On the application of hybrid meshes in hydraulic machinery CFD simulations

M Schlipf, A Tismer and S Riedelbauch
Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70550 Stuttgart, Germany
E-mail: schlipf@ihs.uni-stuttgart.de

Abstract. The application of two different hybrid mesh types for the simulation of a Francis runner for automated optimization processes without user input is investigated. Those mesh types are applied to simplified test cases such as flow around NACA airfoils to identify the special mesh resolution effects with reduced complexity, like rotating cascade flows, as they occur in a turbomachine runner channel. The analysis includes the application of those different meshes on the geometries by keeping defined quality criteria and exploring the influences on the simulation results. All results are compared with reference values gained by simulations with blockstructured hexahedron meshes and the same numerical scheme. This avoids additional inaccuracies caused by further numerical and experimental measurement methods. The results show that a simulation with hybrid meshes built up by a blockstructured domain with hexahedrons around the blade in combination with a tetrahedral far field in the channel is sufficient to get results which are almost as accurate as the results gained by the reference simulation. Furthermore this method is robust enough for automated processes without user input and enables comparable meshes in size, distribution and quality for different similar geometries as occurring in optimization processes.

1. Introduction
Computational Fluid Dynamics (CFD) is a state of the art simulation method in today’s design processes of hydraulic turbomachinery. All hydraulic parts of the machine can be simulated before experimental investigations, i.e. model tests, are started. Besides the reduction of costs due to less time needed for expensive model tests, it is possible to visualize and investigate flow phenomena in detail. A big disadvantage of this method is that the accuracy is usually lower compared with experiments and trustable absolute values for wide operation ranges are hard to gain. Geometry simplifications of the real prototype plants, e.g. seals, are necessary to gain models which can be handled with today’s available computational resources within a reasonable time span of a project. CFD with Reynolds Averaged Navier-Stokes (RANS) equations is common practice to design hydraulic machinery by analyzing some characteristic operation points within the entire operation regime.

The target in the automated design process is not to get the highest accuracy possible in comparison to experiments but to find the best design relative to results gained by simulations of different geometry modifications. If the most important influences are captured by the simulation the best design found by this method is expected to be the best design in reality, too. An important simplification in the CFD process is caused by the discretization of the

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd
computation domain. One of the most critical steps in the CFD process is the partition of the domain into small elements wherein the discretized governing equations are solved. At the same time it is a step with a high user input effort. Only if this step is done accurately and the resulting mesh is of high quality the solution of the following calculation has a chance to be accurate and physically correct [1]. Mesh generation requires some experience and the knowledge of the expected flow solution. Regions with high gradients have to be resolved finer and elements have to be saved at regions with small expected gradients to economize the computational effort. In design optimization processes the user input should be as small as possible to make use of an automated process chain and hence to save manpower. This process consists of loops of geometry generation, meshing, calculation and post-processing. Between the single loops an optimizer creates new parameter sets for the following geometry generation in a new loop. Only if all of those steps run completely without user input the automated optimization process can be run in a meaningful manner and time duration.

As mentioned above, the availability of a good quality mesh is an important requirement to gain precise results. The use of blockstructured hexahedral meshes is recognized to create the basis for highly accurate results [2]. But the automatic modification of the geometry can lead to quality problems, degeneration of the blockstructure or invalid meshes. To avoid this limitation a method is needed which can create a simulation mesh independent from the geometry variation. A promising approach is the use of unstructured meshes which are created newly in every design loop for the new geometry with a constant chosen mesh parameter set while keeping the main mesh sizes [3, 4]. Those meshes are able to capture even complex geometries automatically [5]. The influence of the different meshing parameters is investigated to understand their influence on the results and the accuracy achieved by the use of those mesh types. To gain a high accuracy in combination with high flexibility, hybrid meshes are used which are built up by a combination of different kinds of elements [5]. Herein it is the combination of tetrahedrons with prisms or tetrahedrons with hexahedrons. In order to reduce numerical errors tetrahedral elements are used in the far field and prisms or hexahedrons are placed near walls. Tetrahedral meshes lead to higher discretization errors especially in flows with a dominant direction due to the lack of pairs of parallel faces [6]. Near the wall, the flow has a dominant direction along the wall with high gradients parallel to it. To reduce errors, elements are chosen with a face perpendicular to the flow direction [7]. This leads to the use of prisms or hexahedrons near the wall. Additionally, their size and growth rate perpendicular to the wall can be controlled better than by using tetrahedrons which effectuates advantages for the calculation of the boundary layer. Due to the element shapes, hybrid meshes need a bigger amount of elements in total compared with blockstructured hexahedron meshes and thus, a higher computational effort.

2. Mesh generation methods
The following mesh types are generated by using ANSYS ICEM CFD.

2.1. Blockstructured mesh
The mesh is built up by hexahedrons only. Special block types like O-grids or C-grids [8] are used where they are necessary to guarantee the quality criteria. The main criteria for hexahedrons are angle, which is chosen to be greater than 18°, and the ratio of the smallest and highest determinant of the Jacobian matrix at the vertices in each element, chosen to be greater than 0.3. Because of its special meaning as a reference case, poor elements are avoided.

2.2. Hybrid mesh with prisms
One version of unstructured hybrid mesh is built up by tetrahedrons in the far field and prisms in the near wall boundary layer according to Figure 1. To achieve that combination the geometrical surfaces are meshed with triangles first. Afterwards those triangles are
extruded almost perpendicular to the surface to get prismatic cells [9]. The length of the extrusion defines the height of the first cell and consequently influences the y+ value. To get an amount of prismatic layers the extrusion of the triangle surface cells is repeated a few times. The extrusion length is varied between the single steps to fit a defined growth function. This enables near wall y+ values and smooth volume transition to the tetrahedron domain at the same time. Smoothing functions for the element growing function and direction are added to meet the necessary quality criteria in the prism layers. This leads to a partial deviation from a perpendicular extrusion. After the creation of this near wall boundary layer the triangular surface meshes of the last layer on each wall is used to start the creation of the tetrahedral far field. This is created by using a Delaunay algorithm [10] followed by an additional smoothing of the resulting mesh to reach the desired quality criteria. The minimum criterion for good elements is an aspect ratio of 0.2 for tetrahedrons. Pyramids and prisms are defined by their determinant which is chosen to be at least 0.2 to specify a good element.

The mesh is controlled by setting size parameters and growing ratios on each domain surface and within the domain. Especially the creation of the near wall prism layer underlies constraints in the choice of different parameters. This limits the prism layer sometimes to a not necessarily physically wanted but geometrical dependent size.

2.3. Hybrid mesh with hexahedrons

The creation of unstructured tetrahedron meshes with hexahedrons in the boundary layer is done by creating an interface surface as an offset of the wall in a certain distance first. This interface defines the transition between hexahedral near wall boundary layers and the tetrahedral far field. Between the wall and the interface a blockstructured mesh existing of hexahedrons is created. The height of the first layer next to the wall and the growth ratio of the cell sizes perpendicular to the wall can be controlled in a wide manner. In general, the different mesh parameters can be controlled better in comparison to the possibilities of the method for the creation of prisms as described above. Adding pyramids on top of the hexahedral mesh offers triangular faces to create the tetrahedral far field. According to Figure 2 (left) those pyramids are built up from the quadrangle base with four triangular faces. The tetrahedral far field mesh is also created by a Delaunay algorithm starting from those triangular pyramid faces. Afterwards the mesh is smoothed.

For the Francis runner channel a combination of tetrahedrons, prisms, pyramids and hexahedrons should be used which leads to quality problems with the used mesh generation software. Thus, in addition to the use of pyramids a connection by a general grid interface (GGI) is done according to Figure 2 (right). To use this GGI method, the near wall region is meshed with structured hexahedrons as described above. Afterwards the quadrangles on the interface surface are split to transform them into triangles. Those triangles are used for the following Delaunay algorithm to create the tetrahedral far field. The knots of the hexahedron and tetrahedron domains on the GGI are matching on a wide range. This reduces interpolation errors to a minimum. Only near adjacent walls like shroud or hub they differ a little bit to enable the satisfaction of the quality criteria.
3. Results
The test cases are chosen with different order of simplification. The aim is to reduce the interaction of variable effects to capture the influence of different mesh types on only single flow phenomena. Those investigations show which features and criteria the mesh has to satisfy to provide a sufficient basis for a fluid simulation. Finally, a realistic double bend runner blade with its channel is simulated to get the influence on a common application in turbomachinery design. The hybrid meshes are verified with the results of simulations based on blockstructured meshes. A validation with experimental data is avoided here because they might have boundary conditions unsuited for comparisons with CFD simulation results causing further uncertainties [11]. Furthermore, the governing equations of the numerical scheme are not influenced by the used mesh. Thus, the CFD results based on blockstructured meshes serve as reference value for the comparison with the hybrid mesh results in this investigation. The following results are gained by simulations with the solver ANSYS CFX.

3.1. Numerical setup
The numerical setup is kept constant for all test cases to avoid possible additional errors caused by changes. The meshes are checked to satisfy the required quality criteria [2, 12, 13]. The mesh is valid if at least 99.5% of all elements are of good quality and the worst element satisfies at least the poor quality criteria. Additionally, the position of the at most 0.5% elements of poor quality is observed to estimate their influence on the flow field. The mesh is only permitted if those poor quality elements are scattered so that their influence on the flow field solution is minimized. All simulations are set up stationary with RANS using the high resolution advection scheme being recommended for the simulation of turbomachinery [2]. This scheme is a combination of a first and second order upwind scheme. The second order is blend with a weighting factor which is reduced where ever it is necessary to keep numerical stability. In the worst case the scheme is limited down to first order to avoid unphysical oscillations [14]. The different test case geometries and boundary conditions are chosen in a way that stationary flow fields occur so that the solver converges. Turbulence modeling is done by the SST model by Menter [15] which is also validated in turbomachinery simulations [2].

3.2. NACA0012
The first simplified test case is a symmetric NACA0012 airfoil [16] in a wide domain as shown in Figure 3 with an angle of attack of 8° at a Reynolds number of 3·10⁶. This case reduces a runner channel down to a flow around a blade without the curvature of the channel and the resulting pressure gradients and flow deflections caused additionally by the adjacent runner blades. The domain has a deepness of 1m. The domain size is a compromise of reduction of the influence of the walls and reaction of the boundary conditions on the foil and a limitation of the cell amount to reduce calculation time. As shown by [17] the domain should be bigger to avoid the influence of the domain boundaries but this is not relevant in the presented investigation because no comparison to experimental results is done. The influence is supposed to be independent from the mesh type. The angle of attack is chosen to get a clear deflection of the flow without separation on the foil because this would lead to further challenges which need special treatment in the numerical schemes and turbulence modelling which is not in the scope of this investigation. It reflects one basic target of the design process for hydraulic turbomachinery runners to optimize
the best efficiency point. The CFD results show the behavior of the flow along the blade. Especially the flow at the trailing edge shows special characteristics of the different mesh types. A first grid study with the blockstructured mesh shows a convergence with 910 thousand knots in the whole calculation area. In this configuration ten thousand element knots are placed on the blade surface. Each blade side is discretized with eighty knots in chord direction and the nose area is resolved with forty knots up to the thickest blade position. The asymptotic values for lift and drag by varying the number of knots in the whole calculation domain are $c_l = 0.860$ and $c_d = 0.0139$. The hybrid mesh with hexahedrons is built up with the same mesh on and near the wall. The element size in the surrounding far field part is chosen to get a smooth transition at the interface and high quality pyramids.

The size of the hexahedron near wall domain is varied but halving the volume shows only variations of 0.5% in drag coefficient and 0.3% in lift coefficient. This is similar to the deviation from the blockstructured validation values, where the drag coefficient is 0.2% higher by using the hybrid mesh. Due to the use of the unstructured tetrahedrons in the far field which has negligible influence only 683 thousand knots are needed in total compared to 910 thousand in the blockstructured mesh. This is necessary because the refinements around the blade have to be transported through the whole domain due to the blockstructure.

Using the same knots number on the blade for the hybrid mesh with prisms the drag coefficients differs more. The lift is only 0.8% lower but the drag is 115% significantly higher.

The coefficients react also sensitive to the chosen $y+$ values. By doubling the mean $y+$ value from 62 to 124 and keeping the remaining mesh sizes the drag coefficient lowers by 5.8% and the lift by 3.7%.

![Figure 4](image1.png)

**Figure 4.** Dimensionless pressure $c_p$ along NACA0012 foil’s chord length.

![Figure 5](image2.png)

**Figure 5.** Hybrid mesh with prisms (left) and hexahedrons (right) at NACA0012 trailing edge.
mesh with prisms in the boundary layer creates big volume steps at the trailing edge because of the construction method for the prisms. Due to the perpendicular extrusion only a few elements are created at the trailing edge whose volumes grow significantly with rise of the distance to the surface. This high volume change in adjacent cells leads to insufficient resolution of the flow field in this area.

The pressure coefficient based on the hybrid mesh with hexahedrons also fits well in this area due to using the same mesh in the near blade area with a high control level which is shown in Figure 5 (right). No recognizable effects are caused by the surrounding tetrahedral and pyramidal mesh.

### 3.3. NACA4412 cascade

The cascade is realized by a single rectangular channel with periodic boundary conditions on top and bottom. The channel has a dimension of $0.48m$ height, $0.2m$ deepness and $20m$ length at a Reynolds number of $3 \cdot 10^6$. The NACA4412 airfoil with $1m$ chord length is placed in the middle of the channel. In this case, the flow around the foil is additionally influenced by the adjacent foils and due to that closer to the realistic channel of turbomachinery guide vane and runner. The angle of attack of the NACA is set to $5.7^\circ$ to get an inflow which is tangential to the centerline on the leading edge. This accords to the optimal conditions in runner blade design process and avoids further effects and errors caused by separations.

The course of the pressure coefficient along the foil chord length shows similar behavior as for the NACA0012 case. The flow characteristics at the trailing edge for the different mesh types are not influenced by the external pressure gradient caused by the channel flow.

The NACA4412 case is also used to compare the use of pyramids at the interface to the GGI. By using matching knots on the interface surface, i.e. a one to one relationship between the knots of the adjacent element faces, the differences for the drag coefficient are $0.06\%$ and for the lift coefficient $0.002\%$. Even if the knots on the interface are just similar but not exactly fitting, the error in drag is $0.03\%$ and for the lift $0.009\%$. It would be expected that the convenient mesh would produce more accurate results. But the total knots number varies by $0.12\%$ for the similar mesh and $1.19\%$ for the convenient mesh compared to the mesh with pyramids. That is why the error caused by the GGI interface is considered negligible and the GGI coupling is used for the Francis runner case to avoid bad quality meshes even if the mesh doesn’t fit exactly in the area near adjacent walls.

### 3.4. Francis runner

The realistic test case is represented by a 15 blade Francis runner with an outlet diameter of $1.8m$. The case includes a simulation of one channel as shown in Figure 6 with rotational periodic boundaries. The discharge is set to $6m^3$ and the rotation speed is $90rpm$. The inlet boundary condition is a velocity profile gained from a simulation of a radial guide vane to get realistic conditions. This geometry represents a special case because the blade angles at the trailing edge are comparatively flat. It leads to a high effort in creating a blockstructured grid which satisfies the quality criteria. The hybrid mesh generation algorithm is also problematic in the near wall regions due to the complexity of the geometry with difficult angles and combinations of sharp edges like at the connection between blade and hub or shroud. For the hybrid hexahedron mesh the hexahedral domain around

**Figure 6.** Calculation domain of Francis runner channel.
the blade is combined with prism layers at hub and shroud. This approach avoids constraints for geometry variations due to a given mesh structure. At the connection of hexahedrons, prisms and tetrahedrons quality problems occur and a GGI instead of pyramids is required to satisfy the quality criteria. The interface in hybrid meshes with hexahedrons can be adjusted to reduce this problem but it cannot be totally avoided.

The Francis runner channel is meshed with a convenient rotational periodic mesh for all cases. Reference values are calculated with a blockstructured grid for the Francis runner channel. The Richardson extrapolation \[18\] of the simulated values calculates the asymptotic results for further grid refinement shown in Table 1 with the deviation $\Delta$ from the blockstructured reference values and the particular grid convergence index $GCI_{21}$. This index represents empirically the error of the asymptotic values \[19\] and shows a high reliability of the calculated results in Table 1.

### Table 1. Asymptotic hydraulic values for blockstructured (blk), hybrid with hexahedrons (hex) and hybrid with prisms (prism) mesh.

|          | $H$[m] | $\Delta$[%] | $GCI_{21}$[%] | $P$[W] | $\Delta$[%] | $GCI_{21}$[%] | $\eta$[-] | $\Delta$[%] | $GCI_{21}$[%] |
|----------|--------|-------------|---------------|--------|-------------|---------------|----------|-------------|---------------|
| blk      | 2.193  | 0.28        | 0.002         | 124504 | 0.002       | 0.9704        | 0.32     |
| hex      | 2.185  | −0.36       | 0.000         | 124486 | −0.01       | 0.9734        | +0.31    | 0.001       |
| prism    | 2.195  | +0.09       | 0.38          | 124580 | +0.06       | 0.9715        | +0.11    | 0.17        |

The hybrid meshes reach a high accuracy compared to the blockstructured reference values. The reference values are gained with a mesh which is refined up to 2M knots. The hybrid mesh with hexahedrons needs a similar knot number to reach this accuracy. It is recognizable that the asymptotic value for the head does not change with growing knot number. Although the hybrid mesh with prisms gets closer to the reference head and efficiency up to 3.4M knots are needed to reach sufficient $GCI_{21}$ and an asymptotic behavior as shown in Figure 7.

![Figure 7](image_url)

**Figure 7.** Standardized head, power and efficiency for blockstructured (a), hybrid hexahedron (b) and hybrid prism (c) mesh.

The $y+$ influence is not as clear as observed in NACA0012 free flow. Halving the average $y+$ value of 70 on the blade reduces the head and power only by 0.15%, doubling it reduces those values only by 0.07%. This influence is classified negligible for this operation point because no tendency is recognized. Based on this the $y+$ values for the investigations of the hybrid meshes are adjusted to be close to those used in the blockstructured validation case.

For the hybrid mesh with only prisms in all near wall layers, the average $y+$ values are chosen to be in a similar range compared to the blockstructured case but the lesser amount of control possibilities lead to a higher range of occurring $y+$ values in the channel. The quality criteria
have additional limitations on the mesh parameters. The automatic process for mesh generation requires a very sensitive choice of the parameters which is problematic in the design process because the same parameter set should be used for all geometry versions to avoid additional errors. Thus, it is assessed that this mesh method is not suitable for the fully automated meshing of hydraulic runners because of this lack of robustness in the current software version.

Investigations of the knot number in the far field show that even quadruplicate the knot number in the tetrahedron region varies the hydraulic values by at most 0.36% in head. Additionally, this variation is dependent of the resolution of the structured region. Even variations of the knot number in the near wall field around the blade by maintaining the same far field resolution show deviations in the hydraulic values by less than 0.32%.

In a further investigation the thickness of the structured block is varied. The augmentation of the boundary block leads to a more structured like meshing of the far field. The unstructured meshed part of the far field between interface and periodic channel boundary gets smaller with the growing structured part. The size of the structured region is limited by the mesh quality. Because of the matching rotational periodicity the field in between the mesh parts has to be filled with unstructured tetrahedrons which need a certain space to fulfill the quality criteria. Table 2 shows the variation of the averaged height of the near wall region for a constant element size in the far field. Although a trend is recognizable the influence on the hydraulic values is very small.

Table 2. Variation of structured domain height in hybrid hexahedron mesh

| Struct. domain height [mm] | Head [%] | Power [%] | efficiency [%] |
|---------------------------|---------|----------|---------------|
| 10                        | 100.37  | 100.02   | 99.67         |
| 14                        | 100.05  | 100.00   | 99.97         |
| 20                        | 100     | 100      | 100           |

The theoretical higher dissipation caused by the unstructured tetrahedrons in the far field has no visible relevance in this simulation of the hydraulic runner. The energy loss is represented by the efficiency. Figure 7(b,c) shows that the efficiency increases with increasing number of knots. This means that the energy losses decrease. Furthermore, the energy loss of hybrid mesh simulation is lower than by using the blockstructured reference mesh. Obviously the finer discretization of the calculation domain has a more significant influence than the numerical error caused by the element type and their orientation and arrangement.

Besides those hydraulic values which show good agreement for the different mesh kinds, the velocity distribution downstream the blade is regarded. This is a very important parameter in the design process of hydraulic machinery because the downstream draft tube can generate additional losses which influence the design point.

![Figure 8. Meridional velocity at runner’s outlet.](image-url)
In Figure 8 the blockstructured reference case of the meridian velocity distribution shows a hill close to the hub. This increase is smoothed out in the simulations with hybrid prisms meshes. The hybrid hexahedron mesh reduces this smoothing effect. For the circumferential velocity the deviations are in a negligible range. The influence of the variation on the flow and the resulting energy losses in the draft tube are not generally ascertainable because of very different possible geometrical draft tube shapes.

The computation time for the solver on the mesh types is shown in Table 3. It represents the duration for Francis test cases with similar knot numbers on different meshes. The calculation of blockstructured meshes is the fastest due to the smallest number of elements. The higher duration for hybrid meshes with hexahedrons compared to hybrid prism meshes is expected to result from the use of the GGI, because the NACA4412 case is about 40% faster by using pyramids instead of GGI. Additionally, in the NACA0012 case this mesh kind without GGI is about 15% faster than using prisms.

| Table 3. Computation time solver |
|----------------------------------|
| blk  | hex | prism |
| \(n_{\text{knots}}[\%]\) | 100  | 97.3  | 96.4  |
| \(n_{\text{elements}}[\%]\) | 100  | 290   | 304   |
| duration[\%]                | 100  | 214   | 180   |

The mesh convergence study shows in Figure 7 that the hybrid hexahedron mesh reaches a good convergence for half as much knots as the blockstructured case. Regarding the relative computation duration, those meshes need comparable solver time to reach asymptotic values. The hybrid prism mesh needs approximately 50% more knots to reach an asymptotic value which leads to almost triple computation time in total.

To evaluate the use of coarser meshes, the relative variation for different designs is investigated by keeping the geometry and changing the inlet boundary conditions to get a new flow field. This method should avoid additional deviations caused by different meshes resulting from new geometry shapes. The result shows almost no change in the convergence behavior of the mesh studies for different inlet boundary conditions. Therefore, it is not necessary to simulate very fine meshes to get a relative change for different design geometries. This saves time in the design process. Accurate results could be gained for the final design by increasing the resolution. But additional validations with experiments are necessary to know the error caused by the whole numerical method.

4. Summary and future work
This paper investigated the application of hybrid meshes on CFD analysis in hydraulic turbomachinery. The simplification of the runner is represented by NACA airfoils in channels to get the detailed flow characteristics in the different hybrid mesh types. Those are represented by tetrahedrons combined with prisms and tetrahedrons combined with hexahedrons. The validation results are generated by using blockstructured hexahedron meshes. This approach avoids the occurrence of additional errors caused by experimental methods and disregards inaccuracies caused by the numerical scheme and the simplifications in the RANS method. The simplified test cases show that the use of hybrid meshes with prisms is not sufficient to capture the flow around the blade especially at the trailing edge. This behavior influences the
pressure distribution on the blade and the flow downstream the blade. The hybrid mesh with hexahedrons in the near wall area represents the flow around the foil and the hydraulic values in almost the same manner as blockstructured reference meshes do. The hydraulic values of the Francis runner calculated with a hybrid mesh with hexahedrons around the blade and prisms near hub and shroud are very close to the blockstructured reference values. Considering the accuracy of the used CFD methods the gained differences can be regarded negligible. Also the relative course of the values for different mesh sizes make the use of hybrid meshes with hexahedrons an applicable method for design processes. Computation time can be saved by using coarser meshes for relative comparison of different design variations.

The deviations in the velocity distribution at the draft tube inlet are still a point of uncertainty and to investigate for different draft tubes. The use of this meshing method for axial runners is also a further investigation but the results of the basic studies and the similarity of flows in different kinds of runners hypothesizes that those meshes are also sufficient for the design process in that case.

References

[1] Jasak H 1996 Error Analysis and estimation in the Finite Volume method with application to fluid flows Ph.D. thesis Imperial College, University of London
[2] Braune A 2011 Qualitätssicherung und Best Practice Guidelines für Turbomaschinen Turbomaschinenseminar ANSYS Munich
[3] Irmisch S and Walker D Efficient use of hybrid grids in modern turbomachinery applications 1999 AIAA journal 37
[4] Baker T Mesh generation: Art or science? 2005 Progress in Aerospace Sciences 41 29–63
[5] Thompson J F, Soni B K and Weatherill N P (eds) 1999 Handbook of Grid Generation (CRC Press)
[6] Juretic F and Gosman A D Error analysis of the finite-volume method with respect to mesh type 2010 Num. Heat Transf., B: Fundamentals 57 414–39
[7] Juretic F 2004 Error Analysis in Finite Volume CFD Ph.D. thesis Imperial College, University of London
[8] Blazek J 2005 Computational Fluid Dynamics: Principles and Applications (Elsevier)
[9] Khawaja A and Kallinderis Y Hybrid grid generation for turbomachinery and aerospace applications 2000 Int. J. for Num. Meth. in Eng. 49 145–66
[10] Delaunay B Sur la sphère vide 1934 Bulletin de l’Académie des Sciences de l’URSS 793–800
[11] McCroskey W 1987 A critical assessment of wind tunnel results for the naca 0012 airfoil Tech. rep. DTIC Document
[12] 2014 ANSYS CFX-Solver 16.0 modeling guide ANSYS Inc., Canonsburg, USA
[13] Knupp P M, Ernst C, Thompson D C, Stimpson C and Pebay P P 2006 The verdict geometric quality library Tech. rep. Sandia National Laboratories
[14] 2014 ANSYS CFX-Solver 16.0 theory guide ANSYS Inc., Canonsburg, USA
[15] Menter F R Two-equation eddy-viscosity turbulence models for engineering applications 1994 AIAA journal 32 1598–605
[16] Abbott I and Von Doenhoff A 1959 Theory of Wing Sections, Including a Summary of Airfoil Data Dover Books on Aeronautical Engineering Series (Dover Publications)
[17] Thomas J L and Salas M Far-field boundary conditions for transonic lifting solutions to the euler equations 1986 AIAA journal 24 1074–80
[18] Richardson L F and Gaunt J A The deferred approach to the limit 1927 Transactions of the Royal Society of London Ser. A 299-361
[19] Celik I B, Ghia U, Roache P J, Freitas C J, Coleman H and Raad P E Procedure for estimation and reporting of uncertainty due to discretization in cfd applications 2008 J. of Fluids Eng. 130 1–4