# Using the modern CAD-embedded CFD code for numerical investigation of NASA Common Research Model aerodynamics

**By Dr. A.V. Ivanov, Dr. G.E. Dumnov, Dr. A.A. Sobachkin, T.V. Trebunskikh**
Mentor Graphics Corporation, Mechanical Analysis Division, Moscow, Russia
E-mail: andrey_ivanov@mentor.com

## Abstract

Since 1999, when the first version of FloWorks was introducing, a number of CAD-embedded and CAD-associated tools have appeared. These tools use different numerical technologies as compared to traditional CFD (Computational Fluid Dynamics) ranging from mesh generation to differencing schemes and wall treatment. One of such tool – 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.

This paper takes a look at the technological basis of FloEFD as well as at the methodology used in verifying and validating this CAD-embedded CFD code. The choice of meshing technology in FloEFD is discussed. It is shown how chosen a Cartesian-based mesh has influence on the way the geometry is handled, in particular solid-fluid and solid-solid interfaces, the wall treatment used to capture boundary layer evolution, and calculation of skin friction and heat fluxes. A four-level classification of validation examples and tests for the verifying and validating procedures of FloEFD is proposed. For each level a small selection of FloEFD validation examples are given in the paper.

Finally, numerical investigation results, demonstrating how the FloEFD rectangular Cartesian-based mesh and boundary layer models can be applied to predict aerodynamics of NASA Common Research Model for different configurations, are presented. Comparisons of calculation results with experimental data show a great potential of FloEFD code as quite adequate and useful CFD tool for aerospace engineering design and analysis.

## 1. Introduction

Nowadays it is impossible to produce competitive, high-quality products without computer-aided engineering (CAE) software. Owing to this fact increasing the role of CFD calculations within CAE is observed last years. The largest efficiency in using CAE systems (and CFD in particular) is achieved by inserting them directly into the product design process by utilization of CAE/CFD by mechanical engineers.

It was FloEFD code that was initially intended for mechanical engineers to use during the design process as an integral part of a product lifecycle management (PLM) concept.

The basic concept behind the design of FloEFD is to automate preparing, performing and visualizing CFD predictions of real applied engineering problems. The FloEFD approach is based on two main principles: a) direct use of native CAD as the source of geometry information; b) combination of full 3D CFD modelling with simpler engineering methods in the cases where the mesh resolution is insufficient for full 3D simulation.

Since the EFD technology has to operate within different CAD systems it was developed as a universal CAD/CFD platform that incorporates a number of technologies: 1) CAD data management; 2) Fast mesh generation; 3) Several CFD solvers; 4) Automatic prescription of solution control parameters; 5) Engineering Modelling Technologies; 6) User-friendly pre- and post-processing.

## 2. Technological Basis

Mesh generators in traditional CFD are usually based on body-fitted algorithms which widely used for solving industrial problems. The detailed reviews of basic types of mesh geometries are presented in several publications (e.g. Filipiak, 1996 and Parry and Tatchell, 2008). As a rule, for complicated geometries unstructured meshes are used, formed by constructing irregularly distributed nodes. Where the geometries being meshed are less complex it is often possible to use structured meshes or their combinations (such meshes may be called partially structured or partially unstructured).

A characteristic of body-fitted meshes is that they are highly sensitivity to the quality of the CAD geometry. The EFD technology is based upon the alternative approach which uses an immersed-body mesh. In this approach the creation of the mesh starts independently from geometry itself and the cells can arbitrarily intersect the boundary between solid and fluid. This makes it possible to use a Cartesian-based mesh which is one of the key elements of Meshing Technology of the CAD/CFD bridge for CAD-embedded CFD.

As a result of using Cartesian-based meshes we have cells which are located fully in solid bodies (solid cells), in the fluid (fluid cells) and cells intersected the immersed boundary (which we term 'partial cells'). In the simplest case the partial cell consists from 2 control volumes (CV): a fluid CV and a solid CV (see Fig. 1).

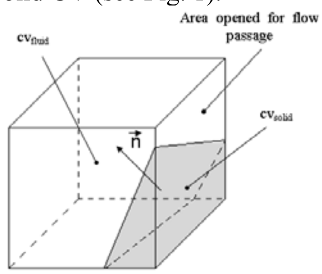


FIGURE 1. Partial cell in the simplest case with 2 control volumes (CV) inside.

Each CV is then fully solid or fully fluid. For each CV all necessary geometrical parameters such as volume and the coordinates of cell centre are calculated. All these data are taken directly from the native CAD model. Such technology allows good resolution of geometry features even in case of relatively coarse meshes (see Fig. 2).

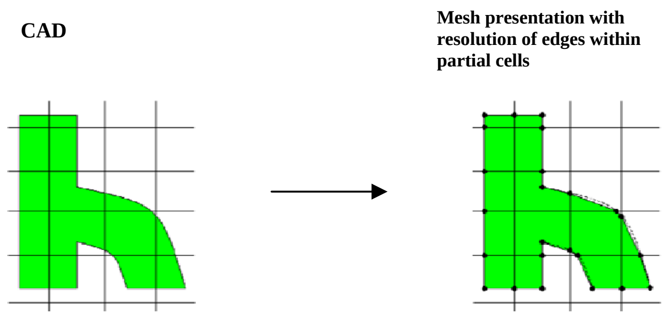


FIGURE 2. Mesh representation of CAD geometry with resolution of solid edges within partial cells.

The mesh can be refined (by splitting each cuboid into 8 geometrically-similar cuboids) using various adaptation criteria to solid body and to flow field structure. These refinement procedures are essential to resolve features of the CAD geometry like surfaces with small curvature, small features, narrow channels, etc. Moreover, the use of such mesh generation technology allows the implementation of efficient and robust automatic tools for meshing.

In general the Cartesian mesh approach used in FloEFD allows performing conjugate multiphysics calculations, using one computation mesh having fluid cells, solid cells and (multi-CV) partial cells: a) fluid flow analysis for fluid regions; b) heat transfer and direct electrical current calculation in solid regions.

To predict turbulent flows in fluid regions FloEFD solves the Favre-averaged Navier-Stokes equations, where time-averaged effects of the flow turbulence on the flow parameters are considered. In addition the relevant equations which are formulations of mass, energy and contaminant conservation laws are solved.

To close this system of equations, FloEFD employs the modified k-ε turbulence model with damping functions proposed by Lam and Bremhorst.

These equations are supplemented by fluid state equations defining the nature of the fluid, and by empirical dependencies of fluid density, viscosity and thermal conductivity on temperature. Special models are used for the description of non-Newtonian fluids, real gases, volume condensation and vaporization, cavitation, combustion and porous media.

Within solid regions FloEFD calculates two kinds of physical phenomena: heat conduction and direct electrical current, with the resulting Joule heating being a source of heat in the energy equation.

The main issue for Cartesian immersed-body meshes is the resolution of boundary layers on coarse meshes. The wall treatment that forms part of the EFD platform technology uses a novel and original Two-Scale Wall Function (2SWF) approach (Mentor Graphics, 2011) that consists of two methods for coupling the boundary layer calculation with the solution of the bulk flow: 1) A “thin” boundary layer treatment that is used when the number of cells across the boundary layer is not enough for direct, or even simplified, determination of the flow and thermal profiles (unique subgrid model is used); 2) A “thick” boundary layer approach when the number of cells across the boundary layer exceeds that required to accurately resolve the boundary layer (modified wall functions are used); and 3) In intermediate cases, a compilation of the two above approaches is used. These treatments are discussed in details in Sobachkin and Dumnov (2013).

FloEFD uses 3 different types of solver and related numerical algorithms for modeling fluid flows. The first solver is optimal for incompressible flows and flows with Mach numbers less than 3.0. Time-implicit approximations of the continuity and convection/diffusion equations (for momentum, enthalpy, etc.) are used together with an operator-splitting technique (see for example Patankar, 1980). This technique is used to efficiently resolve the problem of pressure-velocity decoupling. Following a SIMPLE-like approach (Patankar, 1980) an elliptic type discrete pressure equation is derived by transformations of the originally-derived discrete equations for mass and momentum, taking into account the boundary conditions for velocity.

To solve the asymmetric systems of linear equations that arise from approximations of momentum, temperature and species equations, a preconditioned generalized conjugate gradient method from Saad (1996) is used. Incomplete LU factorization is used for preconditioning. To solve the symmetric algebraic problem for pressure-correction, an original double-preconditioned iterative procedure is used. It is based on a specially-developed multigrid method from Hackbusch (1985).

The second solver is optimal for High Mach number tasks with shock waves and other related phenomena. This explicit numerical solver (see Gavriliouk et al., 1993) is based on modification of Godunov’s method. This solver is also used in FloEFD for modeling of hypersonic flows of air with Mach number up to 30, by taking into account the phenomena of air ionization and dissociation.

The third recently-proposed solver in FloEFD is used for the calculation of flows in liquids with cavitation, using a numerical approach that is essentially new for CFD (see Sobachkin and Dumnov, 2013). This approach combines “pressure-based” and “density-based’ methods in special way. It is named as “hybrid” solver.

## 3. Verification and Validation (V&V) Methodology

There are several methods utilized in code Verification. One of the very popular in developing and testing of commercial CFD codes method is mesh dependency test. But one should make sure that next points are the same during the test: a) equations solved and engineering techniques and models in each subdomain; b) geometry for all meshes; c) mesh topology; d) the order and type of all equations approximations in each subdomain.

Meeting the first requirement is not ensured for real engineering problem calculations using FloEFD because different engineering techniques or their combinations can be used as a mesh gets finer or coarser as mentioned above. For the most examples it is actually impossible to separate Verification and Validation activity. That is because a mesh dependence study will show the integral accuracy of the code and not only correctness of the numerical algorithms.

The four-level classification of validation examples and tests is employed in the current practice for the V&V-procedure of FloEFD code. The first level involves the fundamental (academic) tests which are simple enough in sense of a geometry and problem formulation. At the second level are groups of tests that demonstrate how well complicated functions of the software or particular physical models are working. 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. The last level integrates validation tests and benchmarks from certain industry which some authors associate with code Certification or even Accreditation.

Some of the FloEFD V&V examples and tests are demonstrated by Ivanov et al. (2013) in more details.

## 4. Validation Example: NASA Common Research Model Aerodynamics

One of the industrial problems and benchmarks is an external aerodynamic simulation around the wing/body/horizontal-tail configuration of the NASA Common Research Model (CRM) at a cruise Mach number of M=0.85 and various *Re* numbers and angle of attack. Details on the NASA CRM and its geometry are given in Vassberg et al. (2008).

The main purpose of this investigation was to obtain aerodynamic characteristics of the vehicles such as lift, drag, pitching moment coefficients and pressure coefficient which were compared with experimental data. The main configuration for this simulation was WBT0.

Calculations were provided at the following far field conditions: M=0.85, *Re*=5.0E+06, $P_\infty$=201300 Pa, $T_\infty$=310.9 K and M=0.85, *Re*=30.0E+06, $P_\infty$=317772 Pa, $T_\infty$=116.483 K. The angle of attack varies in whole experimental range (from -3.0$^0$ to +12.0$^0$ for *Re*=5.0E+06 and from -3.0$^0$ to +6.0$^0$ for *Re*=19.8E+06 and 30.0E+06). The best results were obtained on the model with local mesh around the aircraft and several refinements during calculation by Solution Adaptive Refinement (SAR) technology in FloEFD. Attention should be paid to fine mesh resolution in the neighborhood of wing leading edge. It was found that realized in FloEFD unique subgrid model of turbulent boundary layer (or “thin” boundary layer) provided better results as compared with traditional approach based on wall function treatment. The computational mesh with ~4.30E+06 cells is enough to obtain reliable calculation results.

The model of the NASA CRM and surface pressure distribution are presented in Figure 3.

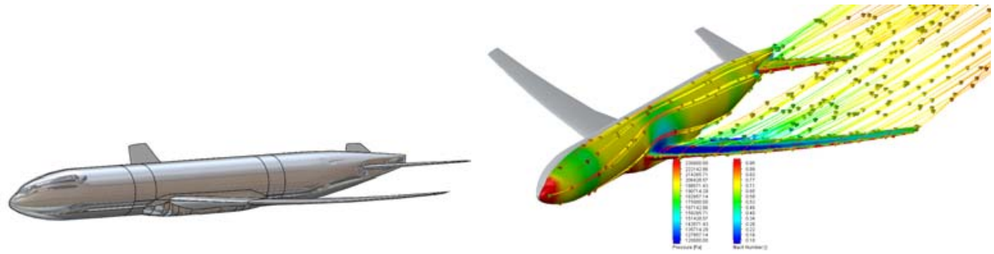

FIGURE 3. WBT0 CAD model (left) and the pressure distribution with flow trajectories colored by Mach number at angle of attack 4° (right).

Predicted chordwise pressure coefficient distribution at 50.2% semi-span and main integral aerodynamics coefficients are compared with experiments (Rivers and Dittberner, 2010) and presented in Figures 4, 5.

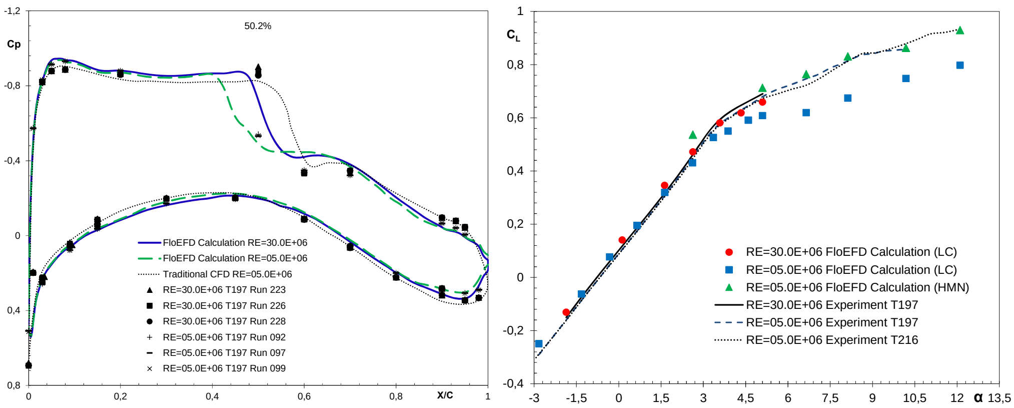


FIGURE 4. Pressure coefficient distribution (left) and lift coefficient (right).

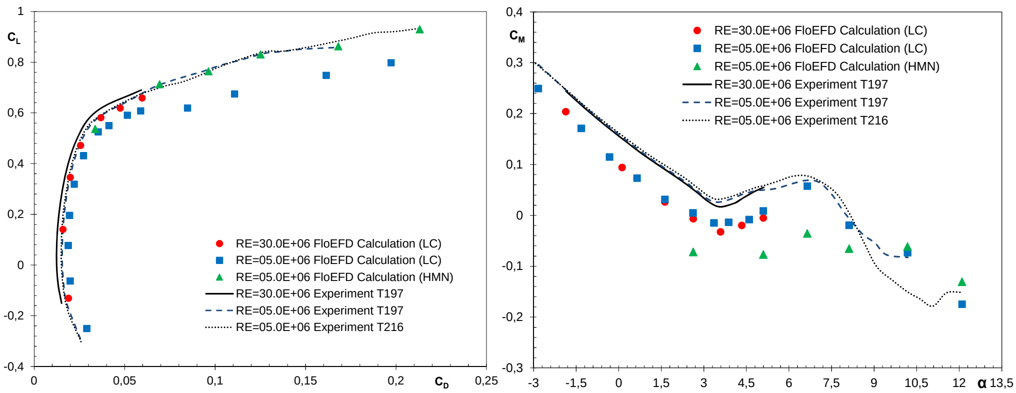


FIGURE 5. Polar (left) and pitching moment coefficient (right).

Rather close agreement was found between calculated and experimental pressure coefficient distributions especially in wing root and central section of the wing (average relative calculation error does not exceed 5%). Acceptable determination of shock wave position for both *Re* numbers should be noted (Fig. 4). Moreover, calculation results reproduce experimental trend that consists in shock wave position shift at *Re* number changing. Decreasing of *Re* number leads to shift of shock wave position upstream.

Good FloEFD predictions of the lift and drag coefficients especially in linear area were achieved for both Re numbers under investigation. It is worth to point out that predicted dependency of integral aerodynamics coefficients on *Re* number variation is in good agreement with experimental one in qualitative and quantitative relations. An exception to this statement is pitching moment coefficient. For this coefficient the discrepancy is bigger, but this is a common tendency for all CFD code. Probable causes of this fact are discussed in Rivers and Hunter (2012). But we suppose another factor is of importance – insufficient accuracy of downwash angle evaluation.

It should be noted that FloEFD makes possible a series of 'what-if' aerodynamic analyses and provides export of pressure and temperature as loads for structural analysis, on a structural mesh in NASTRAN, Creo Simulate etc. format directly, allowing automatic parameter changes rather than a manual approach.

Due to the use of a Cartesian-based mesh coupled with some engineering techniques and methods, FloEFD calculations reach acceptable accuracy on far coarser meshes when compared with traditional CFD codes. Therefore users can make calculations of fluid flow and heat transfer for very complex 3D cases with relatively modest computational resources.

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

FILIPIAK, M. (1996) *Mesh Generation*, Edinburgh Parallel Computing Centre, The University of Edinburgh, Version 1.0, November 1996.

GAVRILIOUK, V.N. DENISOV, O.P. NAKONECHNY, V.P. ODINTSOV, E.V. SERGIENKO, A.A. SOBACHKIN, A.A., (1993) 'Numerical Simulation of Working Processes in Rocket Engine Combustion Chamber', *44th Congress of the international Astronautical Federation*, IAFF-93-S.2.463, October 16-22, Graz, Austria.

HACKBUSCH, W. (1985) *Multi-grid Methods and Applications,* Springer-Verlag, NY, USA.

IVANOV, A.V., TREBUNSKIKH, T.V. and PLATONOVICH V.V. (2013) 'Validation Methodology for Modern CAD-Embedded CFD Code: From Fundamental Tests to Industrial Benchmarks' *Proceedings of NAFEMS World Congress NWC 2013*, Austria, Salzburg, June 09-12, 2013.

MENTOR GRAPHICS CORPORATION, (2011) *Enhanced Turbulence Modeling in FloEFD$^{TM}$*, MGC 02-11, TECH9670-w.

PARRY, J. and TATCHELL, D. (2008) *Flomerics' EFD Meshing Technology*: A White Paper.

PATANKAR, S.V. (1980) *Numerical Heat Transfer and Fluid Flow,* Hemisphere, Washington, D.C.

RIVERS, M.B., and DITTBERNER, A. (2010) 'Experimental Investigation of the NASA Common Research Model (Invited)', *AIAA Paper 2010-4218*, June 2010.

RIVERS, M.B., HUNTER, C.A. (2012) 'Support System Effects on the NASA Common Research Model', *AIAA Paper 2012-0707, 50$^{th}$ AIAA Aerospace Sciences Meeting Including the new Horizons Forum and Aerospace Exposition*, Nashville, Tennessee, 09-12 January, 2012.

SAAD, Y. (1996) *Iterative methods for sparse linear systems,* PWS Publishing Company, Boston.

SOBACHKIN, A.A., DUMNOV, G.E. (2013) 'Numerical Basis of CAD-Embedded CFD' *Proceedings of NAFEMS World Congress NWC 2013*, Austria, Salzburg, June 09-12, 2013.

VASSBERG, J.C., DEHAAN, M.A., RIVERS, S.M. and WAHLS, R.A. (2008) 'Development of a Common Research Model for Applied CFD validation studies', *AIAA Paper 2008-6919, 26$^{th}$ AIAA Applied Aerodynamics Conference*, Hawaii, HI, August, 2008.