VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

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

Dr. A.V. Ivanov, QA Test Manager

THEME

Confidence in Results: Verification & Validation; Benchmarks & Test Cases

SUMMARY

FloEFD is a new class of CFD (Computational Fluid Dynamics) analysis software (called Concurrent CFD) that is fully embedded in the mechanical design environment, for all general engineering applications. FloEFD was developed by Mentor Graphics' Mechanical Analysis Division, which is one of the top three CFD vendors in the world today and has been the leader in multiCAD-embedded CFD for the last 20 years.

As with all novel technologies, considerable attention is paid to Validation and Verification (V&V) of FloEFD. As the end user of Concurrent CFD software is a professional engineer, this places strict requirements on calculation accuracy, reliability and robustness, as well as the usability of the software.

Validation aims to provide users with comprehensive information about software functionality and its ability to correctly simulate the main physical phenomena underlying fluid flow and heat transfer processes, which occur in equipment as designed and in situ (e.g. in process plant).

This paper will describe the methodologies used in the V&V of an immersed boundary CAD-embedded CFD code which involves four distinct levels of testing. The first level involves the fundamental tests which are simple enough in terms of geometry and problem formulation. These tests are used to verify basic physical laws and algorithmic correctness. At the second level there are groups of tests that demonstrate how well a particular function of the product or physical model is working (e.g. combustion, conjugate heat transfer, cavitation, condensation, etc.). The third level is comprised of applied industrial problems and benchmarks. At this level software validations for

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

specific equipment with complex geometry are considered (cyclones, heat exchangers, engines, blowers, pumps, etc). The last level integrates validation tests and benchmarks from a certain industry (e.g. aerospace & defence, electronics, HVAC, process, etc.) as a prerequisite for certification or accreditation. In general the categorization of cases within these levels depends on geometric and flow complexity, availability of reference data and its accuracy, and so on. For each level a small selection of FloEFD validation examples are given in this paper.

Example cases from the first and second levels are provided with the software as CAD geometry plus boundary conditions and other numerical settings needed to mesh and solve the problem, so user can replicate validation cases on their own hardware, and use these to augment the tutorial examples provided with the software.

KEYWORDS

Validation, Verification, Benchmarks, Testing, FloEFD, CFD, CAD-embedded, Engineering

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

1: INTRODUCTION

Nowadays it is impossible to produce competitive, high-quality products without computer-aided engineering (CAE) software. The increasing role of CFD calculations within CAE has been observed in recent years.

The largest efficiency in using CAE systems (and CFD in particular) is achieved by inserting them directly into the product design process by the utilization of CAE/CFD not only by dedicated departments, but also by mechanical engineers engaged directly in design, particularly when used upfront in early design, i.e. design-concurrent CFD (Concurrent CFD). This process, which was initially initiated in aerospace, automotive, electronics and other high-technology industries, now covers practically all engineering fields.

The immersed-boundary CAD-embedded CFD code FloEFD represents a new class of CFD analysis software that was initially intended for mechanical engineers to use during the design process as an integral part of a product lifecycle management (PLM) concept. To develop such a class of CFD software the following question should be answered: what are the specific characteristics of a mechanical design engineer as a CFD user?

1. Using various CAD systems as a main design tool. 3D product model data are both the foundation and starting point for all virtual prototyping and physical simulations today. So, often users do not want to convert geometry into other formats to use it subsequently in a traditional CFD workflow. Moreover, in particular cases, and generally for very complex geometrical assemblies, the adequacy of such conversions is not guaranteed.

2. Lack of a background in CFD as well as the theoretical basis of the numerical algorithms.

3. The need to run multiple optimization calculations with geometric variation rather than single conceptual calculation. In most cases the user needs a "submachine gun" that never jams, rather than a "sniper rifle" that is more exacting.

4. CFD calculations are not the user's primary job function. These are auxiliary tasks, so an individual user may make calculations only occasionally, but then intensively for a period of time. Moreover, these calculations should be made as rapidly as possible, often with limited computational resource availability.

Naturally, the significance of each of the abovementioned characteristics depends on the specific industry in which the engineer is working. Nevertheless all characteristics should be taken into account to make the CFD tool less expensive to use and more suited to the general professional engineer as the target user persona. FloEFD achieves this in part by being fully

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

embedded in the mechanical design environment, for all general engineering applications.

The basic concept behind the design of FloEFD is to automate preparing, performing and visualizing CFD predictions of real applied engineering problems. To accomplish this FloEFD has some specific features, namely: complete integration with all major CAD-systems; totally automatic grid generation; automatic prescription of solution control parameters; user-friendly pre- and post-processing; ability to perform parametric studies and compare results for design variants, etc. The code does not require the tuning of any numerical parameters associated with the underlying algorithms or the choice of one of ten (or more) physical models or numerical schemes. It is important to note that assignment of initial data (boundary and initial conditions), performing the calculation, and analysis of results (including visualization and report generation) takes place inside the CAD system with results displayed directly on and around the CAD model. The export of calculation results in MS Office formats and for import into third party structural analysis codes (for instance Creo Elements/Pro Mechanica, Nastran) is also available.

In comparison with traditional CFD codes oriented towards high-level specialists in CFD, FloEFD is designed for practicing engineers with a different special interest: that of solving daily problems inherent in industrial product design and process optimization. As a rule, the software training period takes about two working days. In the event of a prolonged rest period, minimum effort is needed for the user to revive their proficiency. Entire simulations from initial data handling to result analysis can be performed in the course of a single work day.

FloEFD’s technology exhibits another significant difference from the traditional CFD approach in that it uses a number of engineering techniques and methods that assist the user in obtaining reliable predictions at lower computational and time costs. These combined possibilities allow engineers to accelerate the solution of their everyday problems, but place high demands on the software’s reliability, robustness and accuracy in order to automate these engineering methods. This challenge has been the driver behind FloEFD’s Validation and Verification (V&V) procedures since its inception, which use 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 FloEFD’s V&V methodology and classifications. Details of technology are not considered in the paper.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

2: VALIDATION METHODOLOGY

First and foremost, the essential distinction between code Verification and Validation should be discussed. Following Roache (1998), we adopt the succinct description of “Verification” as “solving the equations right”, and of “Validation” as “solving the right equations”. Another way to make the distinction between Verification and Validation is to follow the classical distinction between mathematics and science. Mathematics is a tool of science, often the predominant language of science. But mathematics exists by itself. Verification is seen to be essentially an activity in mathematics of numerical analysis. Validation is essentially an activity in science (and engineering science): physics, fluid dynamics, chemistry etc.

Most authors (e.g. Roache, 1998) strongly believe that complete Verification of a code (or a calculation) should precede any comparisons with experimental data, i.e., Verification first, then Validation. It is necessary to make some comment on this statement with regard to the immersed boundary CAD-embedded CFD code FloEFD.

There are several methods utilized in code Verification. These are Richardson extrapolation (when applicable), calculation with a high- and low-order method on the same mesh, and straightforward repeat calculations with finer or coarser meshes. The last method, also known as a grid dependency test, is very popular in developing and testing of commercial CFD codes. But one should keep in mind that Verification in strict sense is only realizable if all the following requirements are met during the test:

- the same equations are solved and the same engineering techniques and models (including sub grid scale ones) are used in each computational cell;
- the geometry of all components is retained for all meshes under investigation;
- the mesh topology in the computational domain is the same; and
- the order and type of all equations approximations in each computational cell are the same.

As mentioned above, FloEFD uses a number of engineering techniques and methods. So, meeting the first requirement is not ensured for real engineering problems, because different engineering techniques or their combinations are used automatically 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. That is because a grid dependence study will show the integral accuracy of the code and not only correctness of the numerical algorithms.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

We agree with Melnik et al. (1995) that for project-oriented engineers (and, of course, for code intended for them), the activity of code Verification and Validation almost form a continuum, and these terms are often used together to refer to the suite of activities, and even as an acronym for the process. That important factor has to be taken into account when planning and undertaking any FloEFD validation activity.

Another point that arises from the use of engineering techniques and methods is that FloEFD calculations reach acceptable accuracy on coarser meshes as compared with traditional CFD codes, confirmed by grid dependency tests for most examples and real engineering problems. Due to this, users can solve very complex 3D fluid flow and heat transfer problems using modest computational resources.

Let us now consider code Validation. Validation is the process of determining the degree to which a code, model, simulation, or combination of models and simulations, and their associated data are accurate representations of the real World from the perspective of the intended use (Missile Defense Agency, 2008). Put another way, does the solution of the equations implemented in the code bear any relation to a physical problem of interest?

Naturally, to engineers and scientists Validation is most important. Code Validation comes down to comparison (directly or indirectly) of code predictions with physical experiments, empirical correlations and analytical solutions. The comparison can be direct or indirect. Indirect comparison occurs when a previously validated code is taken as a benchmark.

It should be noted here that absolute certainty regarding the quality of experimental data is a rare occurrence. In many experiments the level of error cannot be determined with confidence. It is now the dominant opinion (Roache, 1998) that there is a continuing need for high quality experiments that are designed specifically for CFD code Validation. Sourcing such experimental data, its analysis and estimation of its accuracy also forms part of the code Validation activity.

To move closer to our current subject of Concurrent CFD, one can formulate that Validation aims to provide users with comprehensive information about software functionality and its ability to correctly simulate the main physical phenomena underlying fluid flow and heat transfer processes, which occur in equipment when in operation.

There exist many classifications of validation examples and several approaches to Verification & Validation have been analysed, e.g. Roache (1998); Stern et al. (1999); Oberkampf and Trucano (2002).

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

One approach is to classify according to the software's ability to simulate certain class of physical phenomena (natural convection, compressible flows etc.). Another is to classify according to the possibility of employing software in certain technical areas and applications (power & energy, rotating machinery etc.). A third approach is a two-level (or two-class) classification, in which benchmarks and validation examples are decomposed into two classes – fundamental tests and applied industrial problems. Each class has its own merits and demerits, but the two types complement one another nicely and formed the V&V procedure of FloEFD code for many years (Balakin et al. 2004). A fourth approach extends the two-level classification to a multilevel one. It is this approach, using four classification levels, that is currently employed in our V&V procedure for FloEFD, which we elaborate upon here.

The first level, as in Balakin et al. (2004), involves the fundamental (academic) tests which are simple in terms of geometry (2D as the rule) and problem formulation.

As mentioned above FloEFD's technology employs a large amount of engineering techniques and methods. These techniques and methods, first of all, touch on the simulation of wall effects (friction and heat transfer). Some of these techniques are unique, and at the same time largely unknown to users familiar with traditional CFD technology. That is why there is a rather comprehensive set of fundamental tests and examples in FloEFD's validation arsenal. These examples are associated with examination and demonstration of fundamental physical laws and phenomena (flows and convective heat transfer on a plate, in pipes, in channels and heat sinks etc.), as well as verifying algorithmic correctness.

The low cost of these tests makes it possible to conduct a parametric study of various regimes of heat and fluid flow over the maximum range investigated experimentally, numerically or analytically. Moreover, these fundamental tests are versatile, allowing the same configuration to be used to investigate various physical effects in either a coupled or segregated manner.

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 is comprised of industrial problems and benchmarks where, in addition to the complicated 3D geometry, a combination of different strongly-coupled physical phenomena takes place. Moreover, the exact values of material properties as well as operating conditions for device components are necessary in this case and so the level of experimental uncertainty is much higher. At this level software validations for specific equipment are considered (cyclones, heat exchangers, engines, blowers, pumps, etc).

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

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 simply 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, the borders between the levels are often fuzzy and the same validation example can be found at more than one level depending on the industrial application. In general the categorization of cases within these levels depends on example complexity, availability of reference data and its accuracy, and so on. As the levels progress in geometric and flow complexity, a tendency for decreased availability and reduced accuracy of experimental data is observed.

The V&V procedure currently employed for FloEFD is shown in Fig. 1. The diagram has a hierarchical structure and looks like an inverted pyramid, with each level being based on the previous one.

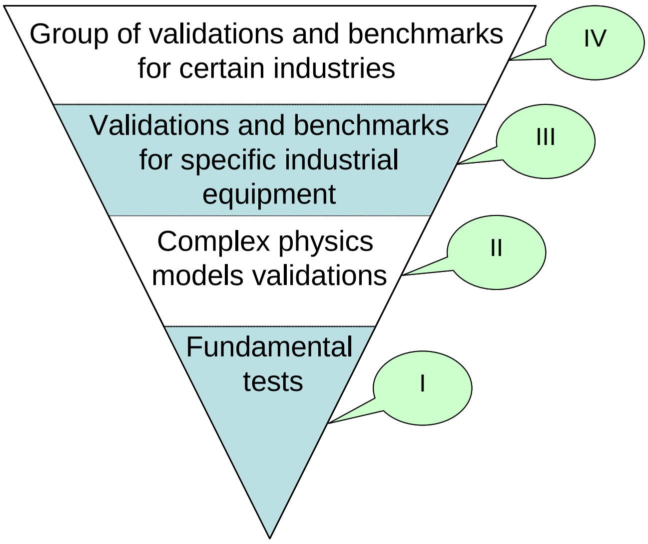

Figure 1: The four-level hierarchy used in FloEFD's Verification & Validation.

FloEFD is a general-purpose tool that has been successfully applied in many industries. Therefore, we actually have several pyramids as shown in Fig. 1. This is the reason why it would be more convenient to represent this pyramid aggregated in a 3D view analogous to the internal structure of Earth (Fig. 2).

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

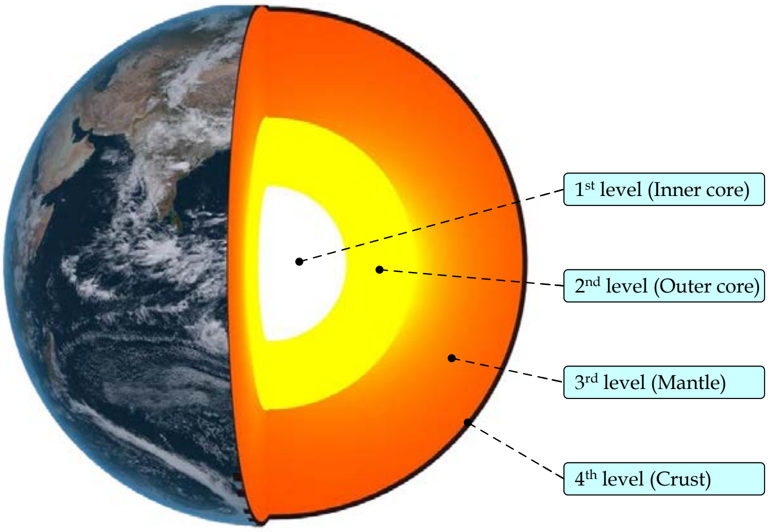

Figure 2: A 3D view of FloEFD code Validation as an "Earth internal structure".

Like Earth, FloEFD also has a stable inner core composed of fundamental validations and tests. Specified industries placed on the surface are analogous to the continents on the surface of Earth. Unlike Earth, FloEFD has a more dynamic structure. As FloEFD is developed Earth grows in size. Functionality, applicability and validity of the FloEFD code are also increased, meaning that new continents appear on Earth's crust and the outer core and mantle increase in thickness.

Another distinction between FloEFD and Earth is that FloEFD's internal structure can be asymmetric. This is because certain continents can be based on the layers with different thicknesses due to the different number of validation examples and physical models required for the different industries at the 4th level (code Certification).

It is also worth noting that as FloEFD is developed Validation activity is shifted to higher levels (mantle and crust) and explains why the previous V&V procedure of FloEFD code, based on two-class classification (Balakin et al., 2004), was replaced by the current V&V procedure based on four classification levels. It may well be that in the future FloEFD code development will lead to another modification of V&V procedure based upon a more advanced classification of validation examples and tests.

The four-level classification of validation examples with its 3D it's analogy to Earth's internal structure seems to be very helpful in support and marketing activities. The four-level classification meets requests from users wanting to see simple validations to understand how well separate physical processes are simulated, and requests from users wanting to see how well the technology can predict complex "real world" equipment performance.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

Validation and its methodology are associated with Quality Assurance (QA). Searching and collecting data for validations, data analysis, selection, performing calculations and documentation of cases examples takes a lot of time and resources of the QA team. These cases collectively form a battery of tests that has to be passed before the release of each software version or service pack. The number of validation examples is steadily increasing.

Example cases from the first and second levels are provided with the software as CAD geometry plus boundary conditions and other solution control settings needed to mesh and solve the problem, so the user can replicate validation cases on their own hardware, and use these to augment the tutorial examples provided with the software. The principle rule applied to the set up and solution of validation example test cases is that automatic settings of the code input parameters should be used in V&V procedure calculations. That means:

- totally automatic mesh generation (for fundamental validations, for other validation levels it is highly advisable); and

- settings for solution control convergence criteria are taken as their default values.

It is also possible to construct mesh in a non-automatic or manual way, e.g. a uniform mesh, or mesh stretching in accordance with user specified input parameters. In general at the first level of Validation fully automated meshing is used. Manual settings become more prevalent at the higher levels. Completeness and consistency of initial data as well as the mesh convergence are studied thoroughly for all examples and tests. Some typical V&V examples and tests are presented below.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

3: VALIDATION TEST AND EXAMPLES

1. Fundamental validations: flow over a plate with heat transfer

A uniform 2D flows with a laminar boundary layer on a heated flat plate is considered. The statement of the problem is presented in Fig. 3. Reynolds number defined on the plate length of 0.31 m is equal to 3.1×10^4, therefore the boundary layer is laminar (Holman, 1997).

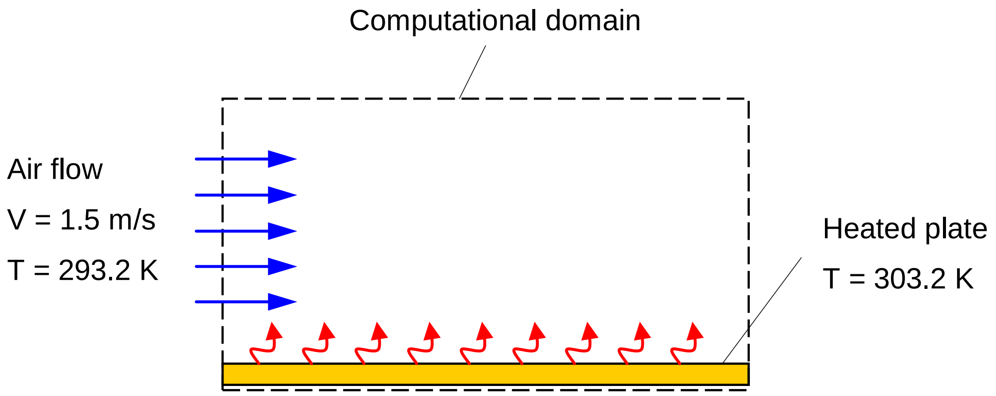

Figure 3: The statement of the problem.

The FloEFD predictions of h and C_f calculated with a fully automatically generated mesh with the result resolution level (RRL) set to 7, and the theoretical curves (Holman, 1997) are shown in Figs. 4 and 5. One can see that the FloEFD predictions are in good agreement.

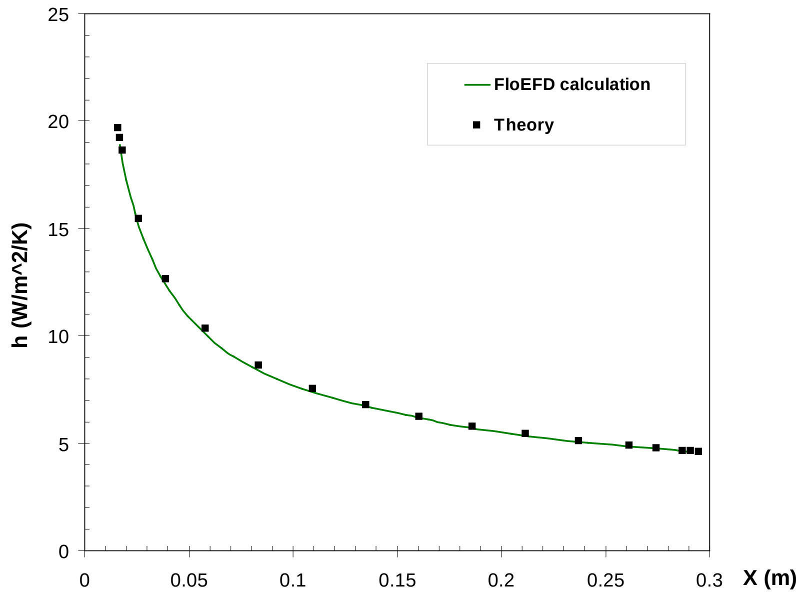

Figure 4: Heat transfer coefficient along the heated plate.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

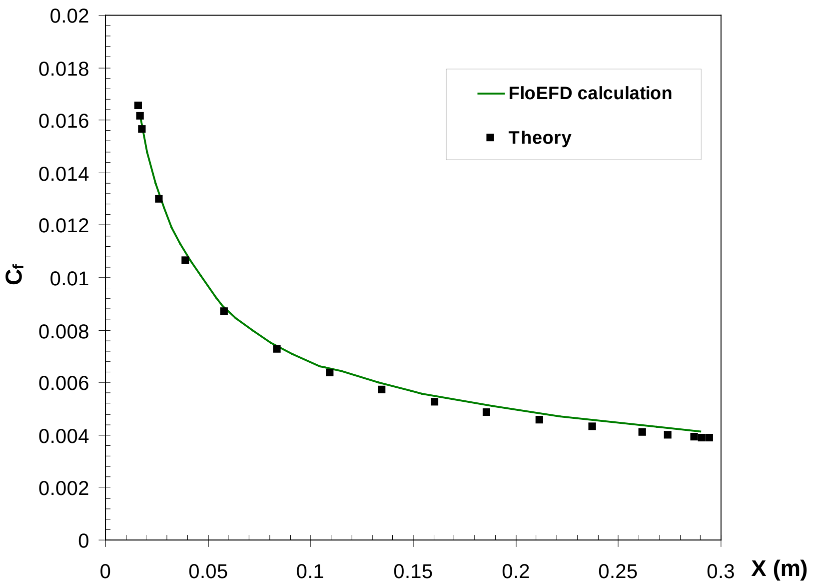

Figure 5: Skin-friction coefficient along the heated plate.

2. Fundamental validations: laminar and turbulent flows in pipes

Prediction of 3D water flow through a long straight pipe with circular cross section is considered (see Fig. 6). A uniform inlet velocity U_{inlet} is set.

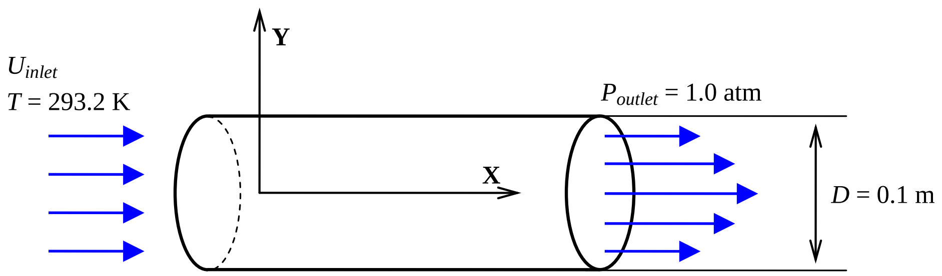

Figure 6: Statement of the problem.

Fig. 7 show the FloEFD predictions performed at RRL=5 for smooth pipes in the entire Re_d range and compared with theoretical values (Schlichting, 1979; White, 1994).

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

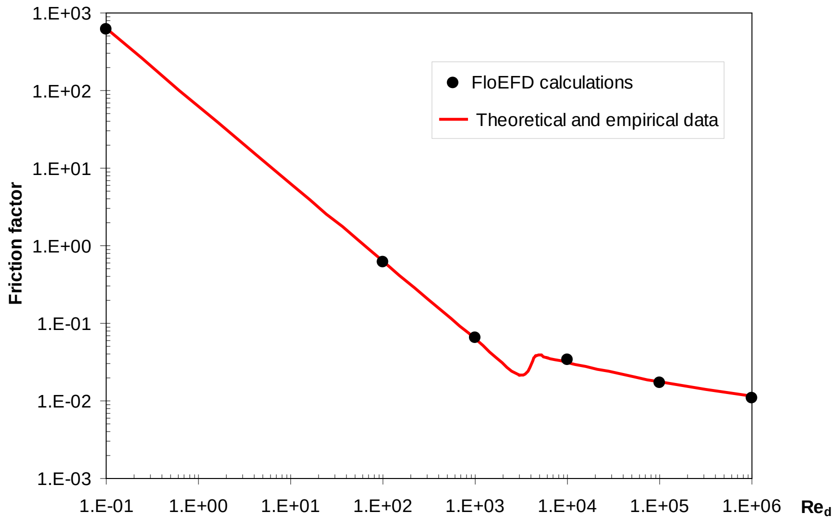

Figure 7: The friction factor for smooth pipes.

It can be seen that the friction factor values predicted for smooth pipes are fairly close to the theoretical and empirical curve. The prediction errors do not exceed 5%.

3. Fundamental validations: flow in a 90-degree bend square duct

In this case a steady-state flow of water in duct is considered (Humphrey et al., 1977). The geometry of the duct is shown in Fig. 8. Re_D = 790 meaning that the flow is laminar.

Inlet temperature is equal to 293.2 K and inlet uniform velocity U_{inlet} = 0.0198 m/s.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

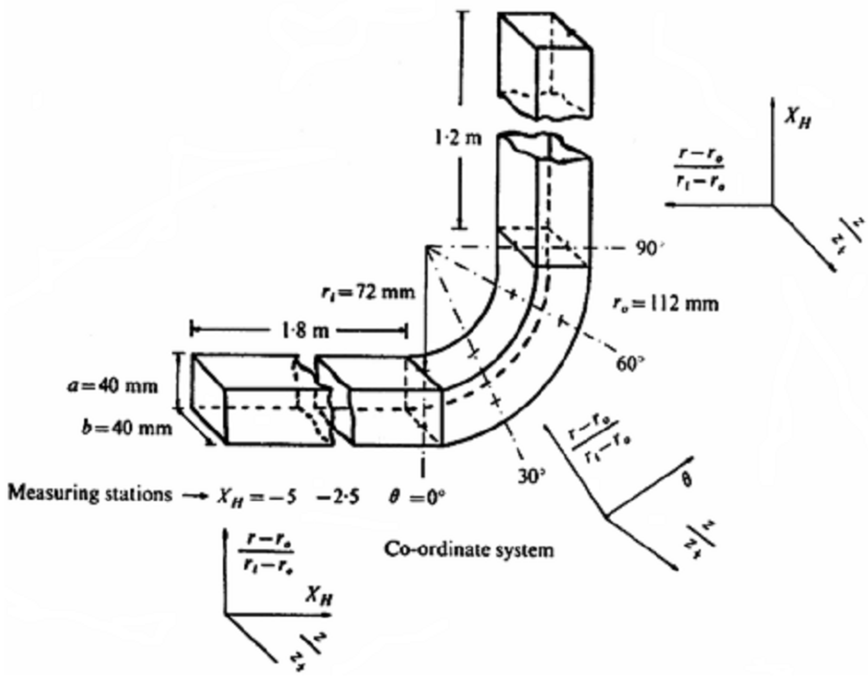

Figure 8: The 90°-bend square duct's configuration indicating the velocity measuring stations and the dimensionless coordinates.

The predicted dimensionless (divided by U_{inlet}) velocity profiles are compared in Figs. 9, 10 with the measured ones at the following duct cross sections: X_H = -5D, -2.5D, 0 (or θ=0°). The *z* and *r* directions are represented by coordinates $(r\text{-}r_0)/(r_i\text{-}r_0)$ and $z/z_{1/2}$, where $z_{1/2}$ = 20 mm.

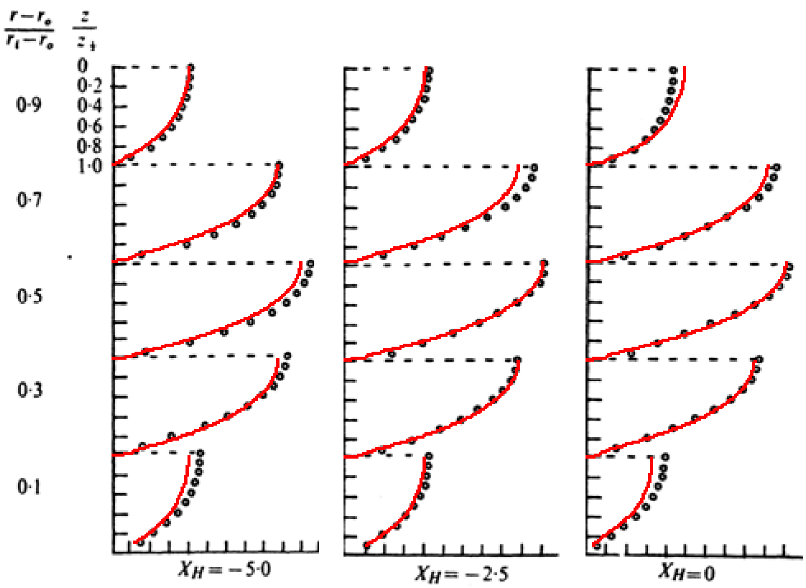

Figure 9: The duct's velocity profiles predicted by FloEFD (red lines) in comparison with the experimental data (circles).

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

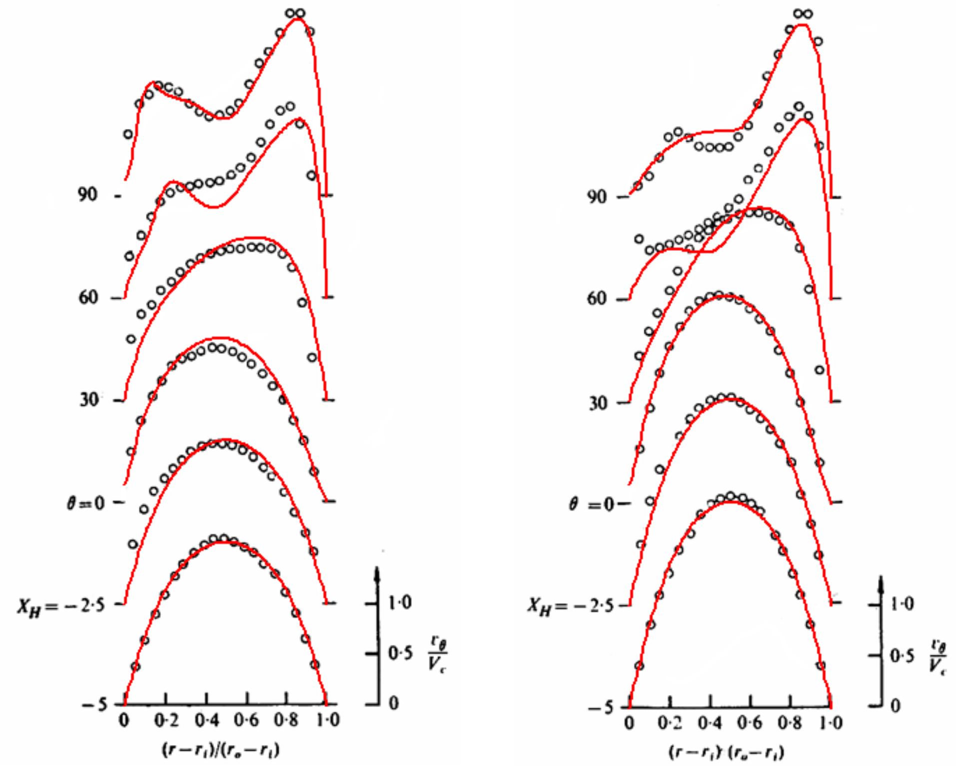

Figure 10: The duct's velocity profiles predicted by FloEFD (red lines) in comparison with the experimental data (circles) at $z/z_{1/2}$=0.5 (left) and at $z/z_{1/2}$=0 (right).

It is seen that the FloEFD predictions are in good agreement with the experimental data (Humphrey et al., 1977).

4. Fundamental validations: flow in 2D channel with unilateral sudden expansion

In this example laminar incompressible steady-state water flow through 2D (plane) channel with unilateral sudden expansion and parallel walls is examined. The sketch of the problem is shown in Fig. 11. Water temperature – 293.2 K, mean velocity – 8.25 mm/s.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

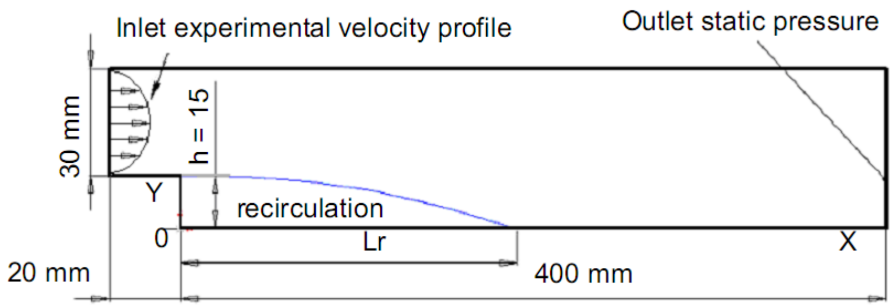

Figure 11: Flow in a 2D (plane) channel with an unilateral sudden expansion.

At the inlet an experimentally measured mean velocity profile (Denham and Patrick, 1974) at the corresponding Re_h=125 is specified. The 10^5 Pa static pressure is specified at the outlet.

The flow velocity field predicted by FloEFD with automatically generated mesh (RRL=8) is compared in Figs. 12-14 with the measured values (Denham and Patrick, 1974).

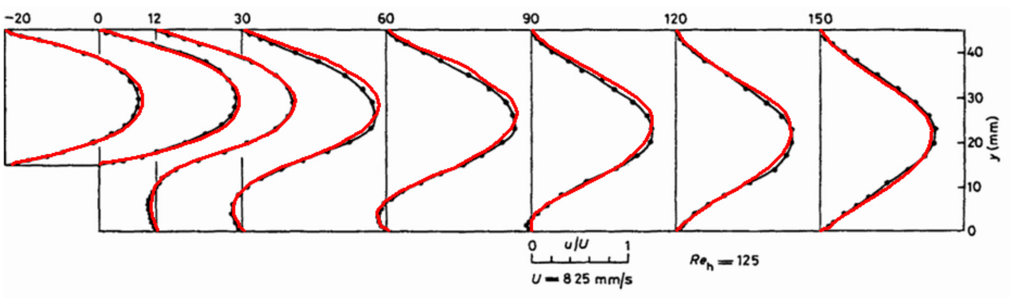

Figure 12: The velocity profiles predicted by FloEFD (red lines) in comparison with the experimental data (black lines with dark circles).

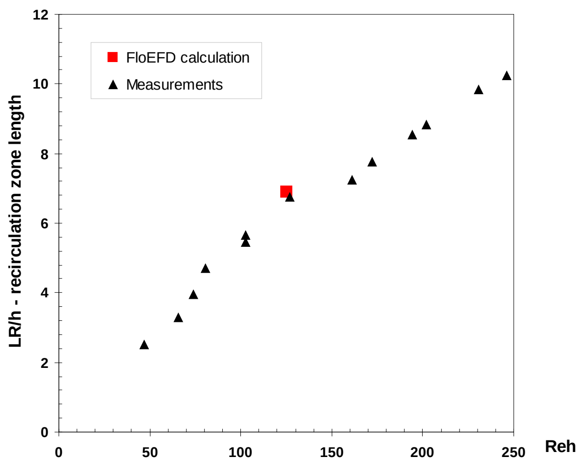

Figure 13: The recirculation zone length predicted by FloEFD in comparison with the experimental data.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

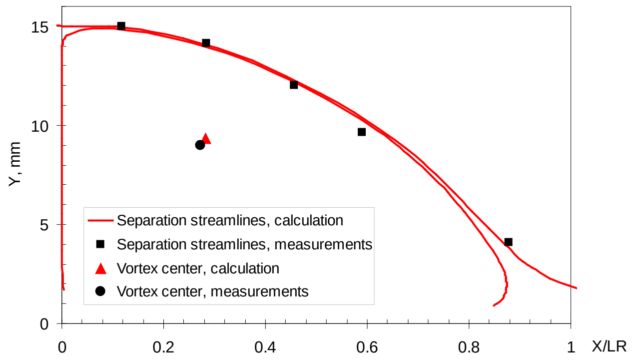

Figure 14: The recirculation zone's separation streamlines and vortex center, both predicted by FloEFD in comparison with experimental data.

The flow *X*-velocity (*u/U*, where *U* = 8.25 mm/s) profiles at several *X* = const cross sections are shown in Fig. 12. It is seen that the predicted flow velocity profiles are very close to the experimental values both in the main stream and in the recirculation zone.

The recirculation zone's characteristics, i.e. its length L_R along the channel's wall, the separation streamline, and the vortex center are shown in Figs. 13, 14. It is seen that they are in excellent agreement with the experimental data.

5. Fundamental validations: flow over a circular cylinder with and without heating

First of all, an incompressible flow over a cylinder without heating has been studied numerically in a wide range of governing parameters as a transient problem. It is well-known that at low Reynolds number $Re_D < Re_{osc}$ (Re_{osc} is about 45) two vortices are formed in a closed near wake. Fig. 15 demonstrates a very good agreement between FloEFD predictions and photo from Van Dyke (1982) for Re_D = 41 predicted with automatically generated mesh (RRL = 7).

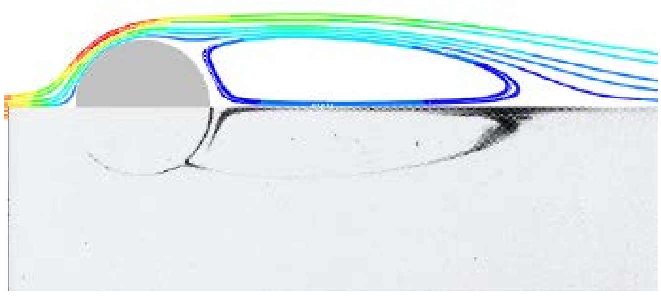

Figure 15: Predicted flow trajectories colored by the pressure magnitude (the upper) and photo from Van Dyke (1982) (the lower part) for Re_D = 41.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

At higher Reynolds numbers the flow becomes unstable and a von Karman vortex street appears in the wake past the cylinder. The FloEFD prediction of Strouhal number in comparison with experimental data (White, 1994) for $Re \geq 10^3$ is shown in Fig. 16.

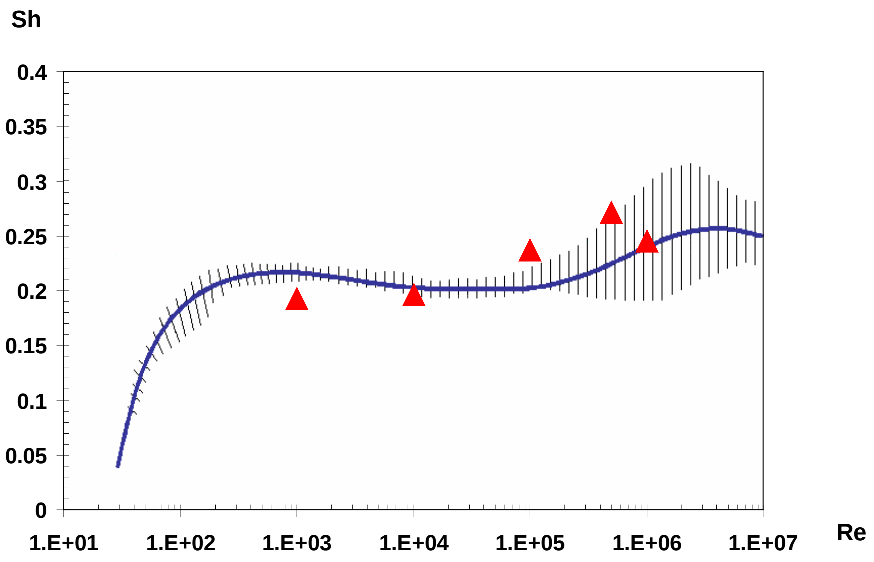

Figure 16: The cylinder flow's Strouhal number predicted with FloEFD (triangles) in comparison with the experimental data (line with dashes).

The calculated at RRL=7 time-averaged cylinder drag coefficient is compared to the well-known experimental data on $C_D(Re)$ (Panton, 1996) in Fig. 17. It is easy to see that numerical results are close to experimental data in wide range of *Re*.

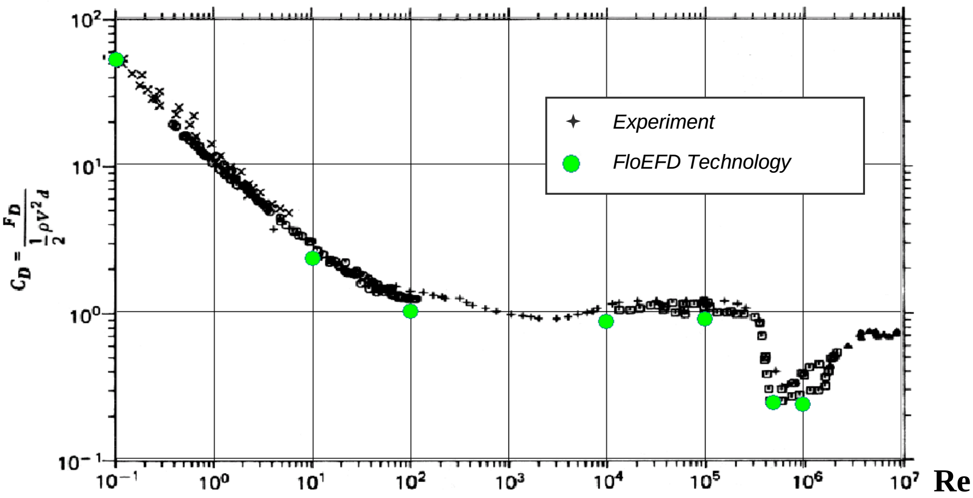

Figure 17: The drag curve for a cylinder.

The simplest modification of this problem is to consider convective heat transfer from a heated circular cylinder (with the total heat generation rate *q*) in an air flow.

An excellent correlation in Nu_D between computations and measurements (Holman, 1997) has been obtained in the whole considered range of *Re* (see Fig. 18).

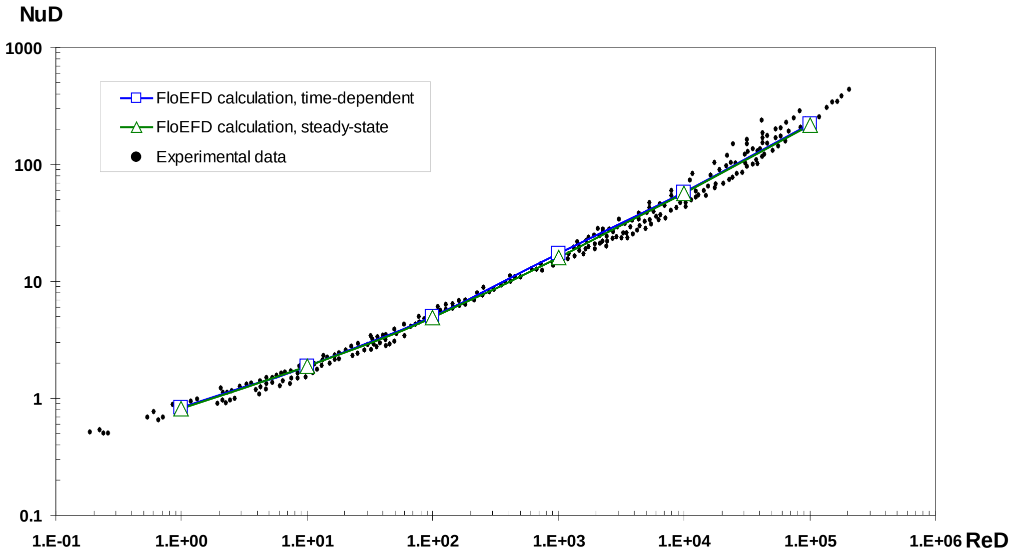

Figure 18: Nusselt number Nu_D for air flow over a heated cylinder.

6. Fundamental validations: buoyancy-driven cavity flow

This 2D test is classical for convective heat transfer. In this test a free convection is considered in a square cavity with isothermal side walls of different temperature value and the thermally insulated top and bottom (see Fig. 19). The cavity is filled with air.

The benchmark solution (Davis, 1983) has been obtained from high-accurate predictions of about 40 computer codes and moreover, it agrees very well with the semi-empirical formula of experimental researches (Emery and Chu, 1965).

The square cavity's side dimension, *L*, is varied within the range of 0.0111...0.111 m in order to vary the cavity's Rayleigh number within the range of 10^3 - 10^6.

A mesh convergence study for considered range of the Rayleigh number is presented in Fig. 20.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

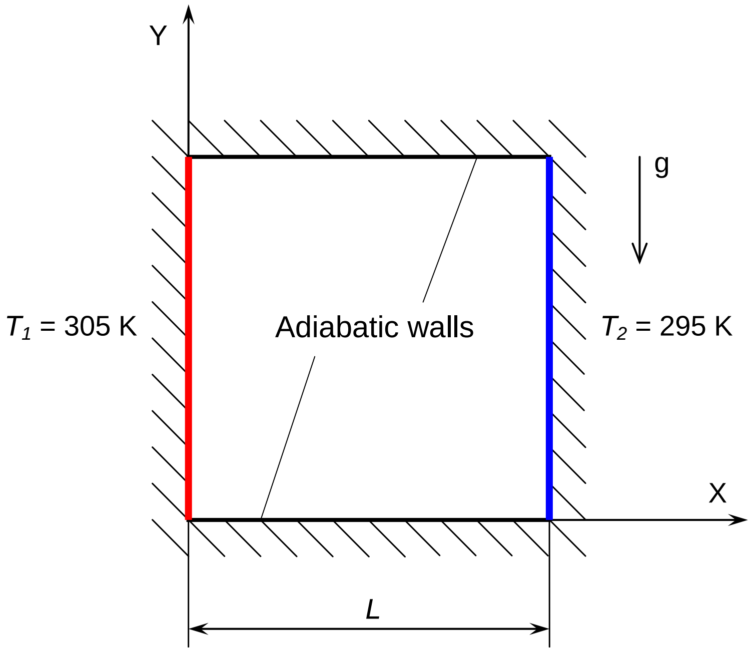

Figure 19: An enclosed 2D square cavity with natural convection.

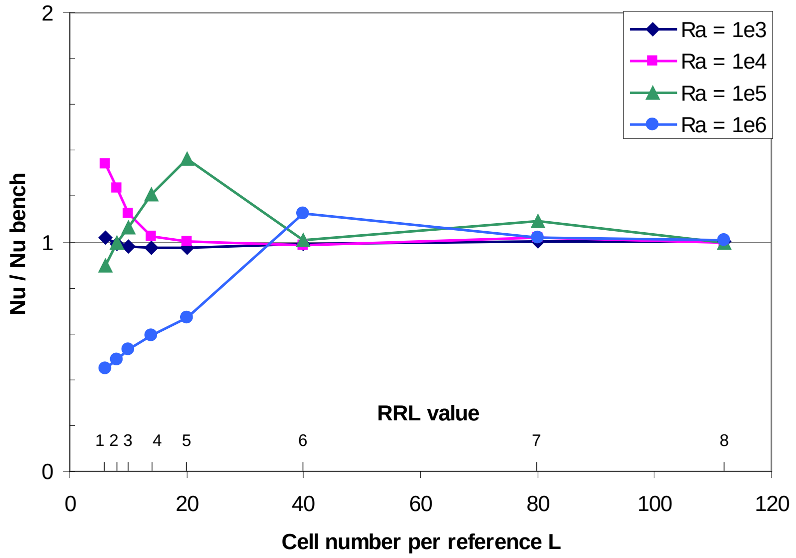

Figure 20: Mesh convergence study for various *Ra*.

This figure demonstrates dependence of ratio $Nu/Nu_{benchmark}$ both on the value of mesh automatic generation level (RRL) and on cell number per reference L (square cavity size). This plot confirms grid convergence achieved at RRL = 8. Numerical results derived at this value of RRL are shown below.

Fig. 21 shows the mesh derived after the dynamic adaptation to the solution peculiarities in the particular case of $Ra = 10^6$ predicted at the highest RRL = 8.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

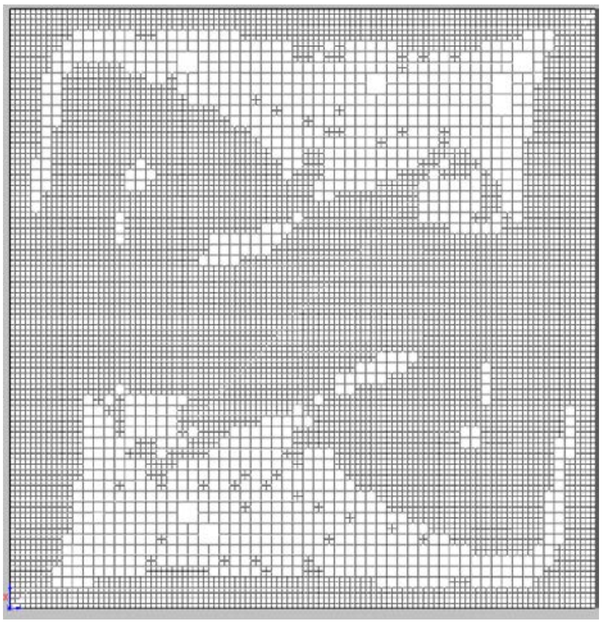

Figure 21: Adapted to the solution mesh at RRL = 8 for *Ra* = 10^6.

The next figures demonstrate a very good agreement between FloEFD predictions and the benchmark solution (Davis, 1983) both in thermal (see Fig. 22) and hydrodynamic (see Fig. 23, 24) fields for all considered Rayleigh numbers.

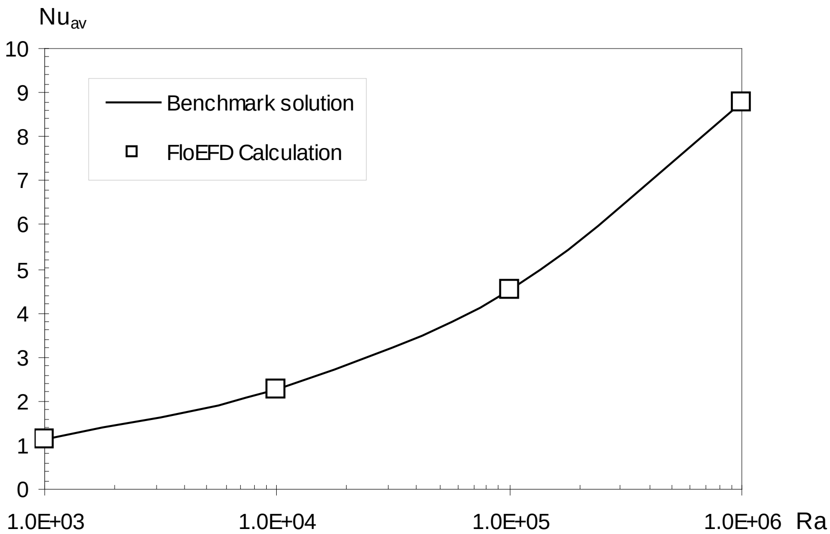

Figure 22: Average Nusselt number vs. Rayleigh number.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

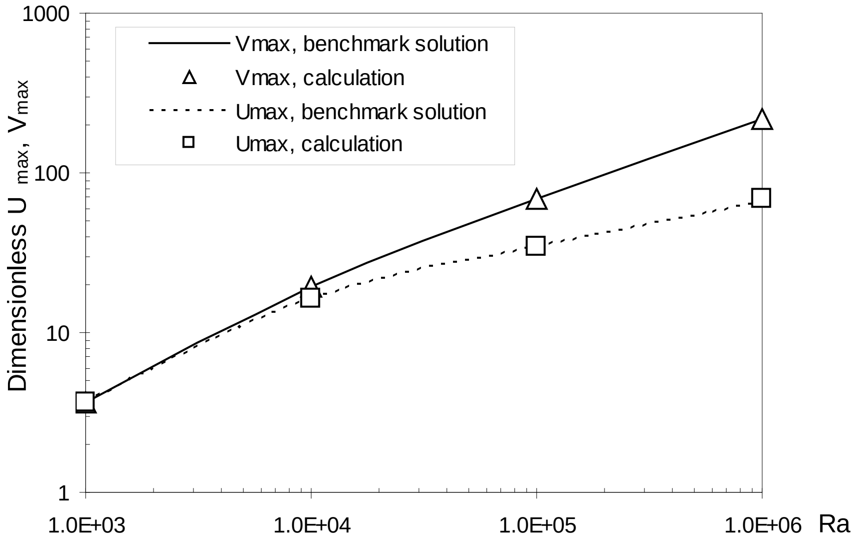

Figure 23: Maximum dimensionless velocity components vs. Rayleigh number.

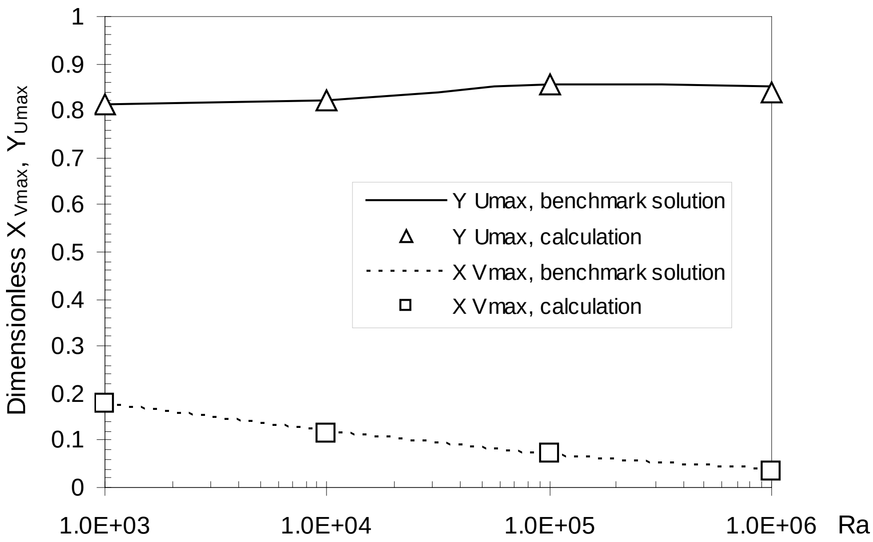

Figure 24: Dimensionless coordinates of the maximum velocities' locations vs Rayleigh number.

7. Fundamental validations: flow over RAE 2822 airfoil

In this example FloEFD prediction of 2D air flow around RAE 2822 airfoils is considered. The airfoil geometry is presented in Fig. 25.

Figure 25: RAE 2822 airfoil.

The airfoil chord length is 1.0 m. Computational domain size is 30×24 m. Computational mesh has 350×200 cells with finer ones in the vicinity of the aerofoil. Total number of mesh cells is about 70000.

Five test cases are considered. Flow conditions specified for each case are shown in Table 1 (Cook et al., 1979).

Case	M	α, °	Re	T, K	P, Pa
1	0.676	2.4	5.7e+6	300	38684.5408
2	0.676	-2.18	5.7e+6	300	38684.5408
3	0.725	2.55	6.5e+6	300	41132.45548
4	0.725	2.92	6.5e+6	300	41132.45548
5	0.728	3.22	6.5e+6	300	40962.95361

Table 1: Flow conditions for prediction flow over RAE 2822 airfoil.

Planar plot of computed Mach number is shown in Fig. 26.

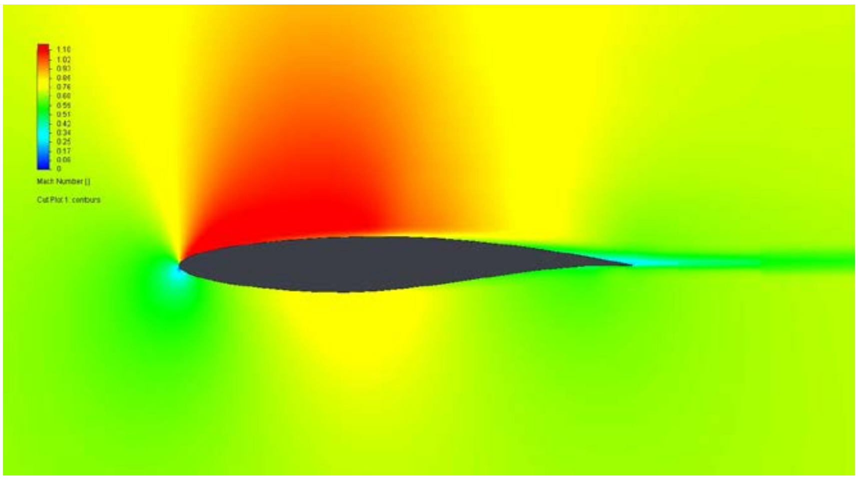

Figure 26: Mach number planar plot around the airfoil for test case 5.

In this test case a strong shock is visible on the upper surface at approximately the mid-chord position, which results in a thickening of the boundary layer downstream.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

The comparison of FloEFD predicted surface pressure coefficient distributions with experimental ones (Cook et al., 1979) for test case 5 is given in Fig. 27. In the presented case (Fig. 27) satisfactory agreement is seen between FloEFD calculation results and experiment both for overall distributions and in the position of the shock.

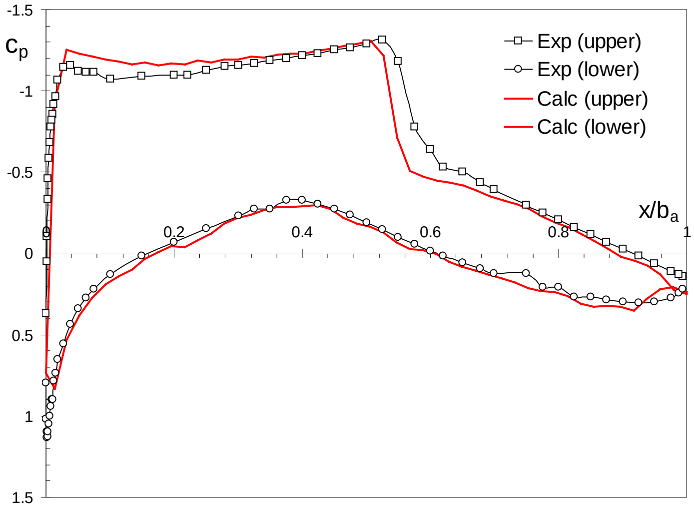

Figure 27: Comparison of computed and measured surface pressure coefficients for case 5.

As regard to integral aerodynamic coefficients C_L and C_D, the calculated ones are also in good agreement with experimental data. The predicted values are C_L =0.807 and C_D =0.0192. They give relative prediction error 0.61% and 9.5%, respectively.

Unfortunately because of lack of space descriptions of tests devoted to Validation of radiation models, heat conduction in solids, flows of non-Newtonian liquids, condensation models, real gases and so on exceed the limits of this paper.

Application of FloEFD to complicated physical phenomena is presented in the next section. For example, a combustor simulation test is given here that demonstrates how well such particular feature of FloEFD as combustion model is working.

8. Particular functionality or physical models validations: FloEFD validation of working process in lean premixed combustor

The combustion chamber, shown in Fig. 28, consists of a test section with a square cross-section and a conical bluff body made of stainless steel which is a flame holder.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

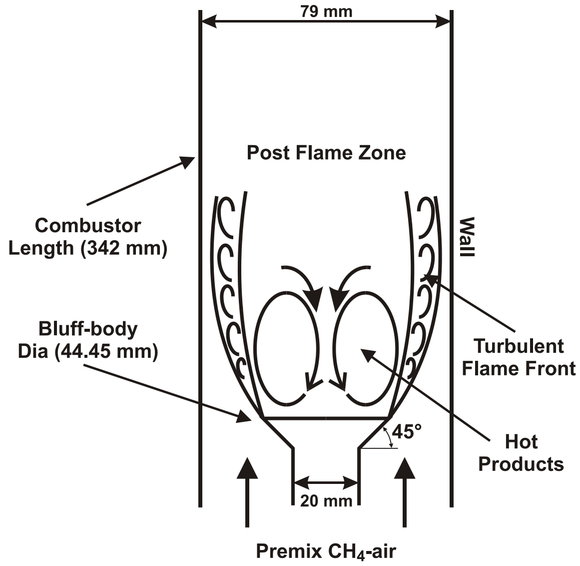

Figure 28: Layout of the rig and the Turbulent Flame Structure in the Bluff-Body-Stabilized Lean Premixed Combustor (Nandula et al., 1996).

At the entrance the following parameters are specified: velocity - 15 m/s; turbulence intensity - 24%; temperature - 300 K; pressure - 1 atm; air flow rate - 3962 slpm (standard liters per minute); fuel flow rate - 244 slpm. It gives an equivalence ratio (φ) of 0.586 and an adiabatic flame temperature (T_{ad}) of 1641 K.

The composition of the fuel and air are given in Table 2.

Fuel	Mass fraction, %
Methane	100.0
Air	
Oxygen	23.3
Nitrogen	76.7

Table 2: Fuel and oxidizer composition.

Calculations were performed in transient regimes. FloEFD limited combustion rate feature was used.

Automatically generated mesh with RRL=5 gave 10609 fluid cells of computational mesh overall. After two solution adaptive refinements, the overall number of fluid cells had grown up to 64415 cells.

The predicted planar temperature field with flow streamlines and final mesh are presented in Fig. 29.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

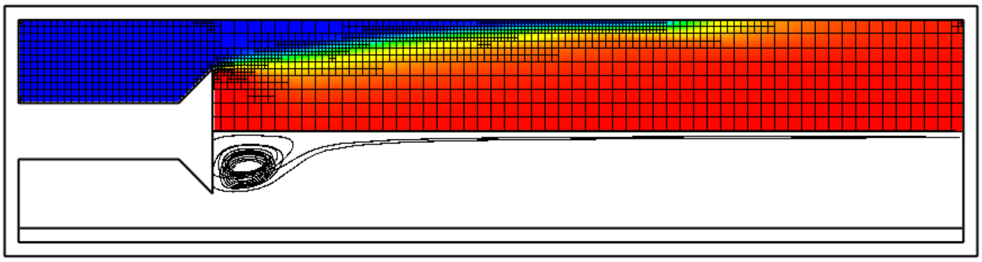

Figure 29: Predicted temperature contours and flow streamline in front (*Z*=0) plane.

The temperature and major combustion products component radial distributions at the distances of 0.1D, 0.6D and 6.0D from the leeward of the bluff body are shown in Figs. 30,31.

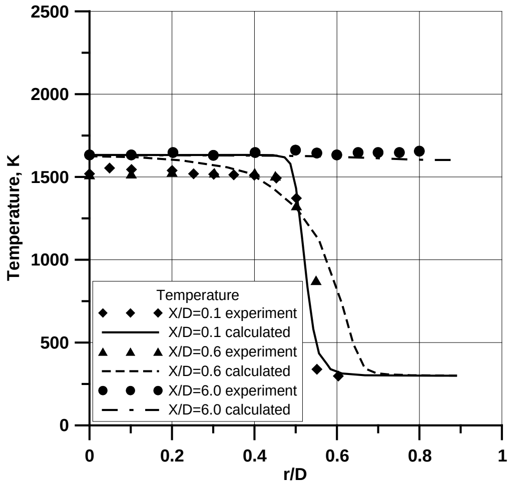

Figure 30: Comparison between the predicted and measured temperature profiles.

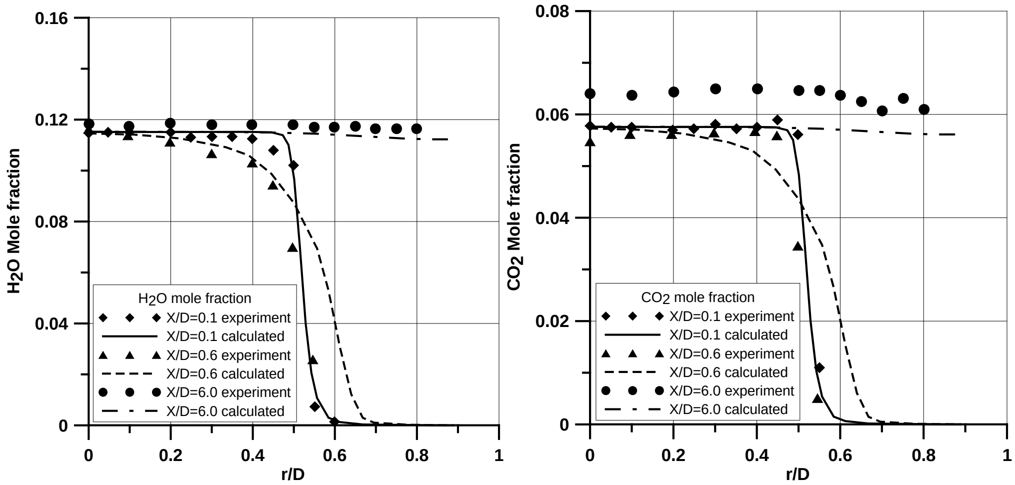

Figure 31: Comparison between the predicted and measured H_2O (left) and CO_2 (right) mole fractions profiles.

Here, the predicted temperature and combustion products components are compared with the measurements reported in Nandula et al. (1996). Very accurate prediction of the temperature and concentration levels inside the flame and around should be pointed out.

Good agreement between predicted and measured values clearly shows that FloEFD code accurately predicts the flow, temperature and major species fields.

In the next sections examples represent applied industrial problems and benchmarks and verify FloEFD code capabilities for specific equipment (cyclones, heat exchangers, engines, blowers, pumps, etc) where in addition to the complicated 3D geometry a combination of different strongly coupled physical phenomena takes place.

9. Industrial problems and benchmarks: flow simulation over a generic car body shape (the Ahmed body)

A classical automotive external aerodynamics wind tunnel test case is the so-called "Ahmed Body" (Lienhart et al., 2000) is considered.

An approaching air flow of 40 m/s at corresponding $Re = 7.68\times10^5$ is evaluated. All parameters of the car body were taken from Lienhart et al. (2000).

FloEFD calculations were performed with a computational mesh of 209 cells in length, 58 cells in height, and 78 cells in width (Fig. 32).

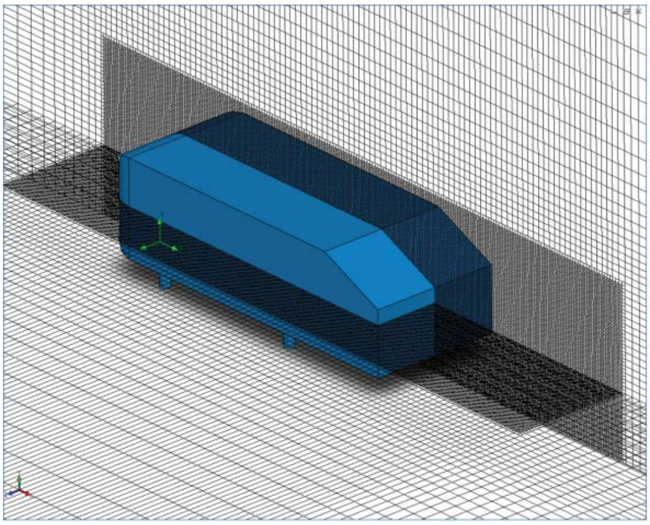

Figure 32: The FloEFD computational mesh over the model car body with the 25^0 rear slope.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

FloEFD calculated flow fields are shown in Fig. 33 for the two sloping rear angles. The FloEFD calculated flow velocity profiles and body drag coefficients in comparison with the experimental ones (Lienhart et al., 2000) are shown in Fig. 34 and Table 3.

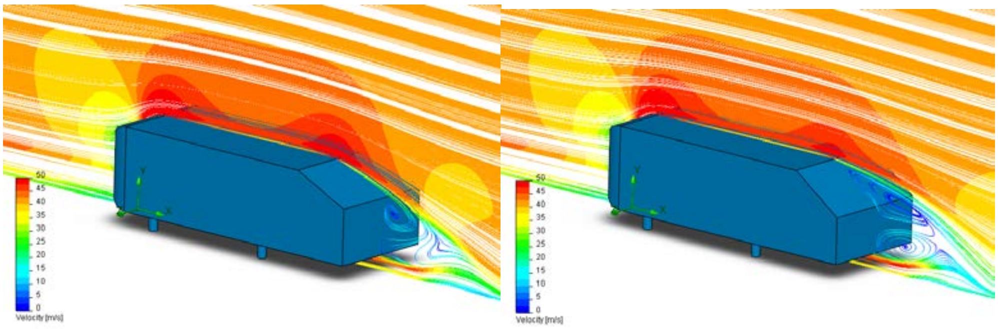

Figure 33: Calculated flow streamlines and velocity contours upstream, over and downstream of the model car body: 25^0 rear slope (left), 35^0 rear slope (right).

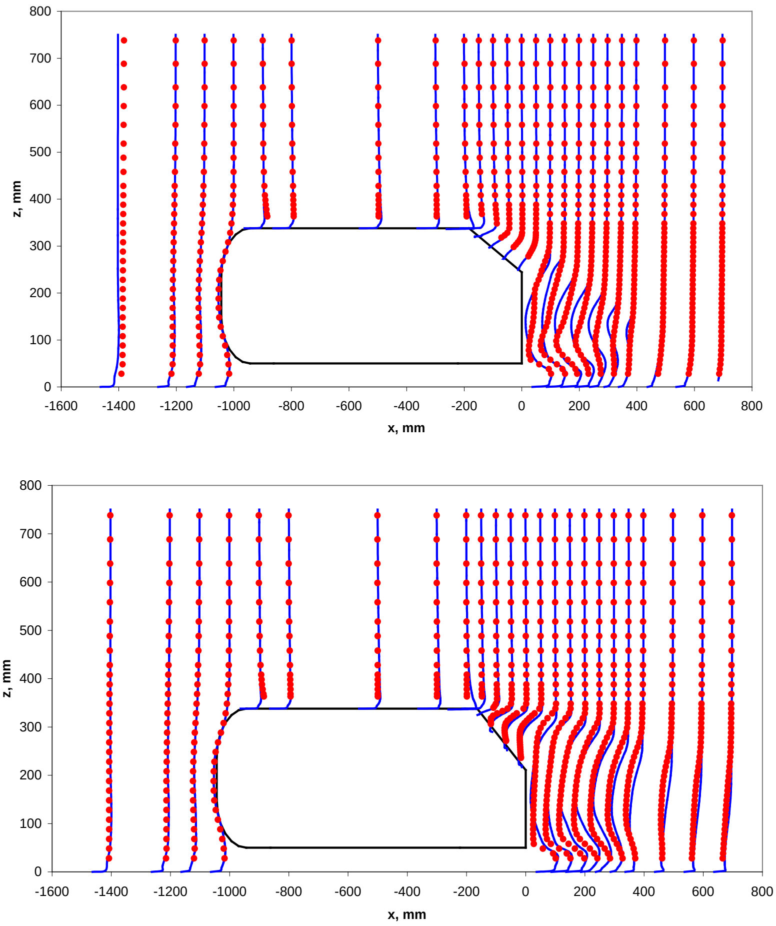

Figure 34: Velocity profiles in the body's symmetry plane at different body's slope angles (lines – calculation; red points – experiment): 25^0 rear slope (upper), 35^0 rear slope (lower).

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

Slope angle	$C_{d,\exp}$	$C_{d,calc}$	Error, %
25°	0.298	0.284	-4.8
35°	0.257	0.274	6.6

Table 3: The model car body's drag coefficient calculated with FloEFD and obtained in experiments.

It can be seen from Figs. 33 and 34 that calculated flow velocity profiles are close to the experimental ones. From Table 3 it is observed that the FloEFD calculated body drag coefficients agree well with the experimental ones.

10. Industrial problems and benchmarks: 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, as can be seen in Fig. 35.

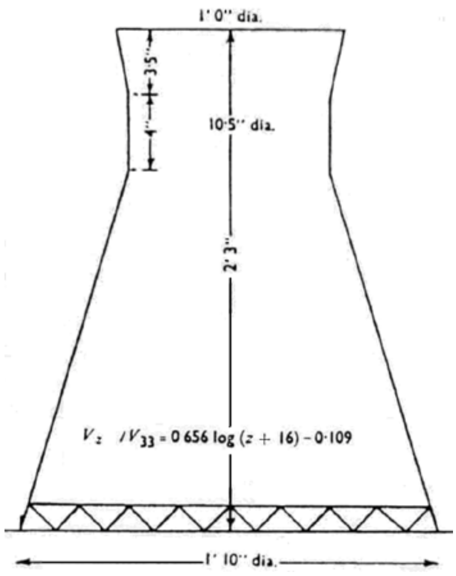

Figure 35: The cooling tower geometry.

The cooling tower base aperture was treated as sealed. The cooling tower was defined by the geometrical parameters given in Table 4. All presented parameters as well as experimental wind tunnel tests data were taken from Zdravkovich (2003), Cowdrey and O'Neill (1956).

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

Geometrical parameters	*Units*	*Value*
Overall height	in	27.0
Base diameter	in	22.0
Throat diameter	in	10.5
Top diameter	in	12.0
Cylindrical throat height	in	4.0
Upper truncated cone height	in	3.5
Air flow properties		
Temperature	K	293.2
Pressure	atm	1.0
Reference velocity V_{33}	m/s	103.9
Friction velocity U_*	m/s	7.86
Reynolds Number		≈6.0E6

Table 4: The cooling tower parameters and flow conditions.

The flow calculation problem was considered in following computational domain: length – 3.75 m, width – 1.25 m and height – 1.4 m. Only one half of cooling tower was taken into account for calculations. FloEFD calculations were performed with the initial mesh of 75 cells in length, 30 cells in height, and 25 cells in width which after refinement in the vicinity of the model gives computational mesh of about 580000 cells.

Fig. 36 shows predicted C_P distributions at Z/H=0.79 as compared with experiment.

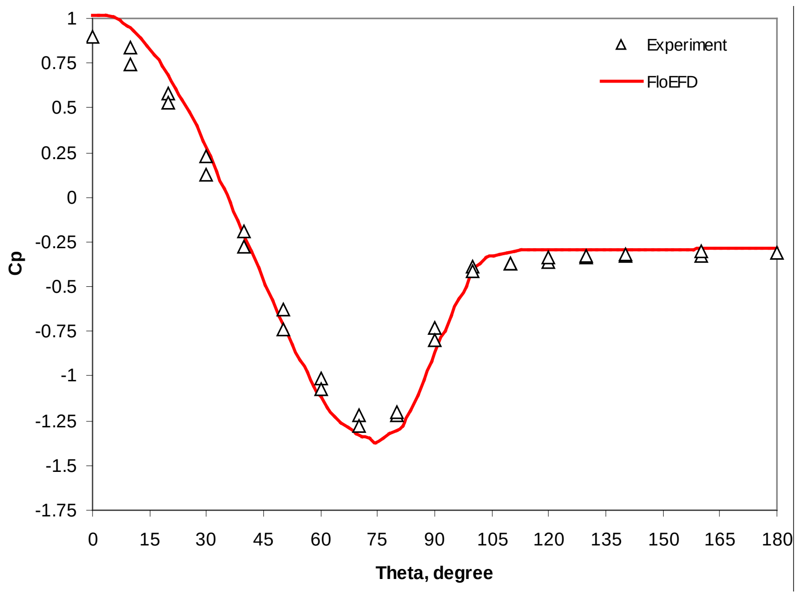

Figure 36: Local C_P distributions around cooling tower at elevation Z/H=0.79.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

As can be seen almost for all angles the calculation results demonstrate good agreement with the experiment.

Distribution of C_P with height in rear side of the model also shows good correlation with experimental data (see Fig. 37).

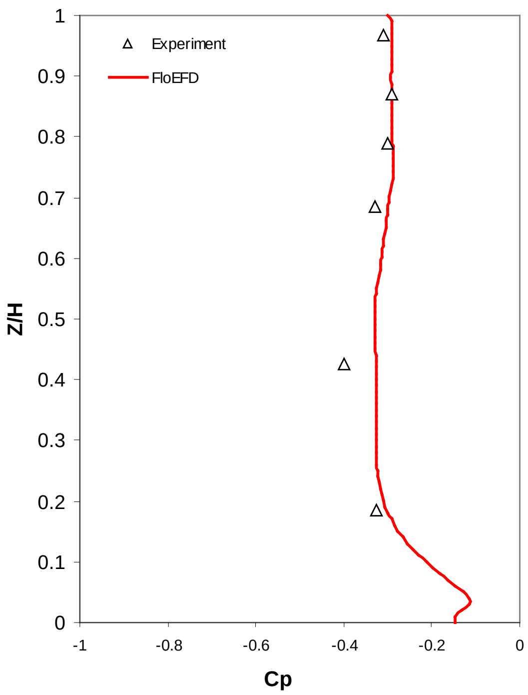

Figure 37: Local C_P distributions with height in rear side of the cooling tower (theta=180).

It should be pointed out very good FloEFD prediction of the positions and the values of maximum suction for all elevations under consideration.

As example of complex multiphysics calculations Figs. 38-39 display the result of prediction of the visible saturated vapour plume formation.

First of all, attention should be paid to excellent resolution of counter-rotating vortex pair (see Figs. 38) which is typical for turbulent buoyant jets in crossflow.

Secondly, 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 (see Fig. 39).

It can be stated here that FloEFD has been successfully validated on the problem of prediction of cooling tower external aerodynamics.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

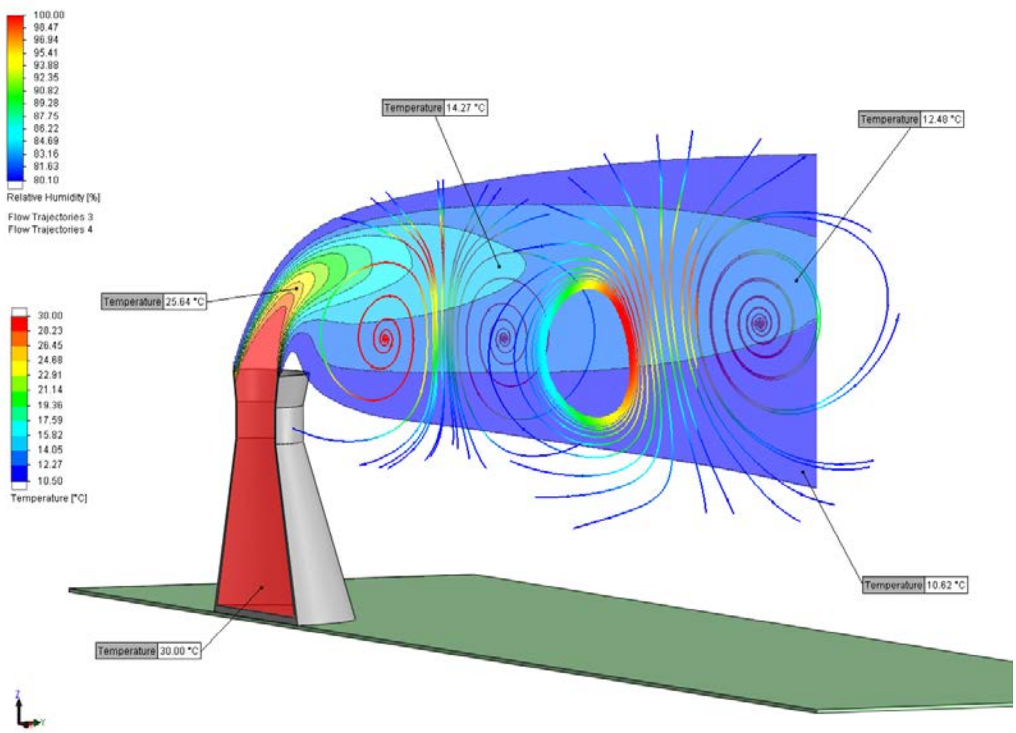

Figure 38: Temperature distribution in vertical symmetry plane along with flow trajectories drawn in two lateral downstream sections and colored by relative humidity magnitude.

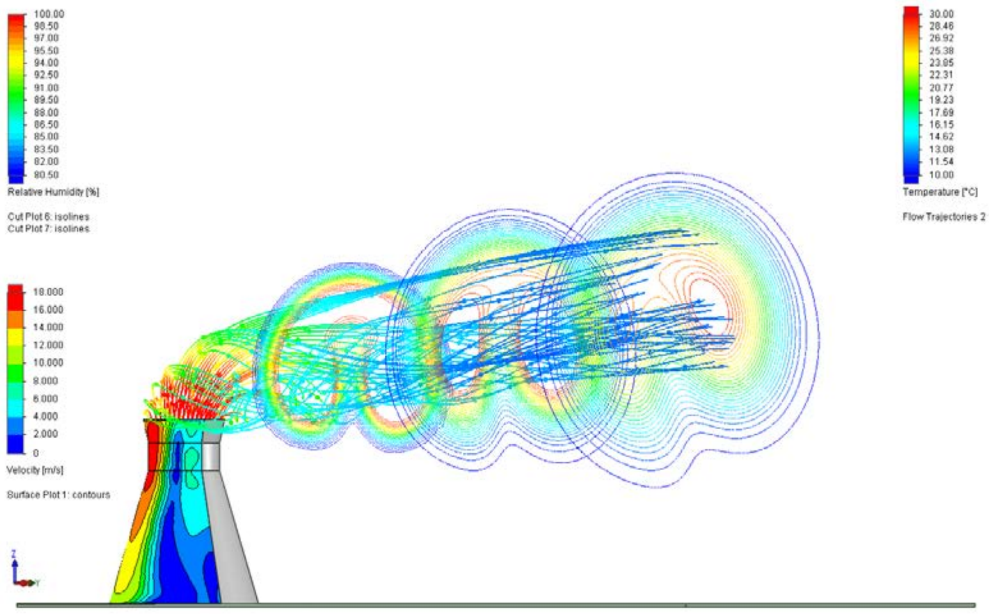

Figure 39: Velocity distribution on cooling tower shell along with flow trajectories colored by temperature magnitude and relative humidity contours in three downstream cross-sections.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

11. Industrial problems and benchmarks: prediction of cyclone performance at extreme temperature

Gas cyclones are the most widely used separation devices which can be found in industry.

Overall view of the cyclone considered for Validation purposes is presented in Fig. 40. The cyclone was defined by the geometrical conditions given in Table 5. All presented parameters as well as experimental data were taken from Lorenz (1994).

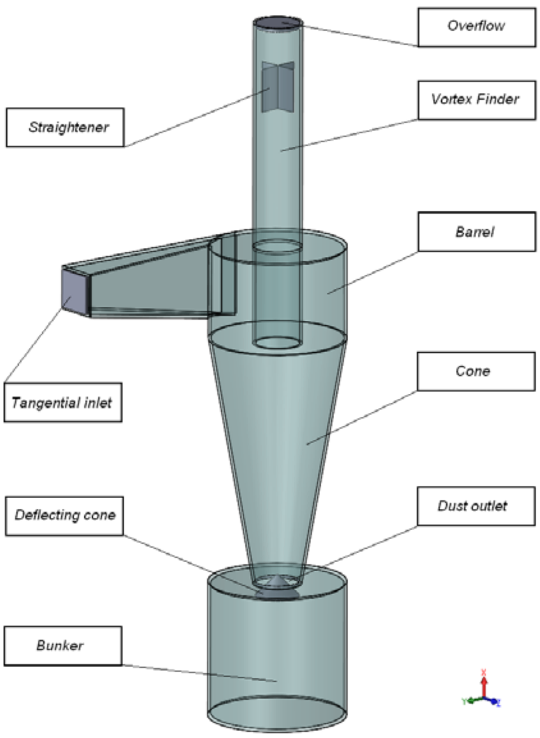

Figure 40: Overall view of the cyclone model.

FloEFD calculations were performed with a computational mesh of 350000 cells.

Transient approach was adopted for simulations. Time step Δt_c can be given in general form:

$$\Delta t_c = \frac{\min(D_d, D_{vf})}{U_{\max}} = \frac{\min(D_d, D_{vf})}{U_{inlet} D_{bar}} D_{vf} , \quad (1)$$

where D_d – dust outlet diameter, D_{vf} – vortex finder diameter, D_{bar} – barrel diameter, U_{inlet} – velocity at cyclone inlet.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

Geometric dimensions	Units	Value
Barrel diameter	m	0.15
Vortex finder diameter	m	0.05
Dust outlet diameter	m	0.05
Overall cyclone height	m	0.387
Inlet duct length	m	0.245
Entrance height	m	0.02
Entrance width	m	0.08
Barrel height	m	0.104
Vortex finder lower length	m	0.11
Vortex finder upper length	m	0.21
Straightener height	m	0.05
Inlet square side length	m	0.044
Gap between deflecting cone and dust outlet	m	0.01
Cone slope angle	deg.	10

Table 5: Main geometric dimensions of the cyclone model.

The results of calculations are shown in Figs. 41-43.

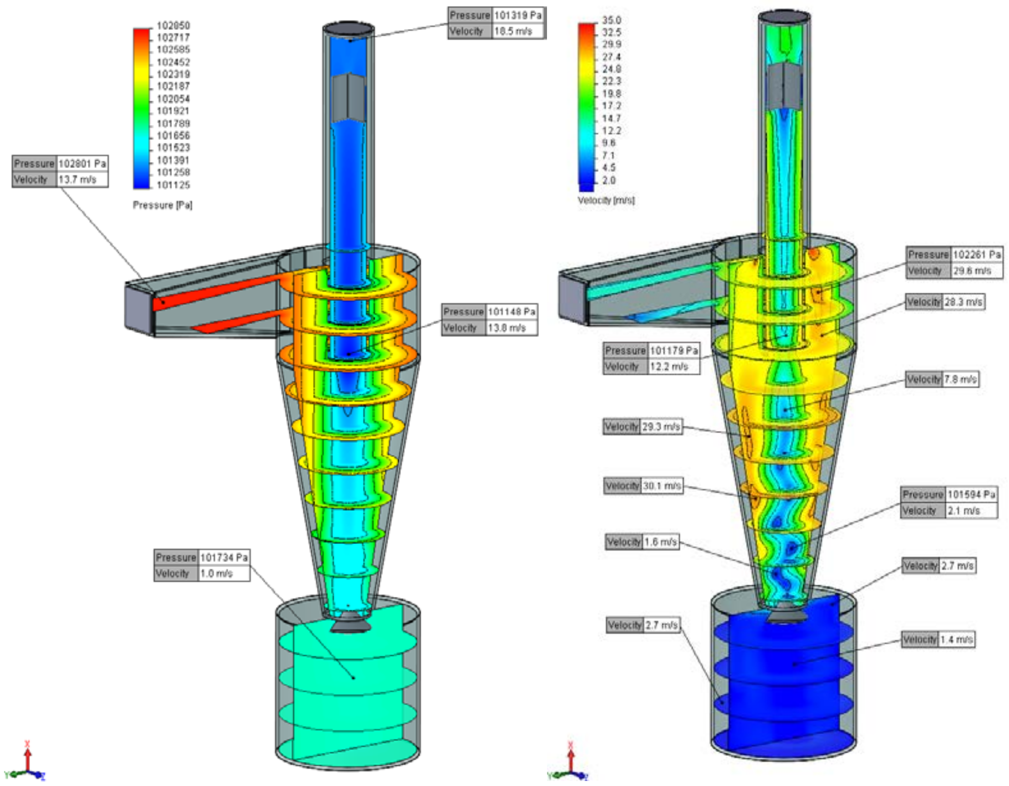

Figure 41: Pressure (left) and velocity (right) distributions within the cyclone for ambient air (20^0C) under volume flow rate of 80 m^3/h after simulation of 3 s of physical time.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

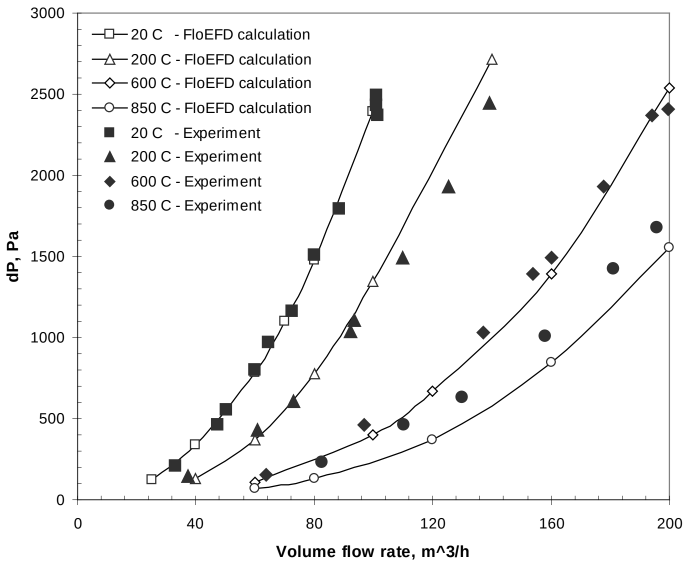

Figure 42: Pressure drops of the cyclone under various temperatures.

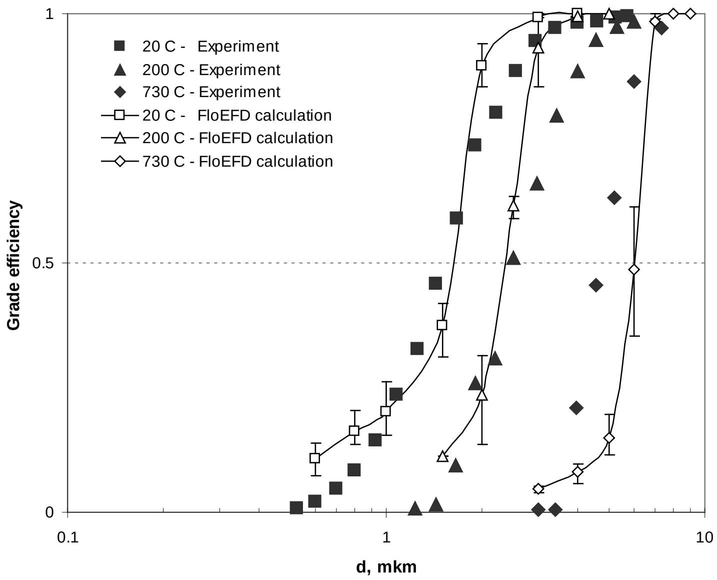

Figure 43: Grade efficiency curves under volume flow rate of 60 m^3/h and various air temperatures.

The flow field within the cyclone is presented in Fig. 41. Typical pressure and velocity distribution can be found there.

Fig. 42 shows the predicted pressure drop compared to the experimental data for different gas temperatures taken from Lorenz (1994). It demonstrates good agreement with the experiments for the most operating conditions. The differences between calculations and experiments are typically within 5-10%. Only for hot gas flow the difference gets a bit higher.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

The FloEFD predictions of cyclone grade efficiency operating from ambient to extreme temperature are shown in Fig. 43. Vertical bars at predicted values denote maximum and minimum removal probabilities obtained in 5 calculation series. The particle density was 2650 kg/m3.

One can see the FloEFD predictions of cyclone grade efficiency are in good agreement with reported data (Lorenz, 1994). Special attention should be paid for cut-off size (particle size under which 50% probability of particle removal is achieved) excellent prediction.

<u>12. Industrial problems and benchmarks: FloEFD simulation of micro-turbine engine</u>

A micro-turbine engine chosen for the study is KJ 66 (Fig. 44), which is one of the most robust small engines with available design.

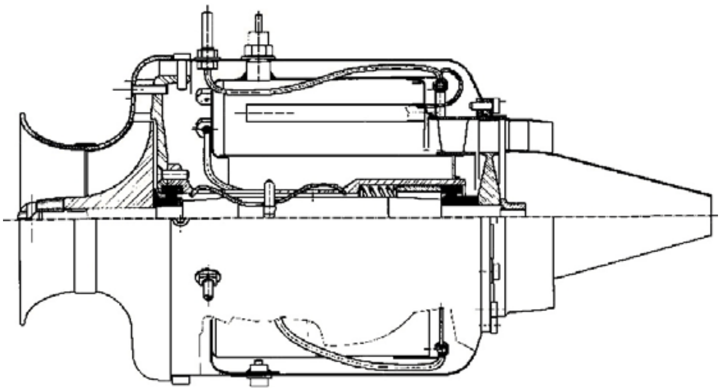

Figure 44: The scheme of KJ 66 micro-turbine engine.

The specification of KJ 66 can be found in Kamps (2005). The model of the engine was built in SolidWorks CAD system and demonstrated in Fig. 45.

Figure 45: The model of KJ 66 engine in FloEFD.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

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.

In Fig. 46 air mass flow at the inlet of the engine at various rotational speeds of the compressor can be seen.

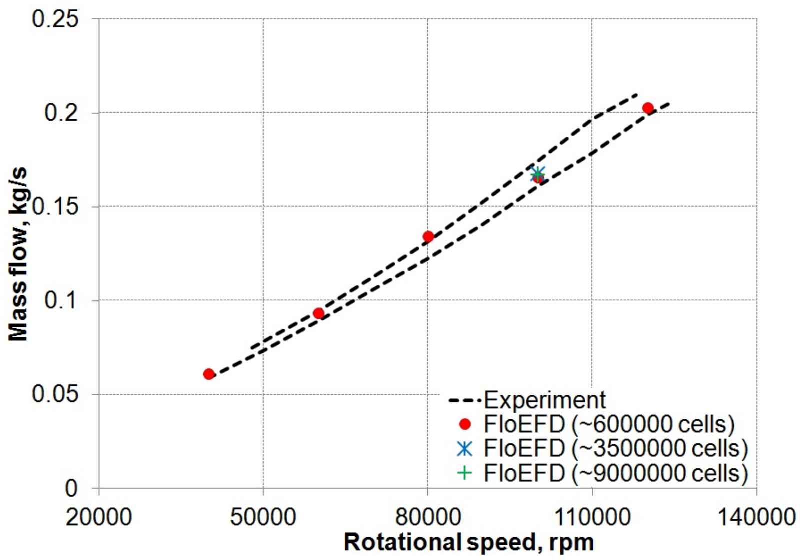

Figure 46: Air mass flow at the inlet of KJ 66 engine.

The FloEFD results are compared with experimental data from Kamps (2005). The values of mass flow match good experimental data and almost do not depend on cells' number. Thereby mesh of ~600000 cells is enough for definition almost all integral parameters here.

Fig. 47 displays fluid temperature and velocity distributions within the engine. Temperature in the combustion chamber reaches ~2400 K.

Pressure distribution on surfaces of the engine is presented in Fig. 48.

Fig. 49 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.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

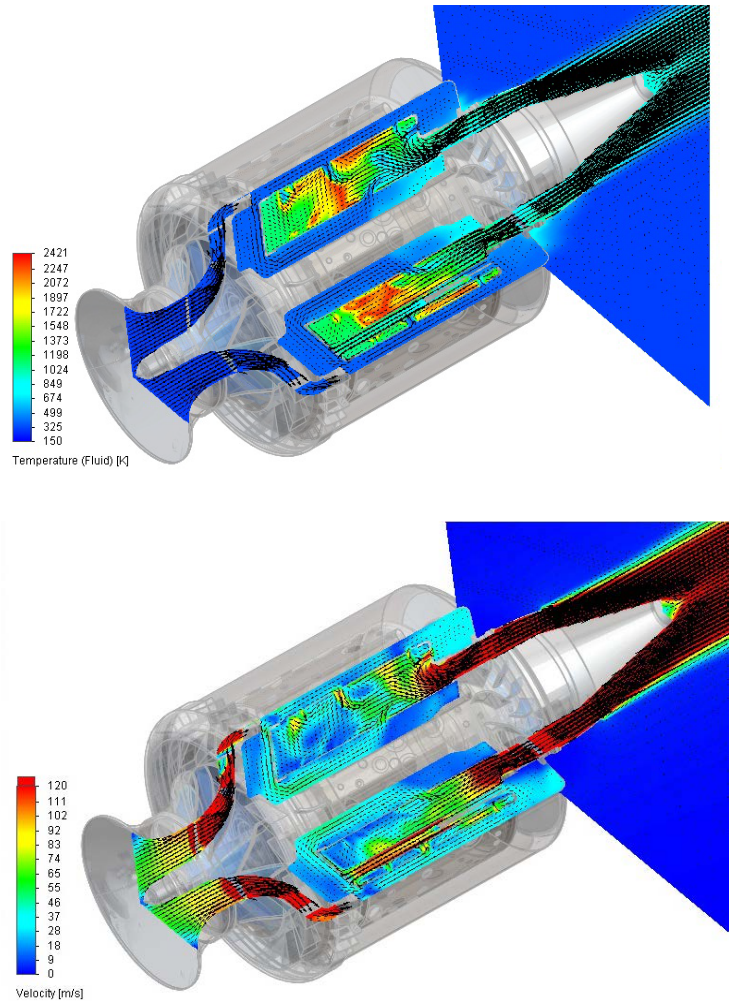

Figure 47: Fluid temperature (upper) and velocity (lower) distributions at two longitudinal sections of the combustion chamber with flow vectors at the normal mode.

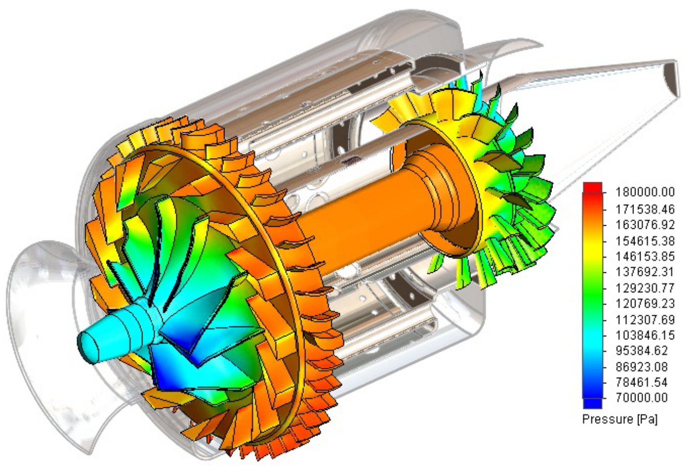

Figure 48: Pressure surface distributions in the engine.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

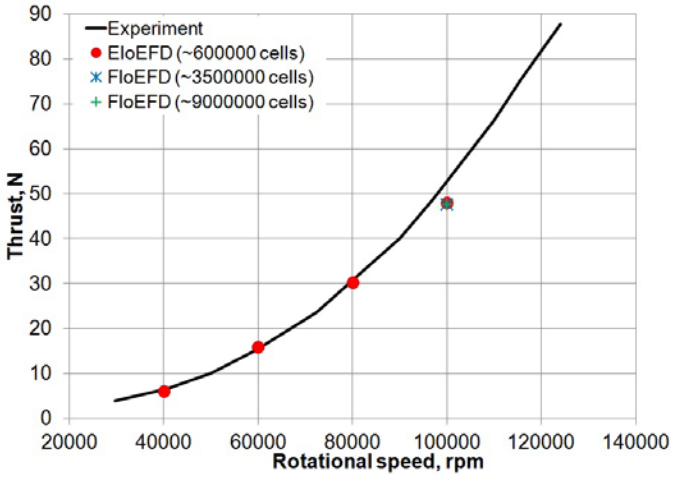

Figure 49: Thrust of KJ 66 engine.

4: CONCLUSIONS

Trend analysis on the worldwide CAE market clearly shows steady growth of market share of CFD calculations in the solution of today's engineering design problems. Formerly CFD calculations were mainly used in aerospace, automotive, power generation and electronic industries, but now such calculations are vitally important in almost for all industries. FloEFD is a typical example of the adaptation of CAE technology (namely fluid dynamics and heat transfer) for the everyday needs of design engineers.

For a code used by project-oriented engineers, it is actually impossible to separate the Verification and Validation procedures for most cases because of the high level of automation built into the code. This means that the activity of code Verification/Validation almost form a continuum, with the terms being used together when referring to a suite of activities and even abbreviated to V&V as an acronym for this.

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

A four-level classification of validation examples and tests is employed in current practice for the V&V procedures used in the QA of FloEFD. This can be portrayed graphically with the four levels displayed on an inverted pyramid, with each level being based upon, and supported by the previous level.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

In general, the categorization of validation examples and test cases within these levels of classification depends on example complexity, availability of reference data and its accuracy, and so on. As the levels progress in geometric and flow complexity, a tendency for decreasing availability and accuracy of experimental data is observed. This four-level classification has dynamic structure. As the FloEFD code is developed, the V&V activity, and particularly the development of new cases, is shifted more towards higher levels.

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.

Cook, P., McDonald, M. and Firmin, M. (1979) ‘Airfoil RAE 2822 — Pressure Distributions, and Boundary Layer Wake Measurements’, AGARD AR-138, p. A6.

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 .

Davis, G. de Vahl (1993) ‘Natural Convection of Air in a Square Cavity: a Bench Mark Numerical Solution’, *Int. J. Numer. Methods Fluids*, Vol. 3, No. 3, pp 249-264.

Denham, M.K. and Patrick, M.A. (1974) ‘Laminar Flow over a Downstream-Facing Step in a Two-Dimensional Flow Channel’, *Trans. Instn. Chem. Engrs.*, Vol. 52, pp. 361-367.

Emery, A. and Chu, T.Y. (1965) ‘Heat Transfer across Vertical Layers’, *Trans. ASME, J. Heat Transfer*, Vol. 87, p 110.

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

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

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

Holman, J.P. (1997) *Heat Transfer,* 8th ed., McGraw-Hill, New York.

Humphrey, J.A.C., Taylor, A.M.K. and Whitelaw, J.H. (1977) ‘Laminar Flow in a Square Duct of Strong Curvature’, *J. Fluid Mech.*, Vol.83, part 3, pp.509-527.

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

Lienhart, H., Stoots, C. and Becker, S. (2000) ‘Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)’, DGLR Fach Symp. der AG STAB, Stuttgart University.

Lorenz, T. (1994) ’Heisgasentstaubung mit Zyklonen’, VDI-Fortschritt-berichte, Reihe 3, Verfahrenstechnik № 366, VDI-Verlag, Dusseldorf .

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.

Missile Defense Agency (2008), *Validation & Accreditation (VV&A) for Models and Simulations*, Department of Defense Documentation of Verification, Missile Defense Agency.

Nandula, S.P., Pitz, R.W., Barlow, R.S. and Fiechtner, G.J. (1996) ‘Rayleigh/Raman/LIF Measurements in a Turbulent Lean Premixed Combustor’, AIAA Paper 96-0937, 34th Aerospace Sciences Meeting & Exhibit, Reno, NV, January 15-18, 1996.

Oberkampf, W.L. and Trucano, T.G. (2002) ‘Verification and Validation in Computational Fluid Dynamics’, *Progress in Aerospace Sciences*, Vol. 38, pp 209-272.

Panton, R.L. (1996) *Incompressible Flow*, 2nd ed., Wiley, New York.

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

Schlichting, H. (1979) *Boundary Layer Theory*, 7th ed., McGraw –Hill, New York.

Stern, F., Wilson, R.V., Coleman, H.W. and Paterson, E.G. (1999) ‘Verification and Validation of CFD Simulations’, IIHR Report No. 407, Iowa Inst. Hydraulic Research, the University of Iowa.

VALIDATION METHODOLOGY FOR MODERN CAD-EMBEDDED CFD CODE: FROM FUNDAMENTAL TESTS TO INDUSTRIAL BENCHMARKS

Van Dyke, M. (1982) *An Album of Fluid Motion*, The Parabolic Press, Stanford, CA.

White, F.M. (1994) *Fluid Mechanics*, 3rd ed., McGraw-Hill, New York.

Zdravkovich, M.M (2003) *Flow Around Circular Cylinders, Vol. 2: Applications*, Oxford University Press, New York.

