RECENT EFFORTS FOR CREDIBLE CFD SIMULATIONS IN CHINA

BAI Wen, LI Li, LIANG Yihua
Aeronautics Computing Technique Research Institute (ACTRI)
China Aviation Industry Corporation I (AVIC I)
Mail: P.O. Box 90, Xi'an 710068, P.R. China
Phone: 86-29-88151165; Fax: 86-29-88151001
E-mail: wenbai@263.net

Keywords: *CFD; Credibility; Uncertainty; Verification; Validation.*

Abstract

Following similar activities in the West such as ECARP and AIAA Drag Prediction Workshops, a workshop on credible Computational Fluid Dynamics (CFD) simulations is organized by CARDC (China Aerodynamics Research and Development Center) and ACTRI. The first effort to hold a workshop in 2003 was abandoned, but resulted in a great promotion of relative researches and a special edition of papers. The 17 papers come from 9 organizations. The selected test cases include DLR-F4 wing-body combination, NLR-7301 two elements airfoil at high lift, and blunt cone at hypersonic speed. The second effort is started in September 2004, and the workshop was held in June 2005. This time the focus is on the capability to calculate high angle of attack flow problem. The full length paper summarizes the progress and the gains. Also reported are the recent computational results on DLR-F6 Wing-/Body-Pylon-Nacelle (WBPN) combination by ACTRI and CARDC, which is the objective model of AIAA Drag Prediction Workshop II in 2003.

Another major effort is an ongoing project at ACTRI to establish a software platform for studying the credibility of CFD solvers and performing credible CFD simulations. The platform, named WiseCFD, is designed to implement a seamless CFD process and to circumvent tedious repeating manual operations. The main components of WiseCFD include test cases, candidate solvers, graphic user interfaces to input computational parameters and monitor the solving process, parameter sensitivity analysis tools, data extract, analysis and comparison tools, and an applied job scheduler. The test cases are organized as three distinct classes, which are verification cases, validation cases and application cases respectively. Under each specific test case, the so-called study cases can be established to perform various analysis tasks. The platform is delivered with built-in test cases. User can also establish his own test cases through interactive operations. Currently five solvers are integrated into the platform as reference computing engines, including a multi-block structured grid RANS solver and a hybrid grid RANS solver developed by ACTRI, two multi-block structured grid RANS solvers by CARDC, and a single block structured grid RANS solver by National Laboratory of CFD. Solvers with pre-defined input and output file formats can be inserted into the platform directly to share the data resource and to be tested. Statistic analysis tools, grid convergence analysis tools, flow topology analysis tools, 2-D curve drawing and 3-D flow field visualization tools are all together integrated into the platform. Critical parameters can be varied in different ways to check the sensitivity of the computational results. In order to deal with the vast computational tasks, a job scheduler for distributed computational environment is developed and integrated into the platform.

Future work on WiseCFD is proposed, and also envisioned is how WiseCFD and the European QNET-CFD Knowledge Base can benefit mutually.

1 Introduction

In the last decades, Computational Fluid Dynamics (CFD) has undergone a strong development and has become a powerful tool both for the analysis and understanding of fluid dynamics phenomena, and for the design and optimization of aerodynamic performance of aircrafts or aerospace vehicles. This progress has been made possible with the advent, in the meantime, of faster and faster supercomputers with increasing memory capabilities, and with the rapid progress of modern numerical computing technology. However, the limitations of CFD need to be addressed due to the lack of knowledge of physical phenomena and appropriate physical modeling as well as limited experimental information. In addition, CFD is strongly affected by the numerical methodology employed (e.g., geometrical modeling, spatial discretization scheme, time accuracy computing, etc.) and computer resources. Therefore, the question of the credibility of the numerical solution is naturally proposed. In other words, to make the numerical simulations practicable, a crux stage is to perform a credibility analysis for the simulation, whose primary target is to assess and eliminate uncertainty in computation as possible. In terms of the authoritative guidance given by AIAA in 1998, the basic activities are CFD verification and validation (V&V) [1].

In developed countries, the activities related to CFD credibility analysis started early, and were paid extensive attentions, especially they had laid many special projects and workshops with the intent of developing large databases of numerical solutions and experimental data of basic reference test cases to be used for CFD V&V. The ECARP (European Computational Aerodynamics Research Project) involved 39 European partners for validation of CFD codes and assessment of turbulence models in Europe during the period 1993 to 1995 with a special research edition for close in 1997 [2]. At the end of 1998, in order to provide the scientific and industrial communities with a validation and experimental methodologies ranging from subsonic, transonic to supersonic and hypersonic regimes, the European Community Commission began to support the set-up of the Thematic Network FLOWNET (Flow Library on the Web network). The ultimate goal of this thematic network is to stimulate collaboration between industrial and research partners in order to evaluate continuously the quality of the simulations and the performance of CFD software, the scope being to improve complex design in aeronautical and aerospace industry [3]. In 2000, 44 participating organizations across Europe were brought together for a four-year project QNET-CFD, EU Network on Quality and Trust in the Industrial Application of Computational Fluid Dynamics [4]. Its main objective was to improve the level of trust that can be placed in industrial CFD calculations by assembling, structuring and collating existing knowledge encapsulating the performance of models underlying the current generation of CFD codes. During the course for the project, QNET-CFD sponsored series of workshops in 2001, 2002, 2003, respectively. Following the end of the EU Network in July 2004, its Knowledge Base passed to the control of a committee of the European Research Community of Flow, Turbulence and Combustion (ERCOFTAC) with the remit of enhancing and expanding the Knowledge Base and bringing it online for the benefit of applied Fluid Dynamicists and CFD users worldwide.

AIAA Applied Aerodynamics Technical Committee (APATC) had sponsored a series of Drag Prediction Workshops since 2001. The objectives of these workshops were (1) to assess state-of-art computational methods as practical aerodynamics tools for aircraft forces and moment prediction, (2) to impartially evaluate the effectiveness of existing computer codes and modeling techniques, and (3) to identify areas needing additional research and development. The first Drag Prediction Workshop (AIAA-DPW-I) was held in June of 2001. Its challenge was to compute the lift, drag and pitching moment for the DLR-F4 subsonic wing-body transport configuration [5]. The second Drag Prediction Workshop (AIAA-DPW-II) was held in June of 2003. At this time the emphasis was on the determination of installed pylon-nacelle drag increments and on grid refinement studies with the hope of seeing reduced code-to-code scatter. The geometries used were DLR-F6 wing-body (WB) and wing-body-pylon-nacelle (WBPN) combinations [6]. The third Drag Prediction Workshop (AIAA-DPW-III) is planned to be held in June 2006. The focus of the workshop will be on "blind" drag prediction accuracy; a priori experimental data will not be available for comparison. In addition to the DLR-F6 wing-body with or without FX2B faring transport models, two wing-alone models, DPW-W1 and DPW-W2, are also included to encourage academic participation and allow more exhaustive grid convergence studies [7].

In March 2004, in order to more broadly assess the current capabilities of different computational methodologies, the CFD Validation of Synthetic Jet and Turbulence Separation Control (CFDVAL2004) workshop was held in Williamsburg, Virginia [8]. Three different test cases, all of which were carried out experimentally by NASA Langley Research Center, exercised various aspects related to the flow physics of separation control. The workshop was structured to the series of Refined Turbulence Modeling workshops sponsored by the ERCOFTAC. 10th of which was held in France in October 2002.

Comparatively, similar activities are rare in the past in China. In recent years the status is gradually changed. The theory and application studies for CFD credibility are paid much more attentions. Since 2000, CFD credibility analysis and experimental verification for aerodynamic numerical software had been determined a national key project among aerodynamic advanced research projects. From then on many efforts for credible CFD simulations were made. In this paper our goal is to outline some outstanding efforts among them. The first important effort is The National CFD Uniform Test Cases Computation Workshop, following similar activities in the West such as ECARP and AIAA Drag Prediction Workshops. Such series of workshops were jointly sponsored by CARDC and ACTRI with intent to assess the state-of-art computational fluid dynamics (CFD) of China and to identify research areas in the near future. Another is WiseCFD software platform for aerodynamic credibility analysis, which is one of prominent outputs of an ongoing project at ACTRI [9]. It is designed to implement a seamless CFD process and to circumvent tedious repeating manual operations. Besides, efforts related to international collaborations for CFD credibility analysis are also reported; especially how WiseCFD and other validation databases, e.g. QNET-CFD, can benefit mutually is envisioned.

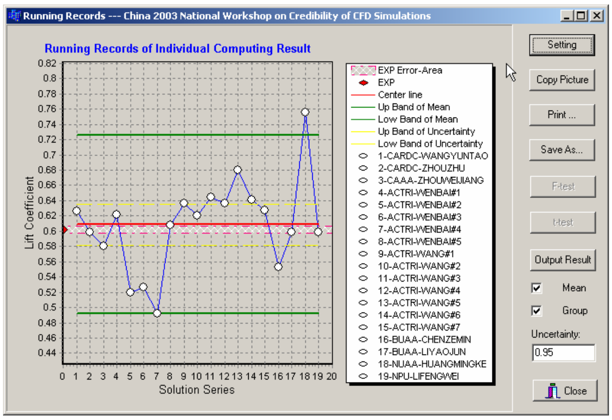
2 China National CFD V&V Workshops

The mainstream collective CFD V&V activities in China are two successive workshops organized by CARDC and ACTRI in 2003 and 2005, respectively. Both of these workshops are to assess the state-of-art of CFD in China, and to identify areas needing additional research and development to promote prediction accuracy.

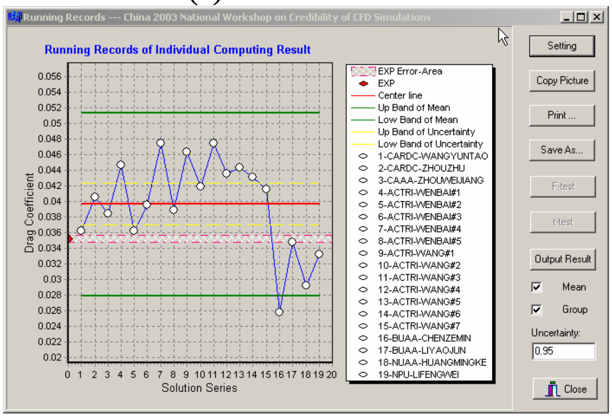
The first effort to hold a workshop in 2003 was abandoned for some reasons, but resulted in a great promotion of relative researches and a special edition of papers [10]. The special edition was composed of 17 papers from 9 organizations. The selected test cases include the DLR-F4 wing-body combination, NLR-7301 two elements airfoil at high lift, and blunt cone at hypersonic speed. The emphasis is the prediction accuracy for drag and heat flux. The participators are required to perform their computations using the geometry and grid files provided by head organizers, and are also encouraged to generate their own grid for simulation. The geometry of DLR-F4 configuration is the same as that of AIAA DPW-I [5]. Its challenge for this workshop is to compute the lift, drag and pressure distribution at typical wing section 0.185, 0.238, 0.331, 0.409, 0.512, 0.636, and 0.844 along the wing for a specific condition ($\mathrm{Re} = 3.\times 10^6$):

$$M_\infty = 0.75, \alpha = 0.93^0 (C_L = 0.6).$$

The dominant computational grid is provided by CARDC. Some of participators also generate their own grid. There are 9 papers reporting 19 computational results. Only 2 results are obtained using Euler computation. The rest



(a) Lift coefficient



(b) Drag coefficient

Fig. 1. Statistical Analysis of Computational Results for DLR-F4 Wing-Body Configuration from the 1st National CFD Uniform Test Cases Computation Workshop

employ Reynolds-Averaged Navier-Stokes (RANS) technique with proper turbulent models. The typical turbulent models include Spalart-Allmaras (SA) and Baldwin-Lomax (BL). A statistical analysis for these results is shown in Fig. 1. It is observed that results from the workshop are quit scattered. Most of the results have over-predicted the lift and drag. For the case of NLR-7301 high-lift airfoil, two geometries are considered depending upon the gap size between the wing and the single-slotted trailing edge flag, respectively, 1.3% and 2.6%, being as a percentage value of the main wing chord. The computational conditions are:

(1) 1.3% flap-gap

$$M_\infty = 0.185, \alpha = 6^0, \mathrm{Re} = 2.51\times10^6$$

(2) 2.6% flap-gap

$$M_\infty = 0.185, \alpha = 13.1^0, \mathrm{Re} = 2.51\times10^6$$

The dominant grids used in this workshop for such configurations are multi-block structured patched ones, which can be also found from ECARP and FLOWNET libraries. Their topology and cells number are completely identical with 9 blocks and 144832 cells. In Fig. 2, a typical grid topology is shown. Results required to be submitted include:

(1) lift coefficient, drag coefficient and their convergent histories
(2) pressure coefficient distributions of the main wing and the flap
(3) skin friction coefficient distributions of the main wing and the flap
(4) velocity profiles of the boundary layer at typical locations.

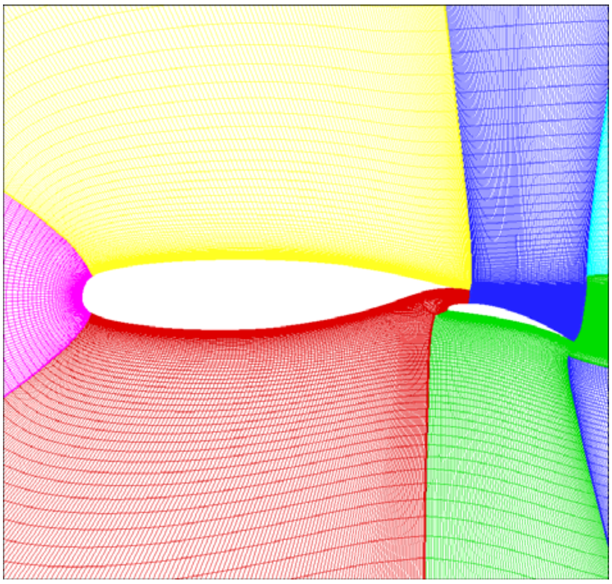


Fig. 2. NLR-7301 Two Elements High Lift Airfoil and Its Typical Grid Topology from ECARP and FLOWNET(2.6% flap-gap)

There are 5 contributors for the test case. Among them, besides results of RANS plus turbulent models, some results of Large Eddy Simulation (LES) are also reported. Fig. 3 shows a typical pressure coefficient distribution computational result by ACTRI employing RANS with SA turbulent model. For this challenge, in order to promote the prediction accuracy for large flow separation near the wall, a wider and more elaborate variety of turbulent models (SA, BL, $k-\omega$, Jones-Launder, JB, and SGS, etc.) have been considered in these contributions. Same scatters are found for this test case. The geometry and its dominant computational grid of the third test case, hypersonic blunt cone, are both provided by CARDC. Its challenge is to accurately predict heat flux for following conditions:

$$M_\infty = 10.6, T_\infty = 47.3K, T_{wall} = 294.44K,$$
$$\alpha = 0^0, 20^0, \mathrm{Re} = 3.937\times 10^6.$$

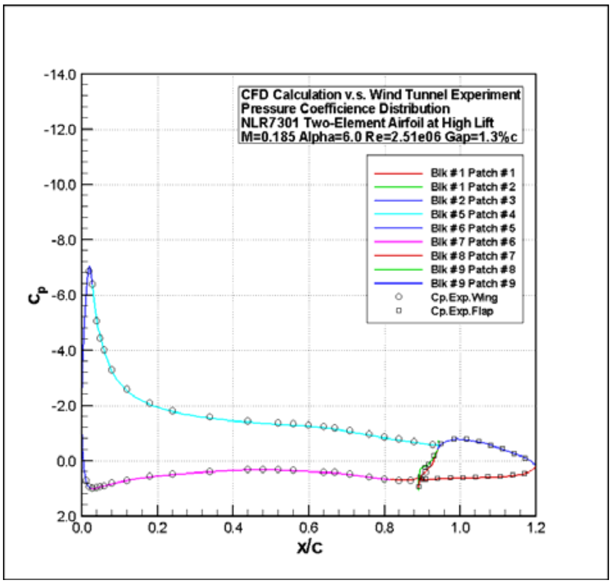
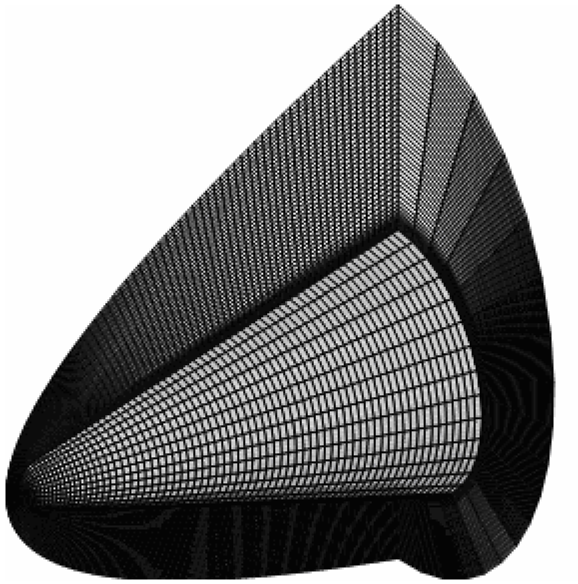


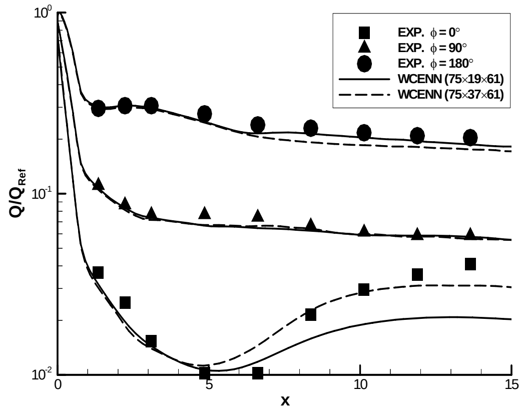
Fig. 3. Pressure Coefficient Distribution by ACTRI for NLR-7301 two elements airfoil (1.3% flap-gap)

There are 6 contributors having reported their results on the workshop. Fig. 4 shows a typical result for this test case by CARDC. From these contributions, one important observation is that a higher order scheme is obviously superior for heart flux prediction based on the same computational grid.

The second national effort for credible CFD simulations started from September 2004. The formal workshop was held in Inner Mongolia, June 2005. The emphasis of this workshop is the simulation of high attack angle problem, where the CT-1 standard model provided by CARDC [11] is computed for six main force coefficients (i.e., axial force coefficient, normal force coefficient, side force coefficient, X force coefficient, Y force coefficient and Z force coefficient), lift coefficient and drag coefficient. The computations were performed at the following flow conditions: $M_\infty = 0.5$, $\mathrm{Re} = 1.4\times 10^6$, angles of attack between -5^0 and 105^0 and sideslip angles $\beta = 0^0, 5^0, 15^0$.



(a) Computational grid and topology

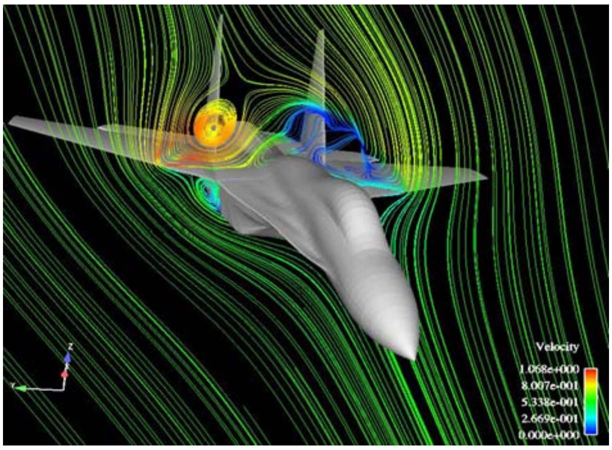


(b) Heat flux

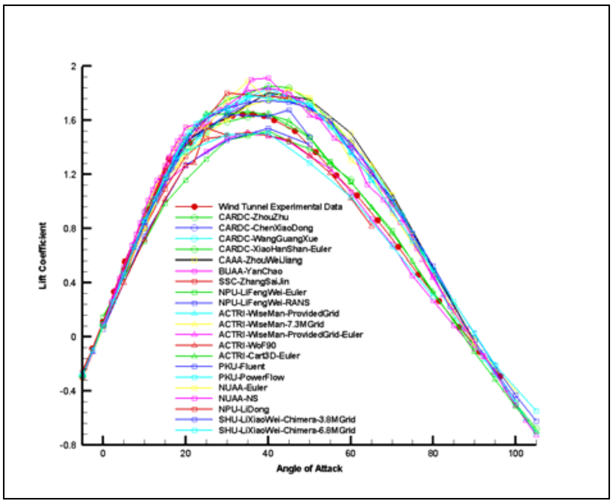
Fig. 4. Typical Computational Results for Hypersonic Blunt Cone Configuration from the 1st National CFD V&V Workshop

There are 34 attendees from 12 workgroups of 9 organizations. Most results are obtained by RANS technique plus proper turbulent models. The turbulent models involved include SA, BL, $k-\omega$ and SST. Part results are from Euler computation. Besides, few results from Detached Eddy Simulation (DES, hybrids of LES and RANS) and Lattice

Boltzmann Method (LBM) are reported. Fig. 5 shows a typical result from the workshop, where the lift coefficients from different contributors are given. Results of this workshop exhibit the state of art of CFD researches in China.



(a) CT-1 Standard Aircraft Model at High AoA



(b) lift coefficients

Fig. 5. A Typical Computational Result for High AoA model from the 2^{nd} National CFD Uniform Test Cases Computation Workshop

It is shown that for a high attack angle problem great progresses have been made in the recent years. Briefly speaking, for CT-1 model, we had concluded that: (1) the computational aerodynamic characteristic (e.g., lift, drag and pitching moment) under medium attack angle is of good agreement with experimental data; (2) the computational angle to occur unsymmetrical side force agrees with experimental data well, while the amplitude of the side force is closed to from experiments; and (3) when the sideslip angle doesn't equal to zero, the computational angle making the yawing moment in reverse agrees with experimental data by and large. Nevertheless, some limits are also observed. They are: (1) the attack angle corresponding to the maximal lift coefficient from computations is offset to that from experiments; (2) there exist obvious gaps between computations and experiments for results at attack angles between 40^0 and 60^0; and (3) prediction accuracy of moment characteristic at high attack angle is not enough.

3 Software Platform for Credible CFD Simulations: WiseCFD

From 2000, ACTRI began to creatively develop a software platform for studying the credibility of CFD solvers and for performing a credible CFD simulation. They named their software as WiseCFD. The main components of WiseCFD include test cases, candidate solvers, parameter input Graphic User Interface (GUI), process monitor, applied job scheduler, parameter sensitivity analysis tools, and other tools for data extract, analysis and comparison.

Currently, the version of WiseCFD is the version 2.0. In Fig. 6, its main GUI is given, which primarily consist of four view areas: toolbox area at the top, case catalog tree area at the left, operating panel area in the middle, and information area at the bottom. Most of functions within WiseCFD v2.0 can be found from these areas. The main functions are (1) to manage CFD solvers, (2) to manage computing jobs, (3) to have data backup; (4) to customize server and outer software; (5) to use auxiliary toolbox; and (6) to use help system. In order to implement these functions, WiseCFD v2.0 tackled several key techniques, e.g. data management technique for CFD credibility analysis, data analysis and comparison technique, and distributed computing technique for sensitivity of parameters. Via these techniques it completes a streamline aerodynamic process of case selection, computational parameter definition, computing job management, distributed computing, and data analysis and comparison, which greatly enhances computing efficiency and efficiently reduces uncertainty in simulation. The main features of WiseCFD v2.0 can be briefly summarized as follows.

(1) With an open extensible software integration framework and standard data exchange interface, it supports the plug and play (PnP) software integration. Origin reference computing engine integrated in WiseCFD v2.0 include a multi block structured grid RANS solver WiseMan, a unstructured RANS solver WoF , both developed by ACTRI, and a single block structured grid hypersonic solver NLCFD-NS3D by National Laboratory of CFD. TRIP v2.0 and HSSPF hypersonic solver developed by CARDC are integrated into current platform in the simple way of PnP.

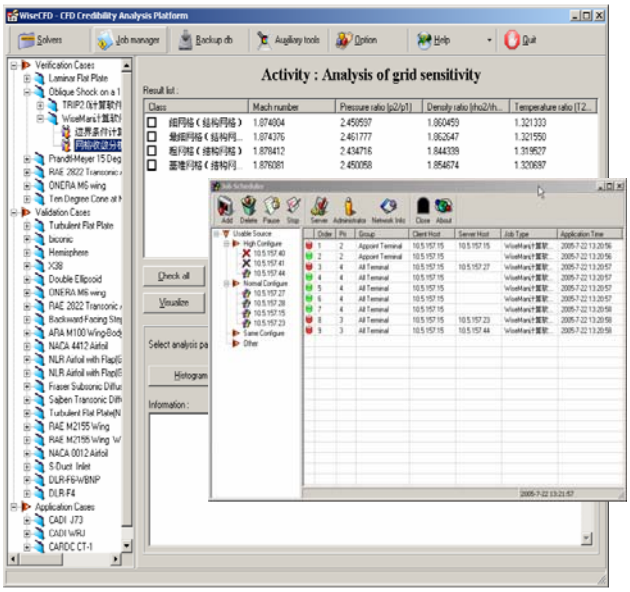


Fig. 6. Graphic User Interface (GUI) of Software Platform WiseCFD

(2) With the power of distributed management and that of real time monitor for batched jobs, it can efficiently utilize available computing resources at most.

(3) It has integrated large number of credible aerodynamic experimental data and CFD data, which results in a database/knowledge base for CFD V&V. WiseCFD v2.0 organizes its aerodynamic data in a form of a tree. Such a hierarchy can be clearly noticed in view area of case catalog tree. Its first level is case type, which is separated into verification case, validation case and application challenge case. The second level is practical test cases. The third is computing engine of test cases. The last level is study case. Here, the study case means a research using the same solver for the same test case with different computing parameters. Currently, it has collected 6 verification cases, 20 validation cases and 3 application challenge cases.

(4) It provides abundant means for data analysis and comparisons. It not only developed and integrated several auxiliary tools such as the aerodynamic data computing software WiseADC, 2D engineering curve drawing software WiseECG, special statistical analysis tool WiseSAT and grid convergence analysis tool WiseGCI, but also customized manifold pre and post processing software by way of script. The customized pre-processing software includes ICEM CFD, Gridgen and Gambit. The post processing software includes Tecplot, Ensight and VisIt. In virtue of the auxiliary tools and outer pre and post processing software, data analysis and comparisons are easily implemented in WiseCFD v2.0.

(5) It has a perfect help system. Within WiseCFD v2.0, there are not only help documents related to operations for the platform, but more important, there is a collection of literatures for CFD credibility analysis. The collection has collected and collated many important literatures related to CFD credibility analysis, which almost cover each level of activities of CFD verification, validation, prediction and calibration, and almost represent the state of art of the related activities in the world.

4 International collaborations

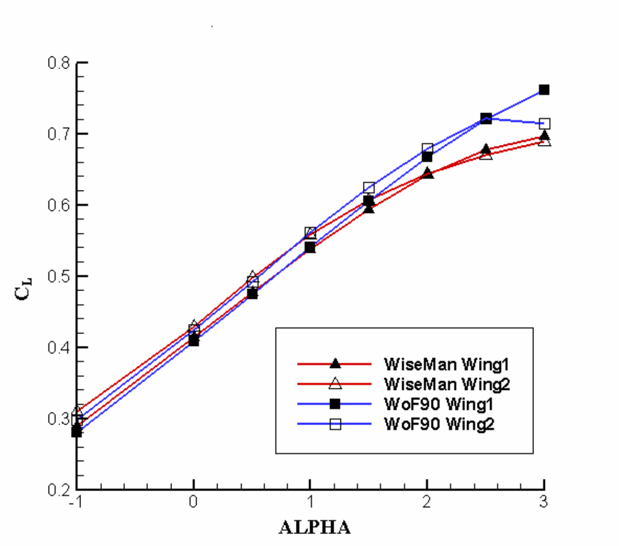
In recent years some outstanding efforts in the field of credible CFD simulations for international collaboration have been made. During October 19th and 22nd in 2005, the East West High Speed Flow Field Conference

(EWHSFF2005) was successfully held in Beijing, China. EWHSFF conference is a sequence of international actives. It was originated as US-Europe Conference and Database Workshops on High Speed Flow Field. The first two conferences were held in Houston, the United States in 1995, and in Naples, Italy in 1997, respectively. Due to their success, the conference committee decided to transform and extend to a new conference entitled: “West East High Speed Flow Field Conference” (WEHSFF), and the first one was held in Kyoto, Japan, in November 1998. The next one took place Marseilles, France, in April 2002, in connection with the European FLOWNET database workshop. This time it was co-organized by Chinese Academic Establishment (CAE) and Beihang University (former Beijing University of Aeronautics and Astronautics, BUAA). The conference was supported by Chinese Government and Chinese Aerospace industries. Its focus was on both experimental and computational progresses on research of high speed flows (from high subsonic to hypersonic regimes) in aerospace applications; especially validation database summary was an important part in the format of the conference. It was believed that the EWHSFF2005 conference was a direct communication of validation database in China with those in the West (e.g., FLOWNET [3], QNET-CFD [4] and PROMUVAL [12]). There were hundreds of participators from a wide variety of countries such as Europe, United States, Russia, Japan and China. The papers from the congress were published in a special proceeding [13].

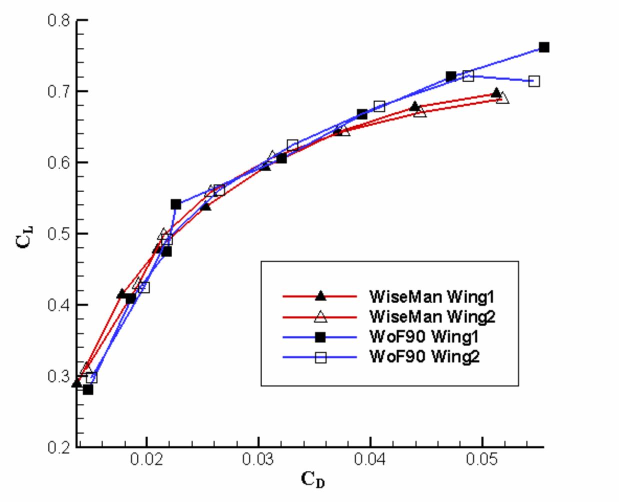
In the early of 2005, CIMNE (Barcelona, Spain) and ACTRI (Xi’an, China) allied to initiate the set-up of an EU-China Network on aerodynamics. The resulted AEROCHINA proposal nowadays had been successfully approved as a Specific Support Action (SSA) project of the 6th Framework Programme (FP6) of European Commission [14]. Its full title reads PROMOTING SCIENTIFIC COOPERATION BETWEEN EUROPE AND CHINA IN THE FIELD OF MULTIPHYSICS MODELING, SIMULATION, EXPERIMENTATION AND DESIGN METHODS IN AERONAUTICS. It is also funded by the Ministry of Science and Technology (MOST) of China. The partners are representatives from aeronautical industry, university and research organizations in Europe (12 partners) and China (12 partners, with the after join of China Academy of Aerospace Aerodynamics).

The aim of the AEROCHINA SSA is to foster the cooperation between a number of industry, university and research organizations in the aeronautics sector in Europe and China in the field of mathematical modeling, computer simulation and code validation, experimental testing and design methods for the solution of multiphysics problems of interest to the aeronautic sector. The spectrum physical disciplines (coupled or not) considered in AEROCHINA which are of interest of European and Chinese partners are Aerodynamics, Structures & Materials, Fluid Dynamics, Aeroacoustics and Aero Elasticity. The general strategic objectives of the project are three folds: (1) to identify areas of mutual RTD interest and the clarification of the skills, experiences and capabilities of the Chinese partners in the relevant technological areas of multiphysics analysis and design, (2) to develop concepts of collaboration in those areas between the European and Chinese partners in order to ensure a win-win situation, and (3) prepare specific RTD activities that are mature for joint proposals for FP7. These project objectives are in accordance in the conclusions of the EU-China workshop in Aeronautics held in Beijing on April 15 2005. Among others, one of the conclusions of the workshop was that “a SSA network could prove to be a very useful tool to reinforce the RTD links between China and Europe”.

ACTRI had taken an active part in the AIAA-DPW-III efforts. We had submitted our computational results for both of the wing-alone and DLR-F6 models to the workshop committee. Two of our colleagues planned to be there in San Francisco to join the workshop and had been scheduled to give a presentation. But it was a real pity that they did not get their visas due to a little bit late submission of the application forms.



(a) Lift Curve



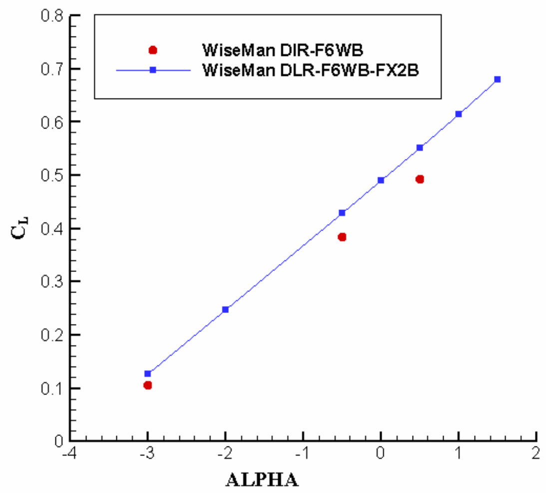
(b) Drag Polar

Fig. 7. Lift Curve and Drag Polar by ACTRI for AIAA-DPW-III Wing1 and Wing2 Models

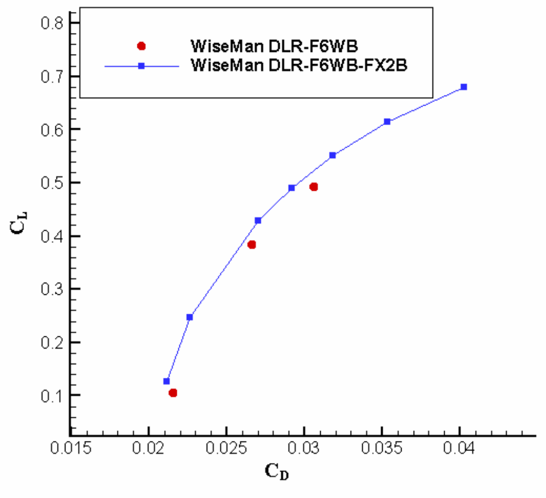
Since the experimental data is not revealed until the completion of this paper, we would like to present some typical results of ACTRI for archiving objective. The lift curves and drag polar calculated are shown in Fig. 7 and Fig. 8.

5 Future work and conclusions

In the present paper we have outlined several outstanding efforts for CFD credible simulations in the recent years, whose basic activities are verification and validation due to international authority. The aspects of verification and validation must always be addressed with an emphasis on the quantification of the uncertainties due to the model assumptions (either physical or geometrical), and to the numerical and experimental approximations. There are various roadmaps for V&V. The regular one is workshop.



(a) Lift Curve



(b) Drag polar

Fig. 8. Lift curve and drag polar by ACTRI for DLR-F6 wing-body with and without FX2B faring transport models from AIAA-DPW-III

Following similar activities in the West such as ECARP and Drag Prediction Workshops, two successive national workshops for V&V of CFD simulations had been organized by CARDC and ACTRI with intent to assess state-of-arts of computational fluid dynamics of China and to identify areas needing additional research and development.

Development of a software platform for performing credibility analysis is a creative idea. The attempt has been tried in ACTRI to develop the WiseCFD platform. The main components of WiseCFD include test cases, candidate CFD solvers, graphic user interfaces to input computational parameters and monitor the solving process, parameter sensitivity analysis tools, data extract, analysis and comparison tools and an applied job scheduler. Large amount of experimental data and CFD results have been integrated, which results in a useful validation database similar to that of FLOWNET and QNET-CFD. The platform provides abundant means for data analysis and comparisons. Through these tools critical parameters can be varied and studied in different ways. To deal with vast computational tasks, a job scheduler for distributed computational environment is also developed and integrated. Besides, the platform has a perfect help system, which not only provides conventional operations but provides collections of useful best practice advices (BPA) for CFD simulations. Nevertheless, there is still much work for WiseCFD furthermore. One urgent aspect, we suppose, is to develop a new version that runs on the internet such that all related jobs can be completed through the Web. The other aspects include to cross different operation system (OS) platforms and to collect more credible test cases.

The verification and validation of CFD requires more participation. China should be more involved in international activities. Through these collaborations, it is expected that the West and China can benefit mutually. For example, in connection with the QNET-CFD thematic network, WiseCFD can contribute and enjoy available resources.

References

[1] AIAA-G-077-1998**.** *Guide for the verification and validation of computational fluid dynamics simulations*. American Institute of Aeronautics and Astronautics, 1998.

[2] Haase W, Chaput E, Elsholz E, Leschziner M A, and Muller U R. ECARP-European computational aerodynamics research project: validation of CFD codes and assessment of turbulent models. *Notes on Numerical Fluid Mechanics*, Vol. 58, 1997.

[3] Marini M, Paoli R, Grasso F, Periaux J and Desideri J A. Verification and validation in computational fluid dynamics: the FLOWNET database experience. *JSME International Journal Series B*, vol. 45 (1), 2002.

[4] Hirsch C. The QNET-CFD project. *The QNET-CFD Network Newsletter*, vol. 2(4), pp 4-5, 2004.

[5] Levy D W, Zickuhr T, Vassberg J and Agrawal S. Summary of Data from the first AIAA CFD drag prediction workshop. AIAA paper 2002-0841.

[6] Hemsch M J and Morrison J H. Statistical analysis of CFD solutions from 2nd drag prediction workshop. AIAA paper 2004-0556.

[7] http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/

[8] Rumsey C L, Gatski T B, Sellers III W L, Vatsa V N, and Viken S A. Summary of the 2004 CFD validation workshop on synthetic jets and turbulent separation control. AIAA paper 2004-2217.

[9] Liang Y H, Yang Y and Zhu Z, et al.. CFD credibility analysis platform WiseCFD. In: *Proceeding of 12th national computational fluid dynamics conference*, Xian: Chinese Aeronautics Computation Technique Research Institute, pp 775-780, 2004. (in Chinese)

[10] Chen Z B and Bai W. Special edition for credible Aerodynamic computations. *Aeronautical Computer Technique*, vol. 32, 2004. (in Chinese)

[11] Wu, J Q, Fan, Z L, He, Z, and Y J Wang, "Research on the Test Technology at High Angles of Attack in 2.4m Transonic Wind Tunnel", *Experiments and Measurements in Fluid Mechanics*, Vol.18, No. 4, 2004, pp 43~48. [in Chinese]

[12] http://www/cimmne.upc.es/promuval

[13] Lee C H. *Proceeding of EWHSFF2005*. Beijing: International Aviation Group, 2005.

[14] http://www.cimmne.upc.es/aerochina