OVERVIEW OF A VALIDATION STRATEGY FOR MODERN CAD-EMBEDDED CFD CODES

Dr. A.V. Ivanov, T.V. Trebunskikh, V.V. Platonovich (Mentor Graphics Corporation, Russia)

T.V. Trebunskikh, Product Manager

SUMMARY

FloEFD is a new class of CFD analysis software (called Concurrent CFD) that is fully embedded in the mechanical design environment, for all general engineering applications.

There is an extremely big attention paid to of Validation, Verification and testing of FloEFD technology. As an end user of Concurrent CFD software is an engineer, enhanced requirements are set for calculation accuracy, reliability and robustness as well as usability and educability of the software.

This presentation will demonstrate the methodologies used in verifying and validating an immersed boundary CAD-embedded CFD code which involves four levels of testing. For each level FloEFD validation examples are given in the presentation.

Using of validation tests at different stage of software testing, promotion and support is shown.

1: CAD-embedded CFD code FloEFD

Nowadays it is impossible to produce competitive high quality products without involvement a computer-aided engineering (CAE) software. Owing to this fact increasing the role of CFD calculations among various CAE systems is observed last years. The largest efficiency in using CAE (and CFD in particular) systems is achieved under inserting them directly to design process and under utilization of CAE/CFD not only by dedicated departments, but also by mechanical engineers on-site.

It is immersed boundary CAD-embedded CFD code FloEFD representing a new class of CFD analysis software (called Concurrent CFD) that was initially intended for mechanical engineers to use during design processes as an integral part of a product lifecycle management (PLM) concept.

FloEFD has some specific features, namely: complete integration with all major CAD-systems; totally automatic grid generation; automatic prescribing of computation control parameters etc.

FloEFD technology has another significant difference from traditional CFD approach in using a large amount of engineering techniques and methods which are called to user's assistance in obtaining reliable predictions at lower computational and time costs. It must satisfy high requirements on reliability, robustness and accuracy of such an automatic approach and engineering methods. That is why during its development FloEFD has been exposed to the detailed Verification and Validation (V&V) procedure on a host of analytical and benchmark solutions as well as on experimental results available from publications and databases (e.g., Freitas, 1995; Fluid Dynamics Databases, 2002). Some of the results are discussed in the present work in framework of Validation methodology and classifications.

2: Verification and Validation procedures

During the test one should make sure that:

- the same equations solved and the same engineering techniques and models (including subgrid ones) are used in each computational cell;

- geometry of all components retains for all meshes under investigation;

- mesh topology in computational domain are the same;

- the order and type of all equations approximations in each computational cell are the same.

As mentioned above the special feature of FloEFD software is using a large amount of engineering techniques and methods. So, meeting first requirement is not ensured for real engineering problem calculations because different engineering techniques or their combinations can be used as mesh gets finer or coarser. Therefore only relatively simple examples are acceptable for code Verification as separate activity. For the rest of examples it is actually impossible to separate Verification and Validation procedures. That is because grid convergence study will show integral accuracy of the code (or calculations) and not numerical algorithms correctness only.

Four-level classification of validation examples and tests is employed in current practice for V&V-procedure of FloEFD code. The diagram, which graphically visualizes the classification, has a hierarchical structure and looks like an inverted pyramid, with each next level being based on previous one.

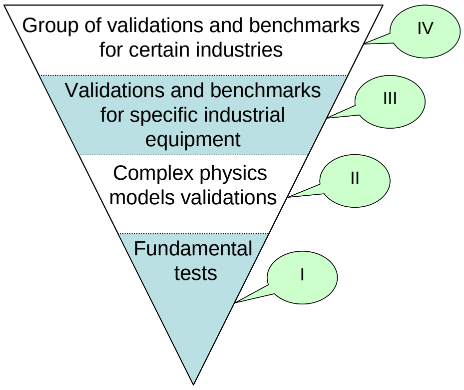

Figure 1: A four-level view of FloEFD code Validation.

The first level, as in Balakin et al (2004), involves the fundamental (academic) tests which are simple enough in sense of geometry (2D as the rule) and problem formulation. (flows and convective heat transfer on a plate, in pipes, in channels and heat sinks etc.)

At the second level are groups of tests that demonstrate how well complicated functions of the software or particular physical models are working (e.g. combustion, conjugate heat transfer, cavitation, condensation, etc.).

The third level comprises applied industrial problems and benchmarks where in addition to the complicated 3D geometry a combination of different strongly coupled physical phenomena takes place. (cyclones, heat exchangers, engines, blowers, pumps, etc).

The last level integrates validation tests and benchmarks from certain industry (aerospace & defense, electronics, HVAC, process industries). Some authors (e.g., Melnik et al, 1995) associate this level with such activity as code Certification or even code Accreditation. A nuance is that code Certification and Accreditation are usually a part of engineering management. These appear to be simple the process of some authority (perhaps legal or regulatory) officially declaring a code to be usable for a specific industry or project (Roache, 1998).

Of course, borders between the levels are frequently fuzzy and the same validation example can be found at different levels depending on the industrial application.

Some of the most typical V&V examples and tests are demonstrated here.

3: Verification/Validation examples

One of fundamental validations is FloEFD prediction of 2D air flow around NLR 7301 airfoil with flap at M = 0.185 and Re = 2.51*106 is considered .

The comparison of FloEFD predicted surface pressure coefficient distributions with experimental ones (Cebeci et al, 1996) at angle of attack of 6° is given in Fig. 2 on the left where satisfactory agreement can be seen. Fig.2 on the right shows comparison of a lift coefficient obtained from the calculation and the experiment (Cebeci et al, 1996) at different angles of attack. Maximum relative prediction error for the lift coefficient is 9%.

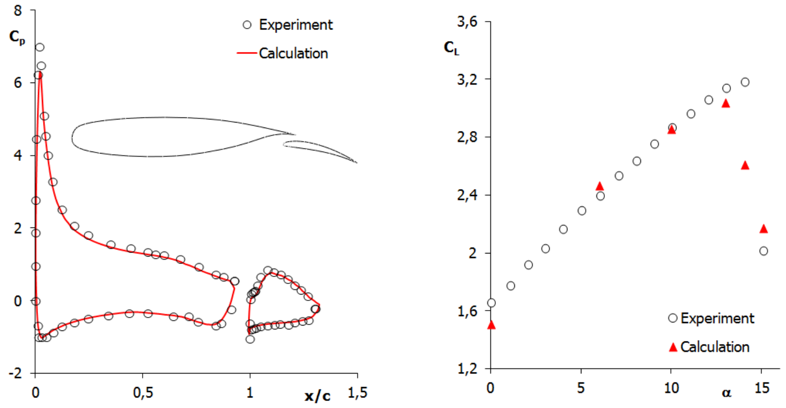

Figure 2: Comparison of computed and measured surface pressure coefficients at angle of attack of 6° (left) and comparison of lift coefficient at different angles of attack (right).

One of the particular functionality or physical models validations is FloEFD validation working process in the bluff body burner for CH4-H2 turbulent combustion. The burner is centred in a co-flowing stream of air and consists of a circular bluff-body with an orifice at its centre for the main fuel. The main fuel jet composition is CH4 and H2 as a fuel and air as an oxidizer. A bulk jet velocity of 118 m/s. The outer flowing air has a 40 m/s velocity and temperature of 300 K.

As can be expected a complex flow pattern forms downstream of the face of the bluff-body with one and possible two recirculation zones (Masri, A.R., Bilger, R.W., 1985). The measured and predicted flow trajectories are presented in Fig. 3 on the left. In Fig. 3 on the right the velocity profiles are compared with measured ones for one cross-section after inlet. The agreement with experimental data is reasonable close. It clearly shows that the main recirculation is correctly predicted by FloEFD code.

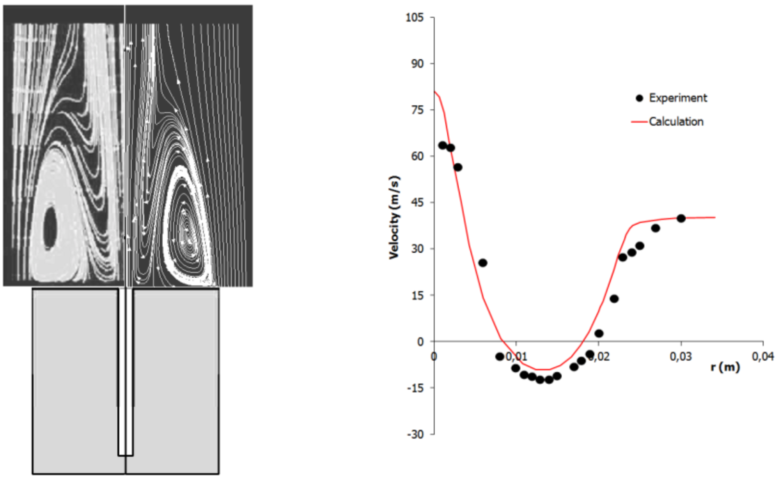

Figure 3: Measured (left part) and predicted (right part) flow trajectories in recirculation zone (left) and axial velocities at Y/D=0.9 (right).

One of the industrial problems and benchmarks is prediction of cooling tower external aerodynamics. This validation example describes the results of FloEFD technology application to analyze the flow around the cooling tower shell. Hyperbolic shape of cooling tower shell is approximated by a short cylindrical throat joined onto two truncated cones. The cooling tower base aperture was treated as sealed. Experimental wind tunnel tests data were taken from Zdravkovich (2003), Cowdrey and O'Neill (1956).

Fig. 4 shows predicted CP distributions at Z/H=0.79 in compared with experiment on the left and distribution of CP with height in rear side of the model on the right. As can be seen that the calculation results demonstrate good agreement with the experiment.

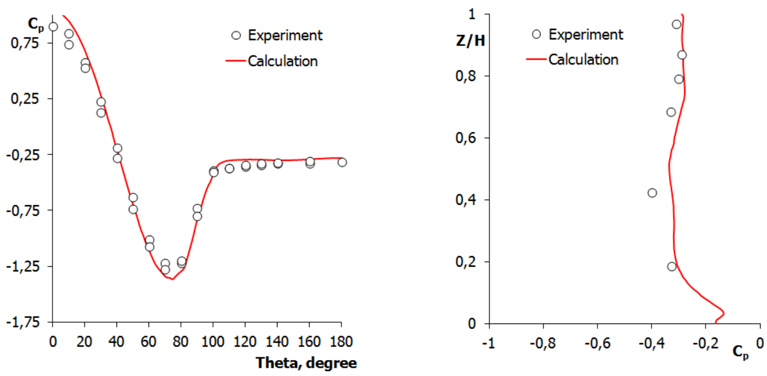

Figure 4: Local CP distributions around cooling tower at elevation Z/H=0.79 (left) and local CP distributions with height in rear side of the cooling tower (theta=180) (right).

As example of complex multiphysics calculations Figs. 5 displays the result of prediction of the visible saturated vapour plume formation. Temperature and relative humidity distributions in downstream transverse cross-sections of the plume fully correspond to the vortex induced scalar parameters fields in turbulent jets.

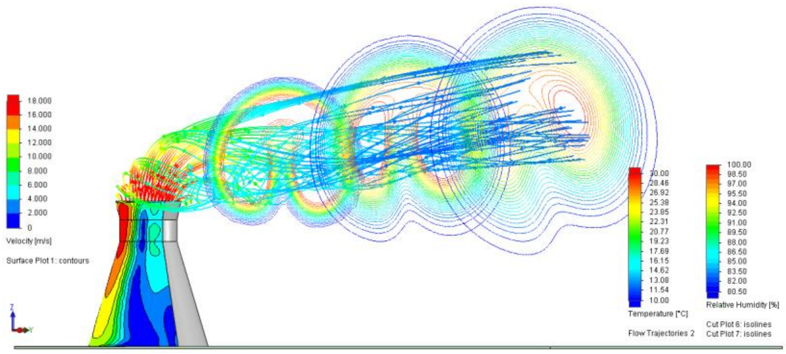

Figure 5: Local CP distributions around cooling tower at elevation Z/H=0.79 (left) and local CP distributions with height in rear side of the cooling tower (theta=180) (right).

Another example of industrial problems and benchmarks is FloEFD simulation of micro-turbine engine. A micro-turbine engine chosen for the study is KJ 66, which is one of the most robust small engines with available design. The specification of KJ 66 can be found in Kamps (2005).

This engine is calculated as a one whole unit (360 degrees without transferred, symmetrical or periodic conditions). Several mesh variants with the total cells' number of ~600000, ~3500000, ~9000000 are examined.

The total pressure and the static temperature of air are 101325 Pa and 288.15 K respectively at the inlet of the engine. At the outlet the same conditions are treated as atmospheric ones. Kerosene is specified as a gas phase. The air fuel ratio is ~65. The calculation is provided in the transient regime.

Fig. 6 on the left displays fluid temperature distribution within the engine. Temperature in the combustion chamber reaches ~2400 K. Fig. 6 on the right shows comparisons of predicted and measured (Kamps, 2005) values of thrust of KJ 66 engine at different modes. It can be seen that experimental and predicted values have a good agreement up to 80000 rpm and at 100000 rpm some discrepancy from experimental data is observed. Also predicted data almost do not depend on cells' number. Thereby mesh of ~600000 cells is enough for definition almost all integral parameters here.

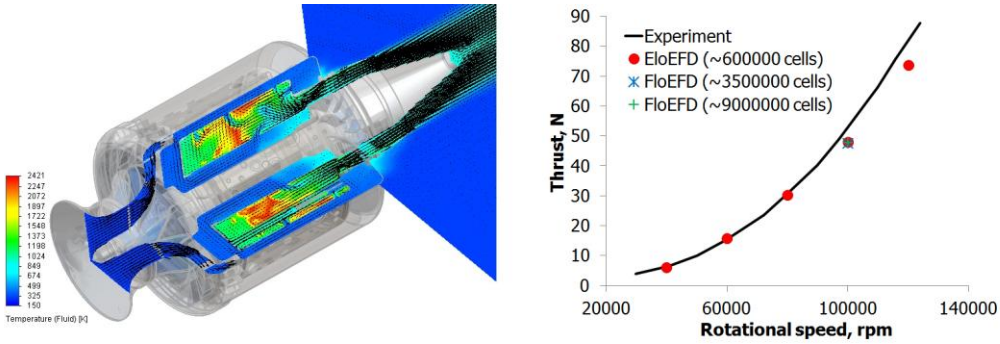

Figure 6: Fluid temperature distribution at two longitudinal sections of the combustion chamber with flow vectors at the normal mode (left) and thrust of KJ 66 engine (right).

One more industrial problems and benchmarks is an external aerodynamic simulation of Tupolev-214 (Tu-214) aircraft. Russian commercial aircraft Tu-214 is a cantilever monoplane of a normal scheme with a low-set swept wing and a tail assembly placed on a fuselage with two turbojet engines mounted on pylons under the wing.

Calculation of this task was provided at following far field conditions: M = 0.6, P_∞ = 101325 Pa and T_∞ = 288.15 K. The angle of attack varies in range from -3° to 18°. Work of the propulsion system was taken into account in these investigations.

Measured (Koshcheev et al, 2009) and predicted lift coefficient and polar are presented in Fig.6. It should be pointed out very good FloEFD prediction of these coefficients at under study Mach number. Also there is good agreement in aerodynamic derivative obtained in FloEFD and experiments and discrepancy is ~1.7%.

Flow trajectories colored by Mach number at M =0.6 and angle of attack 10° is presented in Fig.8.

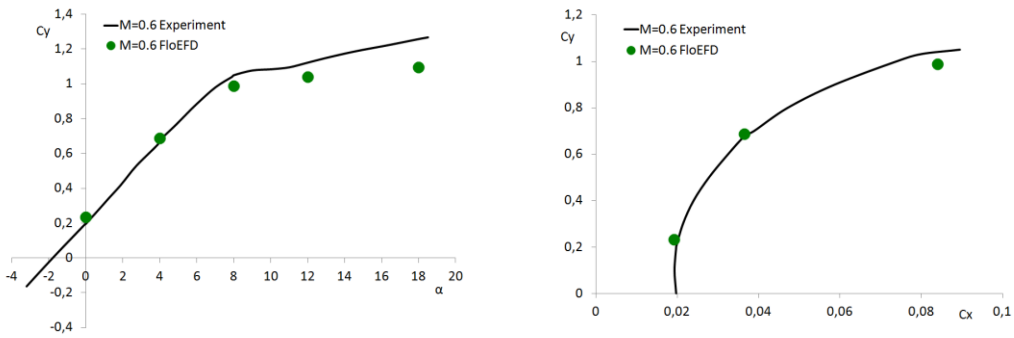

Figure 7: Lift coefficient (left) and polar (right) of Tu-214 (results were provided by PSC "TUPOLEV).

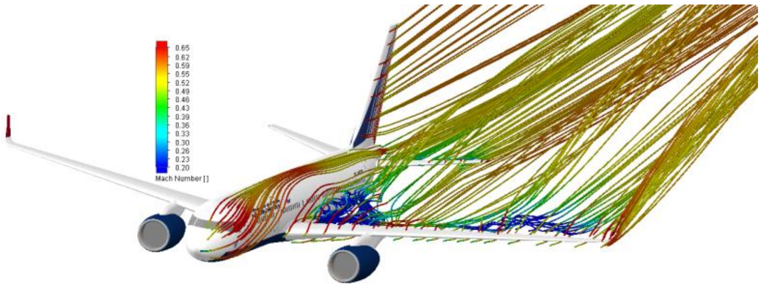

Figure 8: Flow trajectories colored by Mach number at M =0.6 and angle of attack 10° of Tu-214 (results were provided by PSC "TUPOLEV).

4: Conclusions

Presented typical validation examples and tests for each validation level confirm that FloEFD code has been successfully validated on a variety of problems for many years. The experimental data and analytical solutions have been well reproduced numerically via FloEFD simulation with acceptable degree of accuracy. The combination of good performance for relatively coarse mesh, CAD-embedded capability and high level of automation and usability make FloEFD code quite adequate and useful CFD tool for engineering design and analysis.

REFERENCES

Balakin, V., Churbanov, A., Gavriliouk, V., Makarov, M. and Pavlov A. (2004) 'Verification and Validation of EFD.Lab Code for Predicting Heat and Fluid Flow', Proceedings of ICHMT International Symposium on Advances in Computational Heat Transfer, Norway, April 19-24, 2004.

Cebeci, T., Besnard, E., Chen, H.H. (1996) 'Calculation of multielement airfoil flows, including flap wells', AIAA Paper 96-0056, 34th Aerospace Sciences Meeting & Exhibit, Reno, NV, January 15-18, 1996.

Cowdrey, C.F. and O'Neill, P.G.G. (1956) 'Report of Tests on a Model Cooling Tower for CEA: Pressure Measurements at High Reynolds Numbers'. Nat. Phys. Lab., Aero. Rep. 316a .

Fluid Dynamics Databases (2002), ERCOFTAC Bulletin, No. 52.

Freitas, C.J. (1995) 'Perspective: Selected Benchmarks From Commercial CFD Codes', Trans. ASME, J. Fluids Eng., Vol. 117, No. 2, pp 208-218.

Kamps, T. (2005) Model Jet Engines, UK.

Koshcheev, A.B., Platonov, A.A., Khabrov, A.V. (2009) ‘Aircraft aerodynamics of of the family Tu-204/214’, Textbook. M.: Poligon-Press, 304 p. (In Russian)

Masri, A.R., Bilger, R.W. (1985) ‘Turbulent diffusion flames of hydrocarbon fuels stabilized on a bluff body’, 20th Int. Symp. on Combustion, The Combustion Institute, Pittsburgh, pp. 319-326.

Melnik, R.E., Siclari, M.J., Marconi, F., Barber, T., Verhoff, A. (1995) ‘An Overview of a Recent Industry Effort at CFD Code Validation’, AIAA Paper 95-2229, 26th AIAA Fluid Dynamics Conference, San Diego, California, June 19-23, 1995.

Roache, P.J. (1998) Verification and Validation in Computational Science and Engineering, Hermosa, Albuquerque, NM.