QNET-CFD KNOWLEDGE BASE – A PLATFORM FOR THE PRESERVATION OF KNOWLEDGE GENERATED BY EU FUNDED PROJECTS

J.B. Vos[†], A.G. Hutton[‡], Ch. Hirsch^{*} [†]CFS Engineering, PSE-B, 1015 Lausanne, Switzerland [‡]Qinetiq, Cody Technology Park, Farnborough GU14 OLX, U.K. ^{*}NUMECA, 5 Avenue Franklin Roosevelt, 1050 Brussels, Belgium

ABSTRACT

The QNET-CFD project was an EU funded thematic network on Quality and Trust for Industrial application of CFD which ran from 2000 to 2004. The outcome of the project is the Knowledge Base which is organized around 6 Thematic Areas. 53 Industrial Application Challenges and 43 Underlying Flow Regimes are saved in the Knowledge Base. Quality control procedures were implemented to review each of these cases to ensure that and documentation satisfied the data quality requirements. For each of the Application Challenges Best Practice Advice on how to compute this industrial challenge was prepared using the Underlying Flow Regimes. This advice is also saved on the Knowledge Base, making it a unique source of knowledge that will contribute to increase the level of trust in industrial CFD.

At the end of the QNET-CFD project the Knowledge Base was adopted by the ERCOFTAC Association, which will make it available to the public. It is planned that the Knowledge Base will be continuously enriched. One major source for new or updated information on Application Challenges and Underlying Flow Regimes are EU funded projects. Many of such projects generate a wealth of knowledge on using and applying CFD. But in general most of this knowledge is lost a few years after the project is finished and as such it can be considered a waste of effort and money. Saving flow cases from EU funded projects in the QNET-CFD Knowledge Base would preserve the knowledge generated, and, owing to the quality review process, it ensures that the experimental and CFD results meet quality standards.

ACRONYMS

- AC **Application Challenge** AIAA American Institute of Aeronautics and Astronautics **BPA Best Practice Advice** BPG **Best Practice Guidelines** CFD **Computational Fluid Dynamics** DES **Detached Eddy Simulation** ERCOFTAC European Research Community on Flows, **Turbulence and Combustion** ESA European Space Agency EU **European Union**
- KB Knowledge Base

LESLarge Eddy SimulationRANSReynolds Averaged Navier StokesTAThematic AreaUFRUnderlying Flow Regime

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is a relatively recent technology that is driven by the exponential increase in computer power, the development of efficient and robust numerical algorithms and the continuous improvements in physical modeling. Although CFD plays today an essential role in the design of industrial products, there is a common agreement that Computational Fluid Dynamics is a difficult technology to apply with success in an industrial environment because CFD is an uncertain discipline and as such is a knowledge based activity.

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 mid 1990s by methods which solve the Reynolds Averaged Navier-Stokes (RANS) equations. Today the Aeronautical industry is using more and more steady and unsteady CFD methods coupled with structural mechanics simulation tools (static and dynamic Fluid Structure Interaction), or performs unsteady CFD simulations at the extremes of the flight envelope to obtain, for example, unsteady loads needed for structural design.

In the early days of CFD, the principal objective was to gain new knowledge to improve the design of an aircraft, and it was used to complement wind tunnel or flight experiments. Today, confidence in CFD has grown to the extent that it is used in the design, qualification, certification and operation of aircraft because it allows for reduced costs [1]. This is illustrated by the following statement of Johnson et al [2] from the Boeing company:

"The application of CFD has revolutionized the process of aerodynamic design. The effective use of CFD is a key ingredient in the successful design of modern commercial aircraft. The combined pressures of market competitiveness, dedication to the highest of safety standards, and desire to remain a profitable business enterprise all contribute to make intelligent, extensive, and careful use of CFD a major strategy for product development at Boeing. The advances in computing technology over the years have allowed CFD methods to affect the solution of problems of greater and greater

relevance to aircraft design. Use of these methods allowed a more thorough aerodynamic design earlier in the development process, permitting greater concentration on operational and safety-related features".

Although CFD is used on a routine basis, it is not considered a mature technology as is for example Computational Structural Mechanics (CSM) [3]. 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 [4]. CFD results are inherently uncertain; the question is how to assess and to 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.

At the end of the last century the CFD Community 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 [5], which provides 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 (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 [6]. 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 were written for engineers running a CFD code, and in this respect are complementary to the AIAA guide.

As mentioned earlier a large effort has been placed in the validation of CFD codes and in the development of numerical techniques and physical models for specific flow cases. But little effort has been made to preserve the expert knowledge on how to use a CFD code so that the results of a particular simulation can be trusted and used with confidence.

The EU funded thematic network QNET-CFD (Quality and Trust for the Industrial Application of Computational Fluid Dynamics) ran from 2000 to 2004, and was the first attempt to assemble, structure and collate existing knowledge on the industrial application of CFD. The outcomes of the project are the QNET-CFD Knowledge Base, quality control procedures for CFD calculations and experimental data assessment, and Best Practice Advice for using CFD on Underlying Flow Regimes and Industrial Application Challenges.

At the end of the QNET-CFD project, the QNET-CFD Knowledge Base was adopted by the ERCOFTAC association, and it will in the near future become available to the public.

Chapter 2 of this paper gives an overview of the past EU and ESA funded projects with elements of CFD validation. Chapter 3 gives an overview of the QNET-CFD project and presents the QNET-CFD Knowledge Base (KB). Chapter 4 gives some examples of Industrial Application Challenges saved in the KB and Chapter 5 discusses the use of the QNET-CFD Knowledge Base as instrument to preserve knowledge generated in EU funded projects.

2. OVERVIEW OF PAST EU AND ESA FUNDED PROJECTS

2.1. EU-Funded Projects

About 17 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 (European Initiative on Validation of CFD Codes) project [7] 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 the study of shock-wave boundary layer interaction, the DLR-F5 wing, 2D and 3D boundary layer test cases, a wind tunnel 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. For 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 CL for this test case varied between 0.647 to 0.837, with the measured value being 0.803. Besides the mandatory test case, the influence of the grid density, and wind tunnel 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 (Efficient Turbulent Models for Aeronautics) project [8] ran from 1992 to 1995 and aimed at the development of "Numerical Turbulence Models" through

well coordinated efforts on both the physical modeling 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 modeling improvements, accuracy improvements, and more efficient solution algorithms.

The ECARP (European Computational Aerodynamics Research Project) project, which ran from 1993 to 1995, had as its primary aim the improvement in the accuracy, reliability and computational efficiency of industrial CFD codes. One of the activities of this project was validation, which focused on quantifying the predictive accuracy of advanced modeling techniques. The results of the validation studies are published in [9], 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 (Advanced Viscous flow simulation Tools for Complete Civil Transport AirCraft Design) project [10] ran from 1997 to 2000. Its aim was enhance the levels of robustness, efficiency and validity of industrial threedimensional viscous flow simulation tools. The specific objective for validation was to improve the prediction accuracy of key design parameters to within 1-2%, 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.

The LESFOIL (Large Eddy Simulation of Flows around Airfoils) project [11] ran from 1997 to 2001. Its main objective was to assess the feasibility of using Large Eddy Simulation to calculate the flows around airfoils. The Aerospatiale A-Airfoil at maximum lift was used in this study, because experimental data is available, and because this test case was used in two other EC projects (ECARP and EUROVAL), from which RANS simulations results were available. The LESFOIL project addressed all key aspects determining the quality of LES: numerical methods, subgrid scale models, mesh resolution, wall resolution, initial conditions, boundary conditions, time averaging etc. The project results contain a wealth of information on the topic of LES for Airfoils, which are published in a book [12]. The conclusions of the project were that significant advances in understanding of the application of LES to airfoils had been made, and that successful simulations of this type of flow can be made by using a well-resolved LES in which the near wall turbulent structures are adequately resolved and transition is properly simulated.

The FLOMANIA project (FLOw physics Modeling – An Integrated Approach) [13] aimed to develop robust and reliable turbulence models for RANS (and Unsteady RANS) applications. One of its long term goals is to use DES methods for validation and for evaluating the range of validity of RANS models. This EC Funded project started in 2002, and ran June 2004. Although it is not a CFD validation project, the list of test cases is rather large, and it includes several test cases also used in QNET-CFD.

The DESider project (Detached Eddy Simulation for Industrial Aerodynamics) [14] ran from 2004 to 2007 and was greatly motivated by the increasing demand of the European aerospace industries to improve their CFDaided design procedure and analysis for flows that exhibit "massive" separation. The RANS modelling approach commonly used in industry appeared/proved to be poorly when dealing with complex turbulent adapted separated/vortical flows. While LES has shown viable capabilities of resolving the flow structures and achieving more accurate predictions, it is too costly to be used at present in aeronautical applications (even for a single airfoil at high Reynolds numbers). To close the gap between RANS and LES, a class of hybrid RANS-LES methods have been previously developed, among which the DES (Detached Eddy Simulation) approach served as a basis for the DESider project. The ultimate goal of DESider was to obtain RANS-LES method(s) being mature and applicable to industrial real-world application with an improved predictive accuracy. The DESider project finished in June 2007, and the project results will be published in the Notes on Numerical Fluid Mechanics book series

2.2. ESA Funded projects

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 [15], 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. These were followed by a joint US-European-Japanese workshop in 1998. The test case description, experimental and CFD data, and contributed papers of the third INRIA workshop and of the 2 joint US-Europe workshops could be accessed from the WWW (http://hhsfd.math.uh.edu/). These series of workshops are now continued as West-East High Speed Flow Field conferences, with the next one to be held in Moscow in November 2007.

The database system and tools developed during these different workshops continued to be used and improved during the FLOWNET thematic network [16] funded by the European Commission in 1998. The objective of FLOWNET was to build a network of expertise on code validation by setting up a data base tool on the World Wide Web by which contributors can store and compare computational and experimental data—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 [17]. The second FLOWNET workshop was organized at DLR in Gottingen in February 2001, and the third and final FLOWNET Workshop was organized in April 2002 in Marseille [18]. 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.

3. QNET-CFD PROJECT OVERVIEW

3.1. OBJECTIVES

The EU funded thematic network QNET-CFD (Quality and Trust for the Industrial Application of Computational Fluid Dynamics) ran from 2000 to 2004. Its objectives were :

- To assemble, structure and collate existing knowledge on the industrial application of CFD and to make these available to European industry;
- To improve the quality of the industrial application of CFD through a common approach;
- To improve the level of trust that can be place in industrial CFD calculations by assembling, structuring and collating existing knowledge encapsulating the performance of models underlying the current generation of CFD codes;
- To establish a shared database of computational and experimental results to support industrial applications;
- To provide a regular state-of-the-art review on quality and trust;
- To promote technology transfer between industries through workshops, regular meetings and electronic communication;
- To identify gaps in existing knowledge and to stimulate new consortia and projects for EU funding.

3.2. ORGANIZATION

The QNET-CFD project was managed by Prof. Hirsch from the Vrije Universiteit of Brussels (VUB) and had 44 partners from research and industry which included the four major CFD software vendors.

The QNET-CFD project was organized around the six following Thematic Areas (TA):

- TA1 External Aerodynamics
- TA2: Combustion & Heat Transfer
- TA3: Chemical & Process, Thermal Hydraulics and Nuclear Safety

- TA4: Civil Construction & HVAC
- TA5: Environment
- TA6: Turbomachinery

The network steering committee of QNET-CFD was composed of the project coordinator, the assistant coordinator, the coordinators of the six TA's together with the quality coordinator and the scientific coordinator. The role of the quality coordinator was to provide definitions of Industrial Application Challenges (ACs) and Underlying Flow Regimes (UFRs), to establish criteria for assessing the quality of the proposed ACs and UFRs and to assure the quality of the overall project outcome. The role of the scientific coordinator was to assist the quality coordinator in defining the requirements of the documentation of the ACs and UFRs, coordinate the identification of UFRs in the proposed Application Challenges, coordinate the identification of Best Practice Advice (BPA) and edit and review the state of the art report provided by each Thematic Area coordinator.

The 44 partners of the QNET-CFD project were distributed among the Thematic Area's; several partners participated in more than one TA.

3.3. THE QNET-CFD KNOWLEDGE BASE

The outcome of the QNET-CFD project is the Knowledge Base (KB). The design specification of the Knowledge Base required that it would [19]:

- be accessible via the Internet;
- have a user-friendly structure;
- support the interpretation of existing knowledge;
- provide best practice advice;
- promote knowledge transfer across industry sectors;
- help to identify gaps in current knowledge;
- be expandable and grow to include new knowledge.

To streamline the incorporation of documents and numerical data provided by the members of QNET-CFD templates were created which provided guidance on what should be included and the minimum level of detail. The objective was to ensure a common format for the presentation of the data and a common level of information.

These templates were refined in consultation with the QNET-CFD Quality and Scientific Coordinators who defined a rigorous quality review process for all entries into the Knowledge Base. All the Application Challenges, Underlying Flow Regimes and the associated Best Practice Advice were reviewed before being accepted for inclusion. This review process was recorded in the form of 'quality checklists' which are also saved on the

Knowledge Base.

The Knowledge Base is structured around the six Thematic Areas and four categories of Underlying Flow Regimes. Under each Thematic Area there is a library of industrial Application Challenges (ACs) which are linked to the Underlying Flow Regimes (UFRs) that underlie the industrial applications, see Fig. 1.



FIG 1. Structure QNET-CFD Knowledge Base.

An **Application Challenge (AC)** is defined as an industrial test case with the following attributes:

- it represents (wholly or elements of) the fluid dynamic interests of an industrial sector (i.e. the design or assessment challenges faced by the sector);
- Considerable understanding of the challenge is available probably as a result of years of study involving test rigs or field trials or as a result of product experience;
- CFD simulations of the AC are available. These may reveal that CFD is competent but are more likely to demonstrate that CFD is inadequate and poorly understood for this AC.

The parameters, which are of importance to design assessment, must be defined and accurate measurements of these must be available. These last two requirements are clearly a pre-requisite for judging the competency of CFD.

In short, an AC is a test case by which the competency of CFD for the associated industrial sector is to be judged.

An Underlying Flow Regime (UFR) is defined as:

- a generic flow configuration or process which captures a key element of the fluid physics associated with one or more Application Challenges;
- Detailed measurements and well resolved CFD

simulations are available;

 Has undergone a quality review procedure to assess the level of reliability of both experimental and CFD data.

Each UFR can be linked to several Application Challenges, as shown schematically in Fig. 1. The UFRs themselves are grouped in four categories:

- Free flows (for example jets, plumes, free vortex flows);
- Flows around bodies (for example flows around cylinders, airfoils, blades);
- Semi-confined flows (boundary layer flows, boundary layer-wake interaction, shock-boundary layer interaction, buoyant boundary layer on heated wall, wall jets, impinging jet);
- Confined flows (pipe flow, diffusor flow, channel flow, cavity flow).

A key element in the creation of the Knowledge Base was the definition and implementation of **Quality and Control Procedures** which were used to:

- Filter the AC's and the UFR's against quality requirements, assessing the level of reliability of the experimental as well as the related CFD data;
- Identify the best and most representative underlying flow regimes;
- Ensure that the documentation of data and/or calculations connected to the AC or UFR is sufficient.

At the end of the QNET-CFD project in July 2004, 53 Application Challenges and 43 Underlying Flow Regimes together with Best Practice Advice for each of these cases were saved in the Knowledge Base. The QNET-CFD Knowledge Base can be accessed at: http://eddie.mech.surrey.ac.uk/

4. EXAMPLES OF APPLICATION CHALLENGES IN AERONAUTICS

Some application challenges which are representative of the TA1 library are described below

4.1. RAE M2155 wing

The RAE M2155 wing has been the subject of many numerical simulations and was used to validate and assess turbulence models in several EU funded projects. The wing is swept, of low-aspect ratio, and has the plan form shown in Figure 2. Experiments were performed at the DERA 8ft x 6ft transonic wind tunnel in the Mach number range 0.6 - 0.87 and at a Reynolds number (based on the geometric mean chord) of 4×10^6 .

The wing presents a complex flow field with threedimensional separations and triple shock wave structures. The boundary layers are subject to strong adverse pressure gradients (the trailing edges are heavily loaded), a regime which is difficult for numerical methods but of great importance in wing design. Case 2, with a Mach number of 0.806 and an angle of attack of 2.5°, is the most severe, and has been used for QNET-CFD. Application uncertainties for this case are the influence of the tunnel walls and the interaction between the boundary layers on the tunnel wall and the wing. For this reason, CFD simulations should include the wind tunnel walls. The following Underlying Flow Regimes are associated with this application challenge:

UFR3-03: Boundary layers with pressure gradients UFR3-05: Shock - boundary layer interaction UFR3-08: 3D Boundary layers subject to strong adverse pressure gradient causing separation



Figure 2: Pressure contours and skin friction lines RAE M2155 wing

Best Practice Advice particular for this test case are:

- Use at least 10 grid points in the stream wise direction across the shock
- Fix the transition locations in the same way as in experiments
- Use turbulence models with non-linear constitutive relation or the Menter SST k-ω model to predict the shock location, the pressure recovery behind the shocks and velocities in zones with flow separation

4.2. L1T2 MULTI ELEMENT AIRFOIL

This application challenge is focused on one of the 2D high lift configurations (the L1T2 test case). The L1T2 case is a 3 element aerofoil consisting of a main element, a slat forward of the main element (deflection angle 25^{0}), and a Fowler flap aft of the main element (deflection angle 20^{0} , see Figure 3. Measurements were made at two incidences, one at a low angle of attack and one close to maximum lift. The main flow physics is characterized by strong interactions between the turbulent boundary layers and wakes of the slat/wing/flap elements. The interaction

between a wake and a downstream boundary layer can lead to boundary layer thickening and separated flow. The flow is considered to be two-dimensional. The L1T2 configuration was tested at a Mach number of 0.197, using a Reynolds number of 3.52×10^6 (based on retracted chord) and at angles of incidence (corrected) of 4.01° and 20.18°.

The relevant UFRs are UFR3-01 (Boundary Layer - Wake Interaction NLR 7301), UFR3-03 (2D Boundary Layers with pressure gradients) and UFR3-04 (Laminar-turbulent boundary layer transition). However, as no test case is available for UFR3-04, no advice can be derived from this.



Figure 3: Mach number contours L1T2 Multi Element Airfoil

Application uncertainties for this test case concern the transition location, which is not specified and may have a significant influence on the overall forces, in particular at high incidence angle, and the resolution of the wake.

Best Practice Advice for this particular test case is:

- If a far-field circulation correction method is used at the far-field boundary, ensure that the far-field boundary is at least 15 chord lengths away. If not, ensure that the far-field boundary is at least 50 chord lengths from the body.
- Use the full, compressible, Reynolds Averaged Navier-Stokes formulation.
- If an accurate prediction of the pressures on the lifting surfaces (and so lift coefficient) is required, USE the k-ω turbulence model.
- Accurate prediction of profiles on boundary layers and wakes: It is not possible to give advice on which turbulence model to use for the accurate prediction of boundary layer profiles and wakes since the advice of UFR3-01 is inconsistent with the evidence of the Application Challenge.

4.3. CHANNEL FLOW WITH WALL INJECTION

The channel flow with wall injection is representative of the flow in a solid fuel rocket chamber. Figure 4 shows the experimental set-up for this application challenge. The flow in this closed head-end channel is completely induced by the lateral injection of mass. The injection is massive (i.e. the injection velocity has the same order of magnitude as the friction velocity) and the turbulence observed is rather due to gas release than friction, which is an unusual behavior. The channel is ended by a choked nozzle which implies rather large pressure gradients as well as compressibility effects for the flow (aft part of the channel). Transition (which occurs mid-channel) is important for the correct description of the flow. Application uncertainties mainly concern the injection conditions: the injection velocity is slightly non-uniform due to the varying pressure in the feeding tube, and the turbulence level of the injected flow is unknown. The latter uncertainty may alter the transition location.



Figure 4: Experimental set up channel flow with mass injection

The associated UFR for this Application Challenge is UFR4-07 (Developing channel flow with mass injection through wall), which has similar physics.

Best Practice Advice for this particular test case is:

- A 2D formulation is sufficient
- It is not needed to include the choked nozzle in the CFD simulations
- The turbulence intensity at the solid wall with mass injection is a key issue. It is recommended to prescribe a turbulent velocity of the order of 10% of the injection velocity, and a turbulent length scale of the order of the wall pore size. These values are based on experience: it is recommended to carry out a sensitivity analysis for these quantities.
- Mean axial velocity and pressure are accurately predicted for all turbulence models
- It is recommended to use low Reynolds number turbulence models, or high Reynolds number models with a blowing law-of-the wall.
- It is important to fix the transition location to the experimental value.

Classical eddy-viscosity models do not predict the turbulent stresses accurately.

5. EXPLOITATION OF THE QNET-CFD KB

At the end of the QNET-CFD project, the Knowledge Base was adopted by the ERCOFTAC association. In a first phase, copy right permissions associated with the information saved in the Knowledge Base were obtained in order to allow public access to the KB. In a second phase each Application Challenge, Underlying Flow Regime and Best Practice Advice was critically reviewed, and when available new information was added to the information saved. Also the value of the information saved in the KB was improved by adding whenever possible CFD grids. In the near future the QNET-CFD KB will be accessible by paying a license fee.

It is foreseen that the QNET-CFD Knowledge Base will be continuously improved by:

- computing existing flow cases using new physical models, and updating the best practice advice if a better result becomes available;
- 2. insertion of new Underlying Flow Regimes and Industrial Application Challenges together with best practice advice on how to compute these cases.

It is clear that any new result saved in the QNET-CFD Knowledge Base needs to undergo the quality review process to ensure the quality of the Knowledge Base as a whole.

One major source of new flow cases for the QNET-CFD Knowledge Base is EU funded projects. Many such projects have generated in the past a wealth of knowledge on using and applying CFD. But all this knowledge has remained within the project team and has never been properly exploited and made available to the CFD community. In general the knowledge is lost a few years after the project is finished and as such it can be considered a waste of effort and money. Examples mentioned in this paper are the FLOWNET database and the High Speed Flow Field data base which have disappeared from the web.

Saving flow cases from EU (and also ESA) funded projects in the QNET-CFD Knowledge Base will:

- preserve the knowledge generated during an EU funded project and as such will increase the value of the project;
- guarantee that experimental and CFD data generated during an EU funded projects meets a certain quality standard, which will increase the level of trust in the application of CFD;
- facilitate the identification of gaps in knowledge both on the modeling and experimental level, and this information can be used to prepare future EU funded projects.

The QNET-CFD Knowledge Base is a unique tool having an enormous potential to become a standard for preserving knowledge generated in EU and ESA funded projects on the application of CFD. This knowledge can then be used to increase the level of confidence and trust in using CFD in industry.

6. CONCLUSIONS

Since the early 1990's, systematic validation activities have been carried out, partly funded by EC projects. When considering the aeronautical sector, several test cases were initially defined in AGARD Working groups, and were then used in several EC projects, because the combination of good quality experimental data and challenging flow physics made them interesting for validation. Examples of test cases often used for aeronautics are the A-Airfoil, RAE 2822 Airfoil, NLR 7301 multi element airfoil, ONERA M6 wing, RAE M2155 wing, DLR F4 Wing Body, DLR F6 Wing-Body-Pylon-Nacelle, AS28G Wing-Body-Pylon-Nacelle, Delery bump. CFD data for these test cases are distributed among different data bases, for example the RAE M2155 was used in AVTAC and in FLOWNET and is an Application Challenge in QNET-CFD.

It is desirable to set up a central data base in which high quality CFD and experimental data are stored for use by the CFD community. This data base system should contain mechanisms for inserting new CFD data obtained using, for example, a new turbulence model. Comparison plots should be automatically updated to assess the performance of the new model. Quality control review procedures should be implemented to make sure that CFD simulations are made with the necessary care (following for example procedures described in the ERCOFTAC Best Practice Guide).

The QNET-CFD Knowledge Base is a first step in the direction of such a central data base system. It is a unique tool for preserving and thus increasing the value of knowledge generated in EU and ESA funded projects.

REFERENCES

- Perrier P. Verification and validation: industrial issues, VKI Lecture Series on Verification and Validation of CFD, June 2000.
- [2] Johnson F.T., Tinoco E. N., Yu N. J. (2003). Thirty Years of Development and Application of CFD at Boeing Commercial Airplanes, AIAA Paper 2003-3439, 2003.
- [3] 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.
- [4] 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.
- [5] Guide for the Verification and Validation of Computational Fluid Dynamics Simulations,

AIAA G-077-1998.

- [6] ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD - Best Practice Guidelines. ERCOFTAC, 2000.
- [7] 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.
- [8] Dervieux A., Braza, M., Dussauge J.P. Computation and comparison of Efficient Turbulence Models for Aeronautics: European research project ETMA. Notes on Numerical Eluid Mechanics Vol 65.
- Notes on Numerical Fluid Mechanics, Vol. 65, Vieweg, 1998
- [9] 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 1007
- Vieweg, 1997. [10] Gould A. et al. The AVTAC project - A review of European Aerospace CFD.
 - Proceedings of ECCOMAS 2000.
- [11] Mellen C.P., Fröhlich J. and Rodi W. Lessons from the European LESFOIL project on LES of flow around an airfoil, AIAA Paper 2002-0111, 2002.
- [12] Davidson L., Cokljat D., Fröhlich J., Leschziner M., Mellen C. and Rodi W.
 LESFOIL: Large Eddy Simulation of flow around a high lift airfoil.
 Notes on Numerical Fluid Mechanics, Vol. 83, Springer 2003.
- 13] Flomania Flow Physics Modelling An Integrated Approach.
- http://cfd.mace.manchester.ac.uk/flomania 14] Desider – Detached Eddy Simulation for Industrial Aerodynamics
- http://cfd.mace.manchester.ac.uk/desider [15] Abgrall R., Desideri J.-A., Glowinski R., Mallet M.,
- Periaux J. (Eds.) Hypersonic flows for re-entry problems, Vols 1-3, Proc of INRIA-GAMNI/SMAI Workshop Part I and II, Antibes, April 1990-1, Springer, Berlin, 1992.
- [16] FLOWNET: Flow Library on the Web Network http://www-sop.inria.fr/sinus/flownet/index.php3
 INRIA Sophia Antipolis and Dassault Aviation, 1998.
- [17] Grasso F., Periaux J., Deconinck H. (Lecture Series Directors), Verification and validation in CFD. VKI-Lectures Series, June 2000.
- [18] Marini M. and Paoli R. Verification and validation of CFD through the use of a data base.
 Proceedings ECCOMAS CFD Conference 2001.
- [19] Gilham S, Scaperdas, A., The QNET-CFD Knowledge Base.

The QNET-CFD Newsletter, Vol. 2, No. 4, 2004.