Overview of Application Challenges in the Aeronautical Industry

J.B. Vos
CFS Engineering
PSE-B, CH-1015 LAUSANNE

A.W. Rizzi
Aeronautical Engineering
KTH, SE-1044 Stockholm

D. Darracq
CERFACS
42 Av. Coriolis
F-31057 Toulouse

INTRODUCTION

Computational Fluid Dynamics (CFD) is a relatively new technology, driven by the exponential increase in computer power, the development of efficient and robust numerical algorithms and the continuous improvements in physical modelling.

CFD has been used in the Aeronautical industry since the 1960s, first by using panel methods, followed by Euler and boundary layer methods, and since the last 10 years by methods which solve the Reynolds Averaged Navier-Stokes (RANS) equations.

In the early days of CFD, the principal objective of CFD was to gain new knowledge to improve the design of an aircraft, and CFD was used to complement windtunnel or flight experiments. Today, the confidence in CFD has grown such that CFD is used in the design, qualification, certification and operation of aircraft because it allows for reduced costs [1].

Although CFD is used on a routine base, it is not considered a mature technology as is for example Computational Structural Mechanics (CSM) [2]. Reasons for this are the use of turbulence models and/or other simplifying assumptions of the physics involved, the use of distorted grids for complex geometries, the high costs of CFD simulations, and the dependency of the results on the CFD expert running the code.

CFD simulations are still unable to predict absolute values of, for example, the lift and drag of an airplane. Design engineers using CFD therefore pose the question ‘‘What confidence do I have in the computed results on which I will base my design’’, and this question leads directly to the concept of uncertainty management [3]. CFD results are inherently uncertain, the question is how to assess and quantify this uncertainty, and how to translate this into useful information for a CFD user so that he can have trust in the results he obtained.

In the past, substantial effort was made to assess the capability of CFD codes for solving a variety of flow problems, usually in the form of comparison workshops. These efforts were generally focused on issues of numerical accuracy, and the prediction of detailed flow physics for simple problems and geometries. Only few attempts were made to assess the credibility of a complex CFD simulation.

The CFD Community only recently has turned its attention to credibility measurement and uncertainty management. In 1998, the AIAA published the Guide for Verification and Validation of Computational Fluid Dynamics Simulations [4], which gives good definitions of the terminology used in verification and validation. The guide includes sections on Verification Assessment and Validation Assessment, which give guidelines to improve the credibility of CFD simulations. Although the AIAA Guide provides a wealth of useful information, it remains rather conceptual without providing simple guidelines which can be used by an engineer running a CFD code.

In Europe, the Industrial Advisory Committee of the European Research Community on Flows, Turbulence and Combustion Association (ERCOFTAC) created a Special Interest Group on Quality and Trust in Industrial CFD, which commissioned Sulzer Innotec in Switzerland to write the ERCOFTAC Best Practice Guidelines for CFD [5]. The objectives of these guidelines are to give practical advice for making high quality CFD simulations, and to give relevant information to assess the credibility of such simulations. The ERCOFTAC Best Practice Guidelines is written for engineers running a CFD code, and in this respect is complementary to the AIAA guide.

Section 2 of this review will give an overview of the use of CFD in the Aeronautical industry. It is followed by a review of past and current European validation activities, and it is concluded in Section 4 with a discussion on building credibility in CFD simulations.

This review is for a substantial part based on the paper "Computational Aerodynamics for Industrial Airframe Design using Navier Stokes solvers" by J.B. Vos, A.W. Rizzi and D. Darracq, to be published by Progress in Aerospace Sciences.

USE OF CFD IN THE AERONAUTICAL SECTOR

Since the early use of CFD for industrial aircraft design in the 1960s, CFD has grown from a tool used to supplement wind tunnel or flight experiments to an identifiable new technology standing on its own making important important contributions to the early (as well as the later) stages of the design of a flight vehicle. Three factors were instrumental in this progress: 1) the increase in available computer resources, 2) the progress in development of efficient numerical methods, and 3) the progress in physical modeling. Here a short review is given of the different developments starting from the 1960s until the current use of Navier Stokes solvers in the airframe industry. Other authors have reviewed these developments extensively, for recent examples see Jameson [6], Jou [7], and Raj [8].

In the 1960s and early 1970s, CFD applied to aircraft design consisted of simplified (linear) models, i.e. the Laplace or Prandtl-Glauert equation (see [8] for a more thorough overview of progress during the first 30 years of CFD). Initially lifting-line and lifting-surface theories formulated these methods, then vortex-lattice procedures were developed representing the geometry by a mean surface and using vortex filaments as singularities. Since the mid-1960s panel methods arose that discretize the actual surface geometry with either low-order (constant) or higher-order (linear or quadratic) singularity distributions. Boundary-layer methods to study viscous effects became sufficiently mature in the early 1970s to be applicable in design. In this decade, much work was done on coupling panel methods with boundary-layer methods, and on developing nonlinear compressible potential formulations to treat transonic flows with shocks. These nonlinear inviscid methods were later coupled with boundary-layer computations. Perhaps the most outstanding example in this class is the TRANAIR code [9] which after several generations of evolution now includes multipoint design optimization accounting for geometry constraints and off-design optimization.

The advent of vector supercomputers (Cray 1, Cyber 205) at the end of the 1970s, opened the way to using non-linear methods for applications more complicated than isentropic, irrotational flows. The first Euler codes for research associated with aircraft design appeared in the 1980s, followed by the further development of Euler methods coupled with boundary layer codes. These were applied mainly to steady aerodynamics, while panel methods have been extended to handle unsteady problems.

By the end of the 1980s, a further increase in computer capacity became available in the form of (massively) parallel computers. This prompted the move from Euler to Navier Stokes simulations for steady flows, and the use of the unsteady Euler formulation for studying transient phenomena. At the same time concern arose for the need to integrate these research codes into the engineering design environment in order to meet the new challenges posed by the changing forces of the economic market for aircraft. Efforts are now underway to incorporate the extensive and existing body of CFD knowledge into the methods and routines that designers use for specifying new aircraft.

The 1990s brought a major paradigm change to the concept definition and design development of aircraft (as well as to other products). Before the doctrine of higher, faster and farther dominated the design (technology driven), but now the overriding theme is design to cost. The market, both civil and military, now demands a product with the technology to fulfill the mission and at a cost that the customer can afford. This change has motivated a search for ways and means to reduce the cost of designing and manufacturing technologically superior aircraft.

The traditional approach of aircraft design has been sequential, component by component. But the paradoxical fact of sequential aircraft design is that the decisions taken earliest in the design cycle impact most heavily the overall technical and financial success of the aircraft. Raj [8] among others, has pointed out that 70% to 90% of the life-cycle cost of an airplane is locked in during the early stages of design. Thus mistakes in these stages must be minimized because they are very expensive to correct when they are discovered in later phases. Therefore, in order to minimize such errors and correct them when there is still sufficient freedom and low cost, one needs as early as possible in the design cycle as much knowledge as possible about the design concept. New and improved tools are needed to accomplish this, especially for the conceptual, but also the later design phases.

Concurrent engineering address this need by allowing each subsystem design to proceed simultaneously, and it is supported by accurate data obtained through simulation for each contributing sub-discipline as well as for the complex interdisciplinary dependancies between them. As example, a wing design for a recent European civil aircraft required the computer simulation of over 800 design options for which the computer analyses were carried out over 2 years. To reduce this to 3 months, as the market may demand for a future aircraft of even more complexity will require an order of magnitude increase in design efficiency and productivity. This can be accomplished by carrying out the design activities simultaneously in each discipline using analysis and optimization tools, usually by teams working in close concert. These practices accelerate and improve the entire development process, reduce costs and improve the quality of the design.

The concurrent engineering approach has been introduced at the different partners of Airbus. However, within aerodynamics, further enhancement and validation of the numerical tools is necessary to increase the utilization of CFD tools in the design process [10], and to meet future challenges in aircraft design, as for example smart wings, laminar wing designs, etc etc.

PAST AND CURRENT EUROPEAN VALIDATION ACTIVITIES

In Europe, validation activities for CFD codes used in the Aeronautical industry have undergone large changes in the last decade. Until about 10 years ago, validation of CFD codes was mainly an activity of the national aeronautical research establishments in collaboration with the national aeronautical industry, and validation activities were mainly funded using national and/or company funds. Some joint validation activities were going on inside the NATO AGARD Fluid Dynamics Panel (now called NATO/RTO), in which the US and Canada are participating too, and inside the GARTEUR group.

Results of these activities are only partly available in the open literature, as for example the three AGARD reports on experimental test cases for validation [11], [12], [13].

About 10 years ago, driven by the closer collaboration of the European Aeronautical industry in the Airbus consortium, the first international projects concerned with the validation of CFD codes were funded by the European Commission.

The EUROVAL (**Euro**pean Initiative on **Val**idation of CFD Codes) project [14] ran from February 1990 to April 1992, and had as aim to improve CFD codes by careful validation against experiments. The project had 16 partners from 11 European countries. Nine test cases were considered, and they included airfoils (ONERA A-Airfoil, RAE-2822

Airfoil, NLR-7301 two element airfoil), 2D Channels flows for study of shock-wave boundary layer interaction, the DLR-F5 wing, 2D and 3D boundary layer test cases, a windtunnel interference case, and a vortex break down test case. For each of the test cases, a mandatory grid was used with a mandatory set of input parameters. This test case was then computed using different codes, and using different turbulence models. As example, the RAE-2822 Airfoil case 9 was computed 20 times, using 10 different codes and 9 different turbulence models, ranging from algebraic turbulence models to Reynolds stress models. Computed C_L for this test case varied between 0.647 to 0.837, with an experimental value of 0.803. Besides the mandatory test case, the influence of the grid density, and windtunnel correction parameters on the results were studied.

The EUROVAL project was one of the first collaborative European efforts on systematic CFD validation, and it contributed to the creation of an European CFD community.

The ETMA (**E**fficient **T**urbulent **M**odels for **A**eronautics) project [15] ran from 1992 to 1995 and aimed at the development of "Numerical Turbulence Models" through well coordinated efforts on both the physical modelling and numerical methods in order to significantly improve predictions in aeronautical applications. The activities in this project were both numerical and experimental. The numerical activities focused on improving turbulence models for compressible flows, in particular on modelling improvements, accuracy improvements, and more efficient solution algorithms.

The ECARP (**E**uropean **C**omputational **A**erodynamics **R**esearch **P**roject) project ran from 1993 to 1995 had as primary aim to improve the quality of industrial CFD codes by improving their accuracy, reliability and computational efficiency. One of the activities of this project was validation, which focused on quantifying the predictive accuracy of advanced modelling techniques. The results of the validation studies are published in [16], which include a CD-ROM with all the relevant data generated during the project. Test cases considered were high lift, single and multi element airfoils, a wing body configuration, an inclined spheroid, a skewed channel bump and a 2D separating boundary layer.

The AVTAC (**A**dvanced **V**iscous flow simulation **T**ools for Complete Civil Transport **A**ir**C**raft Design) project [17] ran from 1997 to 2000, and had as objectives to enhance the levels of robustness, efficiency and validity of industrial three-dimensional viscous flow simulation tools. The specific objective for validation was to improve the prediction of key design parameters to within 1-2% compared with experiment, compared to 5-10% at the start of the project. Among the test cases were the AS28G wing-body-pylon-nacelle configuration, and the RAE M2155 swept wing. The project results are available on a CD-ROM.

Besides projects funded by the European Commission, other collaborative activities related to the validation of codes started in the early 1990's.

The project to build a European Space Shuttle, called Hermes, has contributed largely to a closer collaboration in Europe between the different aerospace industries, aeronautical research establishments and universities. Three workshops were organized at INRIA Sophia Antipolis in the period 1990-1993. The results were published in a book [18], and are available in a electronic data base. Owing to the rapid progress in date storage capacity, contributors to the workshop were required to submit their data in electronic format, and during the workshop real-time comparisons were made of the different contributions, resulting in an improved understanding of the computed results. The series of workshops continued after the Hermes program with 2 joint US-Europe High Speed Flow Field workshops, the first organized in Houston in 1995, the second organized in Naples in 1997. The test case description, experimental and CFD data, and contributed papers of the third INRIA workshop and of the 2 joint US-Europe workshops can be accessed from the WWW (http://hhsfd.math.uh.edu/).

The database system and tools developed during these different workshops continue to be used and improved in the FLOWNET thematic network [19] funded by the European Commission in 1998. The objective of FLOWNET is to build a network of expertise on code validation by setting up a data base tool on the World Wide Web in which computational and experimental data are stored. The ultimate objective of FLOWNET is to evaluate continuously in terms of accuracy and efficiency CFD software for industrial design. FLOWNET has 26 partners from industry, research establishments and universities, most of them active in the aerospace sector. The first FLOWNET workshop was held in Rome in March 2000, and a Von Karman Institute short course on Validation was organized in June 2000 [20]. The second FLOWNET workshop was organized at DLR in Gottingen in February 2001, and the third and final FLOWNET Workshop will be organized in April 2002 in Marseille [21]. The FLOWNET data base contains 29 test cases for the Aeronautical sector, 3 subsonic ones (including the A-Airfoil and NLR 7301 multi element airfoil used in EUROVAL, and the 3D Spheroid of ECARP), 4 transonic test cases which include the Skewed Channel bump of ECARP and the RAE M2155 of AVTAC, 13 supersonic cases, and 9 hypersonic cases.

The ERCOFTAC (European Research Community on Flows, Turbulence and Combustion) association was founded in 1988. ERCOFTAC promotes joint activities of European research institutes and industries active in all aspects of Flow, Turbulence and Combustion. Members of ERCOFTAC are industries, research establishments and university laboratories, and the actual membership base goes across all industrial sectors. ERCOFTAC has a matrix organization composed of Pilot Centres in a country or region, and Special Interest Groups which are organized on a specific topic in flow, turbulence and combustion.

Since 1993, ERCOFTAC has been active in setting up data bases for validation. A European Commission funded project "Data Validation and Comparison in Fluid Mechanics" aimed to collect experimental and numerical data on turbulent flows, to check the data for their reliability and suitability for test cases, to set up test cases and perform calculations with various turbulence models and to create a

data bank from which the data can be accessed. The project collected data for over 77 flows, and the test cases can be accessed through the the WWW. ERCOFTAC Special Interest Groups are stimulated to organize workshops, and to make the data available in electronic format.
Recently, the ERCOFTAC WWW data base server was completely revised, and it now provides links to different data bases developed through ERCOFTAC activities or made available by ERCOFTAC members. More information on these data bases can be found at http://ercoftac.mech.surrey.ac.uk/.

BUILDING CREDIBILITY IN CFD SIMULATION

CFD will always remain an uncertain discipline, hence the importance to reduce the uncertainty in CFD simulations to increase the credibility of CFD results. The allowed uncertainty of a CFD simulation strongly depends on the application and on the objective of the simulation, and no general rules are available.
Confidence in the results of a CFD simulation is obtained when the code user understands why a CFD code produces a certain answer [3]. Building credibility of CFD simulations is a joint effort between code developers (who should make certain that their codes are running correctly), code users (who should make sure that no errors have slipped in their simulation), and experimental groups (who should provide the CFD community with high quality experimental data for CFD code validation).

CFD CODE DEVELOPERS

Code developers should demonstrate that they have verified that their code solves correctly the equations and boundary conditions of the physical model, and they should provide information on validation test cases run with this particular version of their code. The ERCOFTAC Best Practice Guidelines includes several recommendations to code developers [5]. The most important are listed here, and we believe that it is important that code developers follow these recommendations:

- to use quality control procedures in code development and maintenance ;
- to demonstrate that the code developer has verified that the code solves the equations of the physical model ;
- to publish in electronic format a data base of validation test cases (including the options used to run the code) ;
- to include in the code messages to warn the user when basic rules, for example, concerning grids, are broken ;
- to provide the user with as many possibilities to judge convergence (residuals for each equation, mass flow, ..) and to estimate the error.

CFD CODE USERS

CFD code users should demonstrate that they have eliminated all potential sources of errors in their CFD simulation, and they should assess the uncertainty in the results obtained. Quality Assurance procedures should be defined, and CFD Users should demonstrate that they have followed them. Once CFD Users have made certain that no errors have slipped in their simulation, and that they have obtained a good quality result, they should assess the numerical and modelling uncertainties in the simulated results [3], [22] by studying the:

- influence of grid refinement (or grid coarsening if refinement is not possible).
- A simple method to assess grid convergence is the Grid Convergence Index (GCI) [23], which is based on a Richardson extrapolation involving comparison of numerical solutions at at least 2 different grid spacings. The GCI is a simple means to verify that refining the grid leads to grid convergence ;
- influence of the time convergence level and of the spatial discretization order ;
- sensitivity of the results to physical modelling (turbulence model, transition location, ..) ;
- sensitivity of the results to boundary conditions ;
- sensitivity of the results to geometrical variations.

To perform a credible CFD simulation, CFD users need guidelines on how to run a CFD code, and they need information on how to run the CFD code for the particular application they are studying (grid resolution, most suitable turbulence models, etc), so called Application Procedures.
The guidelines are available as the ERCOFTAC Best Practice Guidelines [5], and Application Procedures will in future become available in the QNET-CFD thematic network as a knowledge base of Application Challenges.

VALIDATION EXPERIMENS

Validation of CFD codes require high quality experimental data on well defined experiments, which are defined in a joint effort between experimental and CFD people [22]. A discussion on the design, execution and analysis of validation experiments can be found in the paper by Oberkampf [24]. This paper includes a section on "Recommended Procedures for Validation Experiments", which is briefly summarized here:

- determine the free-stream calibration data of the flow at the spatial resolution consistent with the requirements of the CFD simulation. These data should include turbulence intensity, and information on the flow uniformity at the inflow ;
- precisely characterize the boundary conditions on the surface of the model. For example, surface roughness, gaps in the model, height of the trip wire, location of instruments, etc may have an influence on the measured results, and should be taken into account in the CFD simulation ;
- do the same experiment in different facilities ;
- apply redundant measurement techniques for critical experimental variables ;
- try to identify random and bias errors ;

- rotate the model 180 degrees, and plot the results for varying incidence for the measurements at the original and rotated position ;
- document the execution of the experiment as detailed as possible.

WORKSHOPS AND DATABASES

Workshops are an excellent means to increase the understanding of the physics of flows and to test and assess physical models. However, very often the results of workshops are not available in electronic form, workshops results are not always complete, and expert comments made during the workshop have not been archived. Workshops will remain very important in CFD code validation, but it is required that:

1. workshops are not limited to simple test cases but also should include industrial applications ;
2. workshop data is checked on quality ;
3. workshop data is archived in electronic data bases which preferably are freely accessible ;
4. workshop results are analysed, and that the conclusions and recommendations are made available together with the data.

The Workshop data base of the future has been defined before [22], [25]: it should be accessible on the WWW, include both experimental and computational data together with synthesis documents and expert comments. Tools should be made available to manipulate data to generate synthesis documents, and allow users to add comments. Groups wanting to validate their code, or test a new physical model for a test case should be given access to the data base data and its tools, and once they are satisfied with their results, they should include their results in the data base. Synthesis documents, which compare CFD calculations with experimental data, can then be updated and saved in the data base.
The FLOWNET data base [14] is a first step in this direction, although for the moment its access is restricted to the members of the FLOWNET network alone.

Finally it should be remarked that data bases die when they are not used anymore. It is important that data bases are being used, that new data is feed in, and that funds are made available to maintain them [22].

REFERENCES

[1] Perrier P., Verification and Validation: Industrial Issues, VKI Lecture Series on Verification and Validation of CFD, June 2000.
[2] Melnik R.E., Siclari M.J., Marconi F., Barber T.An Overview of a Recent Industry Effort at CFD Code Certification, AIAA Paper 95-2229, June 1995.
[3] Rubbert P.E. The use of CFD in Airplane Design, CFD 97 Fifth Annual Conference of the CFD Society of Canada, Victoria, B.C., 1997.
[4] Guide for the Verification and Validation of Computational Fluid Dynamics Simulations, AIAA G-077-1998.
[5] ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD - Best Practice Guidelines. ERCOFTAC, 2000.
[6] Jameson A. Essential Elements of Computational Algorithms for Aerodynamic Analysis and Design NASA CR-97-206268/ICASE Report No. 97-68, 1997.
[7] Jou Wen-Hui, A Systems Approach to CFD Code Development, ICAS-98-2,7,5, 1998.
[8] Raj P. CFD at a Crossroads: An Industry Perspective, In: Frontiers of Computational Fluid Dynamics 1997, Eds. D.A. Caughey and M.M. Hafez
[9] Huffman et al. Practical Design and Optimization in CFD, AIAA Paper 93-3111, 1993.
[10] Szodruch J., Hilbig R. Building the Future Aircraft Design for the Next Century. AIAA 98-0135.
[11] Experimental Data Base for Computer Program Assessment, AGARD-AR-138, 1979
[12] Experimental Data Base for Computer Program Assessment: Addendum, AGARD-AR-138-Addendum, 1984
[13] A Selection of Experimental Test Cases for Validation of CFD Codes, Vol. I and II, AGARD-AR-303, 1994
[14] Haase W., Bradsma F., Elsholz E., Leschziner M., Schwamborn D., EUROVAL - An European Initiative on Validation of CFD codes, Notes on Numerical Fluid Mechanics, Vol. 42, Vieweg, 1992.
[15] Dervieux A., Braza, M., Dussauge J.P. Computation and comparison of Efficient Turbulence Models for Aeronautics: European Research Project ETMA. Notes on Numerical Fluid Mechanics, Vol. 65, Vieweg, 1998
[16] Haase W., Chaput, E., Elsholz, E., Leschziner, M., Muller, U.R., ECARP - European Computational Aerodynamics Research Project: Validation of CFD codes and Assessment of Turbulence Models. Notes on Numerical Fluid Mechanics, Vol. 58, Vieweg, 1997.
[17] Gould A. et al. The AVTAC project - A review of European Aerospace CFD Proceedings of ECCOMAS 2000.
[18] Abgrall R., Desideri J.-A., Glowinski R., Mallet M., Periaux J. (Eds.) Hypersonic Flows for Reentry Problems, Vols 1-3,Proc of INRIA-GAMNI/SMAI Workshop Part I and II, Antibes, April 1990-1, Springer, Berlin, 1992
[19] FLOWNET: Flow Library on the Web Network http://www-sop.inria.fr/sinus/flownet/index.php3 INRIA Sophia Antipolis and Dassault Aviation, 1998.
[20] Grasso F., Periaux J., Deconinck H. (Lecture Series Directors), Verification and Validation in CFD, VKI-Lectures Series, June 2000.
[21] Marini M. and Paoli R. erification and Validation of CFD through the use of a Data base, roceedings ECCOMAS CFD Conference 2001.
[22] Haase W., Electronic Data Base Tools for CFD. Proceedings of ECCOMAS 1998.
[23] Roache P.J. Verification of Codes and Calculations, AIAA Journal, 1998, Vol. 36, No. 5, pp. 696-702.

[24] Oberkampf W.L., Design, Execution, and Analysis of Validation Experiments. In: Verification and Validation in CFD, VKI-Lectures Series, June 2000.

[25] Rizzi A.W., Vos J.B., Toward Establishing Credibility in Computational Fluid Dynamics Simulations, AIAA Journal, 1998, Vol. 36, No. 5, pp. 668-675.