Recent enhancements to OVERFLOW, January 1997, AIAA paper 97-0644.
21. ROGERS, S.E. *ET AL*. Computation of viscous flow for a Boeing 777 aircraft in landing configuration, June 2000, AIAA paper 2000-4221.

https://doi.org/10.1017/S0001924000000865 Published online by Cambridge University Press