Dimensionless Comparison of Different RANS based CFD Methods

György Bicsák*

** Department of Aeronautics, Naval Architecture and Railway Vehicles,*
Budapest University of Technology and Economics, Műegyetem rkp. 3., H-1111, Budapest Hungary
email: gybicsak@vrht.bme.hu

Abstract: The design of airflows and structures related with jet engines, turboprops, helicopter rotors and other turbomachinery related configurations is complex and cost demanding process, especially for having accurate performances and aerodynamic parameters, meanwhile the applications of the proper material properties for the structural integrity are expected. The design and analyses in spatially distributed manner gets more highlight today due to its effectivity not only in the transportation as aviation, ships and marine propulsion, but in the other segments of the industry. The Reynolds Averaged Navier-Stokes (RANS) based Computational Fluid Dynamics (CFD) simulations are one of the most prominently growing part of the fluid flow related investigations. Since it is not an exact science, numerous methods are available for the same problem, with different computational effort requirements. For small aerospace companies it is essential to select the most suitable method, which is worthy for their limited resources. So, in this paper the *CFD Worthiness Number* has been introduced as a dimensionless quantity to compare the requirements of different RANS based CFD methods.

1. 1. INTRODUCTION

Recently the CFD (Computational Fluid Dynamics) is one of the most dynamically improving and frequently used CAE (Computer-Aided Engineering) methodology in the field of design and developments, because it has excellent capabilities in the virtual reality, and represents a good alternative of the highly cost, capacity and time demanding experimental tests in laboratories, wind tunnels and field. Due to the wide range of visualization techniques, easy and fast regeneration, reproducibility, high level parameterization, field and geometric model independence together with the possibilities of coupling more physics and optimization methods together, the simulation techniques often are not just a more cost-efficient solution than experimental measurements, but even provide more details and information about the given problem.

However, the behaviour and main conservation equations of fluid dynamics were laid down more than 100 years ago, and untill 1930 due to the lack of computers the Computational Fluid Mechanics (CFD) was only theoretical. Basically, most of CFD problems are oriented around the Navier-Stokes equations, to define a single-phase fluid flow problem (which can be gas or liquid, but not both). Of course, there are several possibilities to simplify the equations, like removing the viscous terms (in this case we are dealing with the Euler equations), or removing the vorticity-describing terms (full potential equations), or small perturbations in subsonic and supersonic flows (not transonic or hypersonic), which stands for the linearized potential equations. This last case is widely used withinin simple flow conditions applied in aerospace sciences and in simple investigations by using complex potentials.

At first, the linearized potential equations have been developed: 2-dimensional methods, are suitable for investigating the flowfield around a cylinder or an airfoil by solving linearized potential equations. (Milne-Thomson, 2011) However, the actual work was not a success, the first publication of the modern CFD belonged to Richardson (Richardson, 1922), which had been used in the 1940s by the ENIAC. (Hunt, 1998) The real progression in the development of CFD started together with continuously growing computer power, and thus 3 dimensional methods have become available. The main driving force behind this progression was the Los Alamos National Lab (Harlow, 2004), and probably this was the first performed simulation using the Navier-Stokes equations. The T3 group has developed numerous numerical methods, like vorticity-stream function method (Fromm and Harlow, 1963), or the fluid-in-cell method (Gentry et al., 1966) to simulate a wide variety of problems.

Hess and Smith published the first paper with a 3-dimensional model (Hess and Smith, 1967) giving place to the so-called Panel Methods. This simplified method was mainly used in aircraft fuselages and ship hulls since they did not include the lifting flows. This was followed by several methods, developed by the large aircraft manufacturers, like Boeing, Douglas, Lockheed, or the NASA; for example: (Carmichael and Erickson, 1981, Hess and Friedman, 1983, Youngren et al., 1983, Ashby et al., 1991).The advantage of these panel methods is that they have relevantly lower

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

computation requirement, and can provide relatively accurate results, can be applied for airfoil investigations (completed with boundary layer analysis), submarines, automobiles, helicopters, wind turbines, but their applicability is strongly limited. This is the way how program codes like USAREO or XFOIL were born. A more advanced step is the codes that are used Transonic Small Disturbance equations, like the WIBCO code, developed by Boppe of Grumman Aircraft. (Boppe, 1977) Later, to calculate the non-linear flow conditions at transonic speeds, the Full Potential codes started to arise, like Program H. (Bauer et al., 1972)

The next level of flow simulations is solving the Euler equations, which is capable of solving more accurately transonic flows. The Euler equations are a set of quasilinear hyperbolic equations governing adiabatic and inviscid flow, and can be applied to general, but not too complex simulations(Veress, 2004). Numerous codes have been developed to solve these equations, like the Lockheed's SPLITFLOW (Karman, 1995), MGAERO (Tidd et al., 1991), or MSES (Drela, 1990).

But of course, the final target was/is to solve the Navier-Stokes equations in 3-dimensional models, for which purpose several commercial packages have been developed, like ANSYS, OVERFLOW, Abacus CFD, OpenFoam. (Beneda, 2012)

All the flow types generate different criteria for the mathematical and physical models. In the simplest case when a laminar flow is considered, the solution of the Navier-Stokes (N-S) equations is easy. Unfortunately, the laminar flows represent only a small portion of real engineering problems. However, it is possible to directly integrate the N-S equations, the time- and length-scales have to be chosen with care. The Direct Numerical Simulation (DNS) is based on the momentary variables, without applying any turbulence models. The turbulent structures can vary between the smallest dissipative scales (Kolmogorov microscales), to the integral scale, which can be associated with the motions containing most of the kinetic energy. (Spalart, 1988) Consequently, the whole applied spatial and temporal scales have to be resolved, which demands extreme computing capacity. However, the DNS is considered a highly cost-demanding process, and one can state that it will arise only later, but the incredible acceleration of computing technology (like 19 years ago the optimal node number has been maximized in a few ten-thousand (Wilcox, 1998)), numerous studies have shown that the application of DNS is an actual project, of course in possession of the required computation capacity. (Ducoin et al., 2017, Oguic et al., 2016, Kang, 2016, Dairay and Vassilicos, 2016, Wu et al., 2017)

In order to decrease the required memory, different considerations of turbulence have been investigated during the last few decades. As the grid size increases, the turbulent behaviour of the fluid has to be modelled, and depending on the goal of the simulations privilege the more intensive eddies, which contain more energy, while the smaller eddies (which are universal in character) are approximated. The Large Eddy Simulation (LES) is based on the Sub-Grid Scale (SGS) model. (Germano et al., 1991) The first employment of the LES belongs to Smagorinsky for meteorology models. (Smagorinsky, 1963). The first engineering application was presented by Deardoff, which was a turbulent channel flow. (Deardorff, 1970) The method was developed and improved by Schumann for plane channels and annuli. (Schumann, 1975)

The solution of the 3-dimensional governing equations is time-dependent, and relevantly cheaper than DNS. The number of cells, required to resolve the outer layer is proportional to $Re^{0.4}$. (Chapman, 1979) In the viscous sublayer the resolution has to be increased to $Re^{1.8}$ (DNS requires $Re^{9/4}$). Therefore, LES can be applied at Reynolds numbers one order of magnitude higher than DNS. (Blazek, 2015b) Furthermore, the requirement on cell resolution can be decreased by using approximate wall models. According to this, nowadays LES is becoming more and more widespread, for a wide spectrum of applications. (Cinnella and Content, 2016, Amin Allah and Shafiei Mayam, 2017, Kempe and Hantsch, 2017, Brar and Elsayed, 2017, Egerer et al., 2016)

However, numerous projects are running (Rohacs and Rohacs, 2016), which apply DNS or LES, but for companies with limited resources and budgets, these are still not available, particularly for early stage development processes. The Reynolds Averaged Navier-Stokes (RANS) equations are still dominating in most of the ongoing or freshly finished projects (ESPOSA (ESPOSA, 2016)). By averaging the N-S equations the disturbances, caused by the turbulent fluctuations, are eliminated from the simulation, but the time-dependent mean parameters are kept. The generated equations are similar like the laminar flow's N-S equations, but it is completed by the Reynolds stress terms, which represent the non-linear terms in the equation system. To overcome the vagueness of the equation system, turbulence model has to be applied, see discussed later.

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

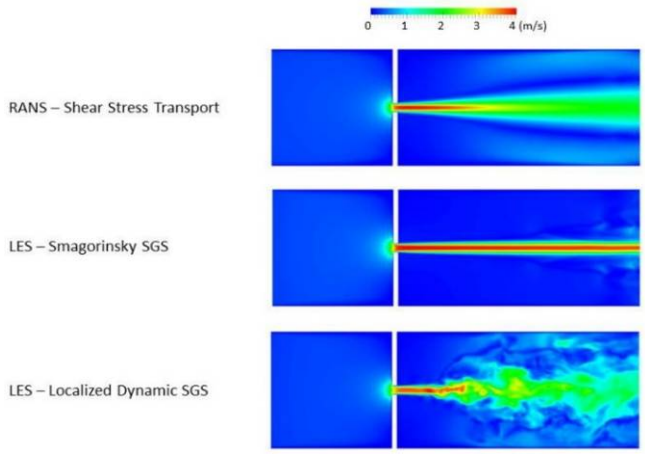

Fig. 1: The effect of different turbulent models on the flow-pattern (2014)

2. NAVIER-STOKES EQUATIONS

For the biggest percent of engineering problems a continuum physics approach can be used, as the Knudsen numbers stays below 0.01. Resolving of the complex unsteady turbulent flows with accompanying fluctuations over the entire or certain time and length scales does not provide additional information about the characteristics (as pressure drop, flow non-uniformities and flow forces) to be determined. Hence, Reynolds Averaged Navier-Stokes (RANS) approach can used instead of the computationally costly Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) methods.

The idea of RANS is relatively simple: the general form of conservation laws can be collected together by applying the general conservation law for a vector quantity. The Navier-Stokes equation introduces the vector of convective fluxes ($\underline{F_c}$) and vector of viscous fluxes ($\underline{F_v}$), from which the first consists the convective transport of quantities but also includes pressure terms (in the momentum and energy equations). The latter one comprises the viscous stresses and heat diffusion. The body forces and volumetric heating are colligated in the $\underline{Q}$ source term. For a Newtonian fluid the basic Navier-Stokes equation can be written in the following form: (Veress, 2004)

$$\frac{\partial}{\partial t}\int_V \underline{W}d\mathbf{v} + \oint_{\partial V}(\underline{F_c} - \underline{F_v})dA = \oint_V \underline{Q}dV\,. \qquad \textbf{(1.)}$$

The first undefined component of the equation is the ‘conservative variables’, and contains the next five components: (Blazek, 2015a)

$$\underline{W} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{bmatrix}. \qquad \textbf{(2.)}$$

The vector of the convective fluxes includes the following: (Blazek, 2015a)

$$\underline{F_c} = \begin{bmatrix} \rho V \\ \rho u V + n_x p \\ \rho v V + n_y p \\ \rho w V + n_z p \\ \rho H V \end{bmatrix}, \qquad \textbf{(3.)}$$

where the contravariant velocity V occurs. This quantity is the scalar product of the velocity vector, and the unit normal vector, and is normal to the surface element dA. The definition is: (Blazek, 2015a)

$$V \equiv \underline{v}\cdot\underline{n} = n_x u + n_y v + n_z w\,. \qquad \textbf{(4.)}$$

The viscous fluxes in a vector has the next form: (Blazek, 2015a)

$$\underline{F_v} = \begin{bmatrix} \mathbf{0} \\ n_x\tau_{xx} + n_y\tau_{xy} + n_z\tau_{xz} \\ n_x\tau_{yx} + n_y\tau_{yy} + n_z\tau_{yz} \\ n_x\tau_{zx} + n_y\tau_{zy} + n_z\tau_{zz} \\ n_x\Theta_x + n_y\Theta_y + n_z\Theta_z \end{bmatrix}, \qquad \textbf{(5.)}$$

where

$$\begin{aligned} \Theta_x &= u\tau_{xx} + v\tau_{xy} + w\tau_{xz} + \lambda\frac{\partial T}{\partial x}, \\ \Theta_y &= u\tau_{yx} + v\tau_{yy} + w\tau_{yz} + \lambda\frac{\partial T}{\partial y}, \\ \Theta_z &= u\tau_{zx} + v\tau_{zy} + w\tau_{zz} + \lambda\frac{\partial T}{\partial z}. \end{aligned} \qquad \textbf{(6.)}$$

These terms are responsible for handling and describe the heat conduction and viscous stresses in the fluid. Finally, the source terms can be written as: (Blazek, 2015a)

$$\underline{F_c} = \begin{bmatrix} \mathbf{0} \\ \rho f_{e,x} \\ \rho f_{e,y} \\ \rho f_{e,z} \\ \rho\underline{f_e}\cdot\underline{v} + \dot{q}_h \end{bmatrix}. \qquad \textbf{(7.)}$$

These equations together are called Navier-Stokes (N-S) equations in the integral formulation and describe the flux of mass, momentum and energy through the boundary dV of a control volume V, fixed in space. The set of equations can include special turbomachinery applications, like when the

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

control volume is rotating about an axis. As it will be discussed later, the N-S equations have to be transformed into a rotating frame of reference, and the Coriolis and centrifugal forces appear in the source term. (Hirsch, 1988) By applying additional terms, which include the relative motion of the dA surface element, the equations can handle the deformation or translation of the control volume (Pulliam and Steger, 1988), furthermore the Geometric Conservation Law has to be fulfilled.(Farhat and Guillard, 1999)

The N-S equations include five conservative variables ($\rho, \rho u, \rho v, \rho w, \rho E$) in 3 dimensions, but the number of unknown variables is 7. Therefore, two additional equations are required to close the system of equations. These desired equations are supported by the thermodynamic relations between the state variables. (Blazek, 2015a)

To handle or predict chaotic movements of the fluid molecules, models have to be set up. The first approach by Reynolds was a further step by averaging the conservation equations to separate the mean flow field – if the local fluctuations and turbulent structures were integrated into mean quantities, they would be eliminated from the simulation. So by using the initial U scalar quantity, the averaging it decomposes to a mean $\overline{U}$ and a deviation from the mean u : (Veynante and Vervisch, 2013)

$$U = \overline{U} + u \text{ and } \overline{u} - 0. \qquad \textbf{(8.)}$$

The ergodic hypothesis states that in a homogeneous and stationary cases the three averaging forms are equivalent. So, after applying the Reynolds averaging on the R-S equations, the next relations can be obtained from the mass and momentum conservation, and are called the Reynolds Averaged Navier-Stokes equations (RANS):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \underline{U}) - 0, \qquad \textbf{(9.)}$$

$$\frac{\partial}{\partial t}(\rho \underline{U}) + \nabla \cdot (\rho \underline{U} \otimes \underline{U}) = \\ = -\nabla p + \nabla \cdot (\bar{\tau} - \rho \overline{u \otimes u}) + S_M, \qquad \textbf{(10.)}$$

$$\frac{\partial}{\partial t}(\rho H) - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \underline{U} H) = \\ = \nabla \cdot (K \nabla T - \rho \overline{uh}) + \nabla \cdot \left(\underline{U} \cdot (\bar{\tau} - \rho \overline{u \otimes u}) \right) + S_E, \qquad \textbf{(11.)}$$

Here τ denotes the molecular stress tensor. It can be seen, that the continuity equation has not been altered, but additionally to the molecular diffusive fluxes, turbulent flux terms are contained by the momentum and scalar conservation equations. These terms are the Reynolds stresses ($\rho \overline{u \otimes u}$) and Reynolds flux. The non-linear convective term in the un-averaged equations provides these terms and proves the initial state in this chapter as due to the turbulent velocity fluctuations the convective transport increases the mixing level above the thermal fluctuations caused mixing at molecular level. Consequently, at high Reynolds numbers, the turbulent fluxes are much larger than the molecular fluxes (mean path of thermal fluctuations). In the energy equation, the supplemented diffusive flux is the viscous work term. Another consequence is that six additional relations are required to close equations (9.),(10.) and (11.).

The simplest way for the Reynolds stress approximation in the Reynolds or/and Favre averaged N-S equations is to apply the first-order closure. These methods are based on the Boussinesq or non-linear eddy-viscosity models and primarily compute the eddy viscosity. However, though a wide range of models is available in this group, mostly the k-ε model, k-ω model and their combination, the Shear Stress Transport model (SST) is applied to model the turbulence.

Furthermore, numerous models can be added to the general RANS equations, like reaction-kinetic equations, combustion models, thermal radiation, species transport etc., which means additional equations to be solved, and mostly treated as source terms in the conservation equations.

The main problem with the Navier-Stokes equations that in engineering applications the analytical solutions do not exist, maybe for the simplest flows under completely ideal conditions. To overcome this problem, numerical approaches have to be applied to obtain solutions for real flows, where algebraic approximations replace the equations, which can be solved by applying a numerical method. Of course, the computation of a smooth elliptic problem is a simple task recently. It takes three main steps, which can be defined as:

1. Selecting a **discretization scheme**: the three available methods are the finite element, finite volume or finite difference representations. In numerous fields, all three methods are equally efficient, but in CFD simulations mainly the finite volume methods are preferred.
2. To deal with the non-linearity of the density, an **iteration method** has to be chosen.
3. For the resolution of the obtained algebraic system, an **algorithm** has to be selected.

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

Of course, the full process consists the determination of boundary conditions between step 2. and 3.

3. DEFINING CFD WORTHINESS NUMBER

Comparison of different RANS based CFD methods is not an exact problem, since numerous variations are available to simulate the same flow conditions. Depending on what kind of method is used, the computational requirements and accuracy can vary in a wide range. A company has to be aware of the applied method's requirement from the viewpoint of engineering skills and knowledge, available manhours, used hardware, software and its features. Of course, these factors incorporates a cost (licenses, hardware, payments, involved outsider professionals, infrastructure) which eventually determines whether the selected method does worth to use it or not. If different methods were compared to each other both in accuracy and from the viewpoint of computational requirements, the need of an independent, numerical number would occurr, which can be fit to these kind of problems and indicates the worthiness of the method.

The *CFD Worthiness Number* ($W_{CFD,i}$) has been carried out, which incorporates the *mesh cell number* of the investigated model given in normal form, resolved to the significant part (N_m^s) and exponent (N_m^e), the iteration number (N_{it}), which is necessary to reach the convergence criteria, a *correction factor* (β_g) to take into account the quality of the mesh, the *processor performance factor* (ψ_p) and finally the *accuracy* (ϵ_i), which is given by the average relative error for the investigated model parameter (and π is only a weight in the equation). The CFD Worthiness Number is determined as follows:

$$W_{CFD,i} = \frac{N_m^e}{\psi_p} * ln\left(N_m^s * \beta_g * N_{it}\right) * e^{\pi\epsilon_i} \qquad (12.)$$

Neither of the parameters have dimensions, thus $W_{CFD,i}$ is dimensionless:

- $[N_m^e] = \emptyset$, e.g. if the cell number was 4.2*10^6, then $N_m^e = 6$,
- $[N_m^s] = \emptyset$, e.g. if the cell number was 4.2*10^6, then $N_m^s = 4.2$,
- $[N_{it}] = \emptyset$,
- $[\beta_g] = \emptyset$, see its possible values later,
- $[\psi_p] = \emptyset$, see the calculation method later,
- $[\epsilon_i] = \emptyset$; meaning 5% average relative error equals to 0.05.

As it can be seen, the value of $W_{CFD,i}$ can various values, it cannot reach zero, since the $\frac{N_m^e}{\psi_p}$ component is always greater than 0, just like the logarithmic and exponent components. The $W_{CFD,i}$ should be as low as it is possible, in theory it can vary in the interval of $[0; +\infty[$, but within the realistic boundaries, its values can change from 10 to 45. In an ideal case, when the simulation perfectly describes the reality, $W_{CFD,i}$ converges to 0, depending the cell number, processor performance factor and iteration number, since it would a reasonable consideration that if the accuracy reaches 100%, then it worth every effort, but actually this is far from the practice. That's why theoretical minimum and maximum values could be calculated, but practically the $W_{CFD,i} = 10 \div 20$ interval should be the basis, where the ideal state is 10, the worst state is 20. So, in this section the point is to determine a $W_{CFD,i}$ value, which is accurate enough, but the simulation does not require unreal efforts.

To settle this value, I carried out a sensitivity analysis to figure out, how every factor influences the worthiness number, but first let's see what the different components incorporate.

1. Mesh cell number (N_m) incorporates the following information about:
   - complexity of the geometry,
   - dimensions – size of the flowfield,
   - turbulence-influencing parameters, such as maximum length scale, Kolmogorov scale, $y+$, Reynolds number, boundary layers,
   - allocable memory size.

   The mesh cell number's normal form contain the significant part (N_m^s) and exponent (N_m^e), from which the significant part is in the logarithmic component, while the exponent part (which has the bigger role in the computational requirement) takes place outside as a multiplier.

   The two values van vary as: $N_m^s = 1.0 \div 9.\dot{9}$, while in practical solutions $N_m^e = 3 \div 12$. In the latter case, below 1000 cell elements there is no really point to investigate whether the given method worth it or not, while above 1000 trillion elements there is no point to apply the method (actually 1 trillion element, so $N_{m.max}^e = 9$ was completely enough for practical problems, 12 is only to establish the theory).

2. The processor performance factor (ψ_p) demonstrates:

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

- performance of the processor (frequency, core number),
- type and efficiency of solver - discretization and Solution of the governing equations,
- number of useable cores (paid license, number of cores).

The factor can be calculated as the following:

$$\psi_p = \omega_f * \sqrt{n_c} = \frac{\omega_p\ [MHz]}{3000\ [MHz]} * \sqrt{n_c} \qquad (13.)$$

ω_f is the processor's frequency ratio, which is a comparison of the applied computer's processor (ω_p) to a baseline 3000 MHz basic processor frequency; and n_c is the number of useable cores (the number of total cores can be reduced by the paid license, the solver's algorithms, overheat, etc.). The square of core numbers is taken, since the higher the core number, the more problematic to allocate them all, in this case the solver has to deal with the core-distribution management also.

So, if the company would own a super computer or cluster, however the cell number was still relevant, but the worthiness of the given process (by applying more useable cores) was significantly better. A computer with $\psi_p = 2$ 2 cores which can run at 3000 MHz , while a cluster with 500 allocable cores, which each can ran at 4500 MHz, $\psi_p = 750$. Recently available processors have: $\omega_p = 0.5 \div 1.8$, while n_c depends on the number of available computers, processors and paid licenses. This way ψ_p can vary between 1 and 50, values below and above are unrealistic.

3. Iteration Number (N_{it}) contains information about:

- convergence characteristics of residuals,
- convergence characteristics of imbalances,
- correctness of boundary conditions and initial boundary conditions,
- existence of perturbations or other phenomena (e.g.: rotating domain – non-rotating domain interface or additional sources).

The minimum and maximum iteration number practically varies between 50 and 50,000. Below 50 iteration steps there is no point to run a CFD simulations, because the imbalances and convergence criteria would probably not reached, while over 50,000 iteration steps we are talking about research type simulations, which has a lot of benefits, but for an industrial application this effort is not remunerative.

- Correction factor (β_g) takes into consideration the quality of the mesh (number of poor elements; type of cell elements – hexa or tetra)

The possible values of β_g is summarised in Table 1.

Table 1: Correction factor values (β_g) for different mesh types

Mesh quality	β_g [-]
Simple, hexa mesh	0.9
Complex, structured hexa mesh with boundary layers and minimal poor elements	0.95
Simple mesh made from tetra elements	1
Complex mesh, made from tetra cells, with minimal poor elements and boundary layers	1.1
Simple, hybrid mesh	1.05
Complex hybrid mesh with minimal poor elements and boundary layers	1.15

4. Accuracy – average relative error (ϵ_i) represent the correctness of:

- boundary conditions,
- applied models,
- input data (calculated and experimental data)
- applied simplifications

However, the accuracy is in the exponent of the equation (12.), thuswise it increases the $W_{CFD.i}$ value in magnitudes; it is recommended to apply a range for the argument. The accuracy should be: $\epsilon_i =]0; 0.05]$ – the 100% accurate model was the optimal solution, but the effort does not worth it, while the average relative error above 5% is not acceptable.

3.1. Sensitivity analysis

Although, $W_{CFD.i}$ does not have upper limit, because of the described practical limits, it cannot reach infinitely high values, or zero. In order to investigate how the number reacts

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

to the change of any of the parameters, a sensitivity analysis had to be completed. By production of the partial derivatives we get:

$$\frac{\partial W_{CFD,i}}{\partial N_m^e} = \frac{e^{\pi\epsilon_i}}{\psi_p} * ln\left(N_m^s * \beta_g * N_{it}\right) \quad (14.)$$

$$\frac{\partial W_{CFD,i}}{\partial N_m^s} = \frac{N_m^e * e^{\pi\epsilon_i}}{\psi_p} * \frac{1}{N_m^s} \quad (15.)$$

$$\frac{\partial W_{CFD,i}}{\partial N_{it}} = \frac{N_m^e * e^{\pi\epsilon_i}}{\psi_p} * \frac{1}{N_{it}} \quad (16.)$$

$$\frac{\partial W_{CFD,i}}{\partial \epsilon_i} = \frac{N_m^e}{\psi_p} * ln\left(N_m^s * \beta_g * N_{it}\right) * \pi e^{\pi\epsilon_i} \quad (17.)$$

With the defined limits, neither of these equations can be equal to zero, since there is no direct minimum point.

In equation (14.) N_m^e falls out in the derivation process, so this equation varies only depending on the values of the rest of the components. The minimum value (theoretically) is 0.31, while the maximum value is 3.81.

In equation (15.) and (16.) the derivatives have hyperbolic form, and regarding to the arguments of the respective parameters, they also have minimum and maximum values, but not global minimum or maximum locations. Equation (15.) changes from 0.0023 to 3, while equation (16.) varies between $4.68*10^{-7}$ and 0.06.

The most interesting parameter was to be the accuracy (equation (17.)), but since it was stated that it should not be worse than 5% average relative error for a given parameter, this equation is also limited between 0.98 and 35.88.

As a result, it can be stated, that the exponent of the mesh cell number, the number of allocable processor cores and accuracy have the most dominant effect on the CFD Worthiness Number, but the significant of mesh number, mesh quality, processors' operation frequency and iteration number is also relevant, and the carried developed dimensionless number is capable to demonstrate the resource-requirements of the investigated CFD method.

By selecting a simulation method, and in the possession of the geometry, this number can be estimated, because it consist only predictable parameters. If the estimated CFD Worthiness Number was significantly lower than 20, it can be concluded that the applied method is affordable for the given company. Above 20, however the selected method can be really accurate, the efforts required to run the simulation and get the results would be too high, and probably not affordable.

By going through several possibilities, where different scenarios have been compared to each other, I state that for a small company with limited resources both in manhours, in experience or knowledge, and in computational effort, for earlier stage development processes those CFD simulation processes should be applied, which has lower CFD Worthiness Number than 20:

$$W_{CFD} \leqq 20 \quad (18.)$$

See some simulation cases in **Hiba! A hivatkozási forrás nem található.**, carried out by the author in the last few years, how the CFD Worthiness Number varies:

Table 2: Examples for CFD Worthiness Number values based on former CFD simulations cases

	RDM	*ADMv1*	*ADMv2*	*ADMv3*	*Porous media*	*Detailed HX*	*Particle tracking*	*Idealistic case*
N_m	1,91E+07	1,34E+07	1,61E+07	1,61E+07	5,35E+05	4,60E+10	1,16E+07	1
N_m^e	7	7	7	7	5	10	7	1
N_m^s	1,9065034	1,3428609	1,6054875	1,6054875	5,34591	4,5974826	1,1557515	1
N_{it}	3500	200	200	200	50	1000	250	2
β_g	1,05	1,05	1,05	1,05	0,9	1	1,1	1
ϵ_i	0,045	0,041	0,04	0,03	0,016	0,03	0,05	0
n_c	4	4	4	4	4	4	4	500
ω_p	3600	3600	3600	3600	3600	3600	3600	5000
ψ_p	2,4	2,4	2,4	2,4	2,4	2,4	2,4	37,27
$W_{CFD,i}$	29,7	18,7	19,3	18,6	12,0	38,6	19,7	0,019

In the second column of **Hiba! A hivatkozási forrás nem található.**, RDM was a simulation where one side of a small aircraft has been investigated within the framework of the ESPOSA project. The rotor's effect has been simulated by applying rotating domain model. The detailed mesh had a strict boundary layer over every wall boundary, and because of the interface between the stationary and rotating domain, the solver required more iteration steps to reach the convergence criteria. Although, the simulation provided really accurate results, it took more than 3 weeks to complete

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

the simulation, which means that despite the accuracy this method is not suitable to be applied for an optimisation process.

The next three columns dealt with the same problem, but instead of simulating the fluid flow condition adjacent to the propeller blades, this part of the geometry has been removed, and by implementing Schmitz's method the Actuator Disk Method has been applied with different settings. In the case of ADMv1 the induced velocities has been simulated by completely removing the rotor domain and inserting boundary conditions. In the case of ADMv2 and ADMv3 the rotor domain has been kept and the results of Schmitz's method has been considered with momentum source terms. So, in the last two case more cell elements were required, but as soon as the accuracy dropped to 3%, the CFD Worthiness Number has become lower than in the case of ADMv1.

The 'Porous media' and 'Detailed HX' are an approximation to investigate the behaviour of a plate-fin type, oil-to air heat exchanger, particularly its effect to the external airflow around an aircraft engine nacelle. The 'Detailed HX' simulation consisted a detailed mesh, fitted to the small channels of the heat exchanger, with the proper boundary layers and mesh size regarding to the size scale of the fluid flow. The 'Porous media' handled the same problem, but instead of using an extremely high cell number, the heat exchanger has been treated as a porous media, which parameters have been determined with applying Darcy's law. In this case, the necessary mesh number was significantly lower. So, while the 'Detailed HX' model requires a high performance cluster and high computational time, the porous media can provide similarly accurate results and the simulation can run even on a notebook or ultrabook.

The 'Particle Tracking' simulation was created in order to determine the particle separation efficiency of an engine air intake device, which particle can be solid or liquid. A more detailed mesh was required, and because of the additional models (particle collision, drag, etc.) the simulation took longer to reach the converge criteria. That's why in this case the CFD Worthiness Number is close to 20, but it is still in the acceptable region.

Finally, as it was mentioned earlier, the CFD Worthiness Number theoretically can vary between 0 and ∞ , practically a lot of factors will limit its value. Like the last column shows: if a CFD "simulation" is run in one cell for 2 iteration steps (1 step is not an iteration), on 500 allocable CPU cores, which run at 5000 MHz (available maximum processor speed at the moment), the CFD Worthiness Number was 0.019. Of course, this situation is not lifelike, in most cases the CFD Worthiness Number varies between 10 and 45, to reach lower or higher values extreme simulation cases are required. The applicable region is below 20, where if it was closer lower than 15, the computational requirements are minimal (an notebook or ultrabook can handle the simulation), while if , it needs a stronger Personal Computer (PC). Over 20, the simulation needs a high performance hardware and also it has significant time demand. So, these problems should be taken into consideration in R&D problems, but for small aerospace companies during development it generally not worth to apply.

4. CONCLUSIONS

In the present paper after a short overview of the most recently used steady state CFD simulation theory, a CFD Worthiness number has been carried out in order to create an indicator to compare different kind of RANS simulation cases. This indicator incorporates the mesh cell number of the investigated model given in normal form, resolved to the significant part and exponent, the iteration number, which is necessary to reach the convergence criteria, a correction factor to take into account the quality of the mesh, the processor performance factor and finally the accuracy, which is given by the average relative error for the investigated model parameter (and π is only a weight in the equation).

By going through several possibilities, where different scenarios have been compared to each other, for a small company with limited resources both in manhours, in experience or knowledge, and in computational effort, for earlier stage development processes those CFD simulation processes should be applied, which has lower CFD Worthiness Number than 20.

5. ACCOMPLISHMENT

The investigation supports applications, like the " Small aircraft hybrid propulsion system development" supported by Hungarian national EFOP-3.6.1-16-2016-00014 project titled " Investigation and development of the disruptive technologies for e-mobility and their integration into the engineering education".

REFERENCES

2014. *Validation: Turbulent jet flow* [Online]. Exploratory Research Space RWTH Aachen. Available: http://www.enmodes.de/references/validation/validation-turbulence/ [Accessed 23.02. 2017].

AMIN ALLAH, V. & SHAFIEI MAYAM, M. H. 2017. Large Eddy Simulation of flow around a single and two in-line horizontal-axis wind turbines. *Energy,* 121**,** 533-544.

ASHBY, D. L., DUDLEY, M. R., IGUCHI, S. K., BROWNE, L. & KATZ, J. 1991. Potential Flow Theory and Operation Guide for the Panel Code PMARC.

BAUER, F., GARABEDIAN, P. & KORN, D. 1972. *A Theory of Supercritical Wing Sections, with*

Computer Programs and Examples, Springer-Verlag Berlin Heidelberg.

BENEDA, K. 2012. Numerical Solution of MEMS-based Blade Load Distribution Control in Centrifugal Compressor Surge Supression. *ICNPAA 2012 Congress.* American Institute of Physics, Conference Proceedings 1493.

BLAZEK, J. 2015a. Chapter 2 - Governing Equations. *Computational Fluid Dynamics: Principles and Applications (Third Edition).* Oxford: Butterworth-Heinemann.

BLAZEK, J. 2015b. Chapter 7 - Turbulence Modeling. *Computational Fluid Dynamics: Principles and Applications (Third Edition).* Oxford: Butterworth-Heinemann.

BOPPE, C. 1977. Calculation of transonic wing flows by grid embedding. *15th Aerospace Sciences Meeting.* American Institute of Aeronautics and Astronautics.

BRAR, L. S. & ELSAYED, K. 2017. Analysis and optimization of multi-inlet gas cyclones using large eddy simulation and artificial neural network. *Powder Technology,* 311**,** 465-483.

CARMICHAEL, R. & ERICKSON, L. 1981. PAN AIR - A higher order panel method for predicting subsonic or supersonic linear potential flows about arbitrary configurations. *14th Fluid and Plasma Dynamics Conference.* American Institute of Aeronautics and Astronautics.

CHAPMAN, D. R. 1979. Computational Aerodynamics Development and Outlook. *AIAA Journal,* 17**,** 1293-1313.

CINNELLA, P. & CONTENT, C. 2016. High-order implicit residual smoothing time scheme for direct and large eddy simulations of compressible flows. *Journal of Computational Physics,* 326**,** 1-29.

DAIRAY, T. & VASSILICOS, J. C. 2016. Direct numerical simulation of a turbulent wake: The non-equilibrium dissipation law. *International Journal of Heat and Fluid Flow,* 62, Part A**,** 68-74.

DEARDORFF, J. W. 1970. A Numerical Study of Three-Dimensional Turbulent Channel flow at Large Reynolds Numbers. *Journal of Fluid Mechanics,* 41**,** 45-480.

DRELA, M. 1990. Newton solution of coupled viscous/inviscid multielement airfoil flows. *21st Fluid Dynamics, Plasma Dynamics and Lasers Conference.* American Institute of Aeronautics and Astronautics.

DUCOIN, A., SHADLOO, M. S. & ROY, S. 2017. Direct Numerical Simulation of flow instabilities over Savonius style wind turbine blades. *Renewable Energy,* 105**,** 374-385.

EGERER, C. P., SCHMIDT, S. J., HICKEL, S. & ADAMS, N. A. 2016. Efficient implicit LES method for the simulation of turbulent cavitating flows. *Journal of Computational Physics,* 316**,** 453-469.

ESPOSA 2016. Periodic Report Summary - ESPOSA (Efficient Systems and Propulsion for Small Aircraft). *AAT.2011.4.4-4. - Integrated approach to efficient propulsion and related aircraft systems for small-size aircraft* Czech Republic.

FARHAT, C. & GUILLARD, H. On the Significance of the GCL for Flow Computations on Moving Meshes. 37th Aerospace Sciences Meeting and Exhibit, 1999 Reno, Nevada. AIAA Paper.

FROMM, J. E. & HARLOW, F. H. 1963. Numerical Solution of the Problem of Vortex Street Development. *Physics of Fluids,* 6.

GENTRY, R. A., MARTIN, R. E. & DALY, B. J. 1966. An Eulerian differencing method for unsteady compressible flow problems. *Journal of Computational Physics,* 1**,** 87-118.

GERMANO, M., PIOMELLI, U., MOIN, P. & CABOT, W. H. 1991. A Dynamic Subgrid-Scale Eddy Viscosity Model. *Physics of Fluids,* A3**,** 1760-1765.

HARLOW, F. H. 2004. Fluid dynamics in Group T-3 Los Alamos National Laboratory: (LA-UR-03-3852). *Journal of Computational Physics,* 195**,** 414-433.

HESS, J. & FRIEDMAN, D. 1983. Analysis of complex inlet configurations using a higher-order panel method. *Applied Aerodynamics Conference.* American Institute of Aeronautics and Astronautics.

HESS, J. L. & SMITH, A. M. O. 1967. Calculation of potential flow about arbitrary bodies. *Progress in Aerospace Sciences,* 8**,** 1-138.

HIRSCH, C. 1988. *Numerical Computation of Internal and External Flows*, John Wiley & Sons.

HUNT, J. C. R. 1998. Lewis Fry Richardson and his Contributions to Mathematics, Meteorology, and Models of Conflict. *Annual Review of Fluid Mechanics,* 30.

KANG, S. H. 2016. Direct numerical simulation of the turbulent premixed flame propagation with radiation effects. *International Journal of Heat and Mass Transfer,* 102**,** 323-330.

KARMAN, L. J. S. 1995. SPLITFLOW - A 3D unstructured Cartesian/prismatic grid CFD code for complex geometries. *33rd Aerospace Sciences Meeting and Exhibit.* American Institute of Aeronautics and Astronautics.

KEMPE, T. & HANTSCH, A. 2017. Large-eddy simulation of indoor air flow using an efficient finite-volume method. *Building and Environment,* 115**,** 291-305.

MILNE-THOMSON, L. M. 2011. *Theoretical Aerodynamics*, Dover Publications.

OGUIC, R., PONCET, S. & VIAZZO, S. 2016. High-order direct numerical simulations of a turbulent round impinging jet onto a rotating heated disk in a highly confined cavity. *International Journal of Heat and Fluid Flow,* 61, Part B**,** 366-378.

PULLIAM, T. H. & STEGER, J. L. 1988. Recent Improvements in Efficiency, Accuracy, and Convergence for Implicit Approximate Factorization

Online: ISBN 978-963-88875-3-5
CD: ISBN 978-963-88875-2-8

Algorithms. *AlAA 23rd Aerospace Sciences Meeting.* Reno, Nevada.

RICHARDSON, L. F. 1922. *Weather Prediction by Numerical Process,* Cambridge, The University press.

ROHACS, D. & ROHACS, J. 2016. Magnetic levitation assisted aircraft take-off and landing (feasibility study – GABRIEL concept). *Progress in Aerospace Sciences,* 85**,** 33-50.

SCHUMANN, U. 1975. Subgrid-Scale Model for Finite-Difference Simulations of Turbulent Flows in Plane Channels and Annuli. *Journal of Computational Physics,* 18**,** 376-404.

SMAGORINSKY, J. 1963. General Circulation Experiments with the Primitive Equations. *Monthly Weather Review,* 91**,** 99-165.

SPALART, P. R. 1988. Direct Numerical Simulation of a Turbulent Boundary Layer up to Re=1410. *Journal of Fluid Mechanics,* 187**,** 61-98.

TIDD, D., STRASH, D., EPSTEIN, B., LUNTZ, A., NACHSHON, A. & RUBIN, T. 1991. Application of an efficient 3-D multigrid Euler method (MGAERO) to complete aircraft configurations. *9th Applied Aerodynamics Conference.* American Institute of Aeronautics and Astronautics.

VERESS, Á. 2004. *Numerikus módszerek és alkalmazások a hő- és áramlástechnikai gépekben lezajló folyamatok modellezésére.* PhD, University of Technology and Economics.

VEYNANTE, D. & VERVISCH, L. 2013. *Turbulent Combustion Modeling*, von Karman Institute for Fluid Dynamics.

WILCOX, D. C. 1998. *Turbulence Modelling for CFD*, DCW Industries.

WU, Z., LAURENCE, D. & AFGAN, I. 2017. Direct numerical simulation of a low momentum round jet in channel crossflow. *Nuclear Engineering and Design,* 313**,** 273-284.

YOUNGREN, H., BOUCHARD, E., COOPERSMITH, R. & MIRANDA, L. 1983. Comparison of panel method formulations and its influence on the development of QUADPAN, an advanced low-order method. *Applied Aerodynamics Conference.* American Institute of Aeronautics and Astronautics.