UDC 621.45.01

V.V. Otroshchenko, M.O. Pikul
(National Aviation University, Ukraine)

Types of computational grids for modelling of flow in impeller machines and their features

The paper considers the characteristic features of computational grids for modeling of flow in impeller machines

Introduction

The correct choice and construction of a computational grid for performing mathematical modelling of flow in impeller machines using computational fluid dynamics (CFD) are important tasks for obtaining accurate and reliable results when conducting a study by the method of numerical experiment.

Types of computational grids

The concept of building a computational grid is that the object of the study is divided into smaller cells that allows to accurately determine and reproduce its geometry.

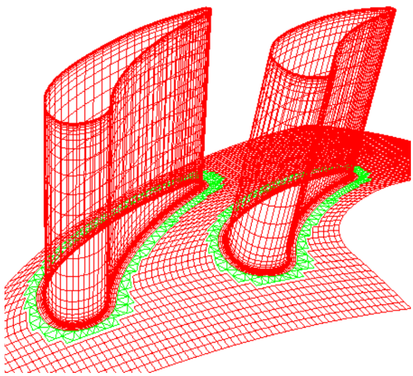

Fig. 1. Visualization of the computational grid applied to the turbine blades

A computational grid, depending on the topology of the elements that fill the computational domain, can be structured, with cells usually in the shape of a hexahedron, or unstructured, with cells of almost any geometric shape. Hybrid meshes are also a combination of structured and unstructured meshes. It should be noted that the use of a Cartesian computational grid in modelling of flow in impeller machines is not possible, due to the fact that the correct shape of the cells of such a grid cannot be fit into the objects and processes with complex geometry [1].

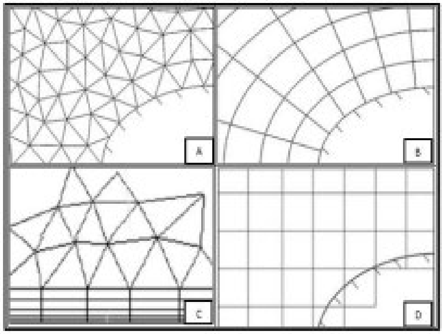

Fig. 2. Types of computational grids: A – unstructured,
B – structured, C – hybrid, D – Cartesian

For an object with a complex geometry, generating a structured grid can be a complicated task in certain cases. It may require additional time to manually divide particular sections due to its geometry. At the same time, the advantage of an unstructured computational grid is that the process of generating begins with the creation of individual points and then determines the connections between them. Since the process of generating an unstructured computational grid is automated, its generation takes less time, but the accuracy and reliability of the results of the numerical experiment may be unfavorable to the researcher compared to a structured mesh, especially in the boundary layer areas [2].

In addition, an important parameter to consider when constructing a computational mesh is its density. When modelling the flow, a computational grid with a higher mesh density should be used, because areas with a high velocity gradient or boundary layer areas near curved surfaces of the object require more accurate calculations.

At the same time, the correct choice of cell number of the computational grid directly affects the amount of time that will be spent on the calculation in software. Thus, if it is necessary to conduct a single numerical experiment a large number of times, it may take more than one day to wait for the results.

In works [1, 3], the authors investigated the issue of finding a balance between the parameters of the computational grid and the amount of time during which the calculation will take place, provided that the required accuracy of the results is achieved, using the example of selecting the optimal mesh for a centrifugal compressor. The research has shown that if the number of elements in the Ansys Workbench Student software is limited to 512,000 elements, the most optimal ratio of the accuracy of the results obtained, the number of design mesh elements and the time spent on the calculation is achieved when the number of grid elements is 489,602, which is 95.6% of the maximum quantity.

It was studied [4] that compares the results of modelling a single-impeller pump using two approaches: Fluent and CFX. The study was conducted using two different grids, but with the same settings. The conclusions after evaluating the modelling results include the following points: the Fluent mesh allowed to reduce the number of elements in the model, but it was not possible to achieve a qualitative agreement; the mesh density does not affect the qualitative accuracy, and the discrepancies may be the result of the model imperfection. This makes it possible to perform three-dimensional unsteady-state modelling with less time. The conclusion of the paper is that the tested CFD model is suitable for approximate modelling of a single-vane pump, but for more accurate results, a higher mesh quality and consideration of all the features of fluid motion and mechanical losses are required.

In the paper [5] was investigated the implementation of the Kagome truss-core structure as an internal topology for gas turbine rotor blades. An operating gas turbine was modelled using ANSYS CFX and one-way FSI analysis. The results showed that reducing the number of truss-core elements and increasing the proportion of voids leads to a decrease in the maximum stresses on the rotor blade surface. This becomes especially noticeable when the gas turbine rotation speed increases. It was found that the optimal number of truss rods for a particular blade geometry and operating conditions is 18. The use of the Kagome truss-core structure increases the strength of the gas turbine rotor blades and reduces the overall weight without negative consequences. The proposed design may also allow the use of unconventional materials such as stainless steel, which is not currently used in gas turbine blades.

Scientists in [6] considered the issue of convergence of the number and shape of cells of a structured and unstructured computational grid to obtain the closest possible results of a numerical experiment to the data received from a test physical experiment. It was found that for the structured hexagonal grid, with a gradual increase in the mesh density, the results obtained approach the verified data of the physical experiment with less gradualness than when refining the parameters of the unstructured hexahedral or tetrahedral grid.

Conclusions

To save time it is better to use an automatically generated, unstructured computational grid for objects with complex geometry.

Among the many types of computational grids and their features, the correct choice of the type and parameters of the computational grid when conducting a numerical experiment is an important part of any study in terms of achieving the required accuracy of the results of mathematical flow modelling while spending the optimal amount of time on calculations.

References

1. Sautereau, M. Research of the optimal mesh for a centrifugal compressor's volute using the GCE method Sautereau. Regensburg Applied Research Conference. Ostbayerische Technische Hochschule Regensburg Turbomachinery Laboratory Regensburg. – Germany, 2020. – P. 95–103.
2. Sadrehaghighi, I. Mesh Generation in CFD. Annapolis, MD. – 433 p.
3. Lecheler, S. Computational Fluid Dynamics. Springer. – 2022. – 210 p.
4. Matej K., Branislav K., Róbert O. Approach to 3D Unsteady CFD Analysis of a Single-Blade Pump. MATEC Web of Conferences. – EDP Sciences, 2020. – P. 328. – 02016.

5. Akzhigitov D. et al. Structural and Aerodynamical Parametric Study of Truss-Core Gas Turbine Rotor Blade. Journal of Applied and Computational Mechanics. – 2021. – 7(2). – P. 831-838.
6. Baker, N., Kelly, G., & O'Sullivan, P. D. A grid convergence index study of mesh style effect on the accuracy of the numerical results for an indoor airflow profile. International Journal of Ventilation, - 2020. – 19(4), – P. 300-314.