PERIODICA POLYTECHNICA SER. TRANSP. ENG. VOL. 34, NO. 1–2, PP. 59–68 (2006)

SPECIAL TOPICS ON MAP MESHING IN TURBOMACHINERY

Corneliu BERBENTE* Irina Carmen ANDREI*, Nicoleta Mihaela ENE*,
Florin FRUNZULICA* and Endre KORODY**

'POLITEHNICA' University of Bucharest
Faculty of Aerospace Engineering
'Elie Carafoli' Department of Aerospace Sciences
RO – 011061 Bucharest, Str. Gh. Polizu nr. 1-7.
e-mail: irinaandre@yahoo.com, nmene@yahoo.com, ffrunzi@yahoo.com,
**Budapest University of Technology and Economics
Department of Aircraft and Ships
H–1111 Budapest, Stoczek u. 6. korody@rht.bme.hu

Received: December 6, 2005

Abstract

The purpose of this paper is to present various kinds of options as regards the computational mesh generation that is required by the analysis of flow in axial turbomachinery (compressors and turbines). Specific cases of interest as both the rotating and fixed blade row profiles and axial stage are focused. There were highlighted the very best options of mesh generation that lead to a good level of computational accuracy. Therefore, one may consider this paper as a successful attempt to be a useful guide of the first steps in computational analysis of flow in turbomachinery.

Keywords: mesh generation, structured/unstructured mesh, multi-block, grid, mesh, computational domain: (airfoil, blade row, cascade/stage, axial compressor, turbine), mesh mapping.

1. Introduction

The analysis of inner flow (i.e. along the fluid flow path of a jet engine) represents a difficult task, as it requires the obtaining of accurate numerical solutions for intricate problems. Therefore, one must take into account the types and the characteristic features of the thermodynamically processes that take place distinctly in each of the jet engine's parts (*Fig. 1*).

For instance, the thermodynamics of flow inside the inlet and compressor is different from thermodynamics of flow inside the burning chamber[1], turbine and exhaust system[2]. For the hot section[3] of the jet engine, one must include the heat (and mass) transfer analysis. According to the thermodynamics of each of the jet engine's parts, one has to build the adequate mathematical model, such that to

[1] Burning chamber = burner (equivalent).

[2] By exhaust system we refer to the duct and the nozzle and/ or the afterburner.

[3] By hot section one refers to the burning chamber, turbine, exhaust system and/or afterburner, while by the cold section one means the inlet and the compressors (centrifugal, axial, fan, low pressure compressor, high pressure compressor).

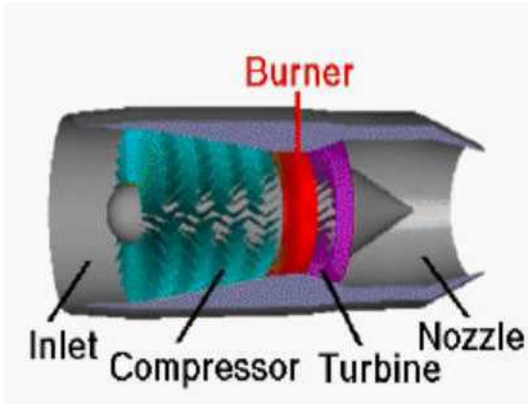


Fig. 1. The main parts of a jet engine – schematic diagram

describe the flow and/or the heat transfer. For most of the modern solutions of jet engines, the direction of flow is axial for both the compressor and the turbine (as one can notice from *Fig. 1*) and *Fig. 2* these parts are also referred as turbomachinery parts.

At present, for almost all the applications regarding the analysis of inner flow, the Navier-Stokes equations are used. The Navier-Stokes equations can be written in many ways, such as the Favre-averaged Navier-Stokes equations, the Reynolds-averaged Navier-Stokes equations, or the thin shear layer Navier-Stokes equations.

The mathematical model based on the Navier-Stokes equations must be completed with an improved turbulence model. According to Weber & Platzer the best level of accuracy can be obtained if either one of the turbulence models [1] Baldwin and Barth, (2) Spalart and Allmaras are used. According to the same source, the algebraic Baldwin and Lomax model does not offer an acceptable match of the numerical results with the experimental data. Nevertheless, a criterion for appreciating the laminar-to-turbulent transition domain should also be provided.

The present trends in Computational Fluid Dynamics (CFD) are expressed by the development and the use of the integrated 3D CFD models, which represent a combination of 3D modeling and CFD techniques.

An important step of any CFD processing, which is also the first one to be done, is the pre-processing. The results of the pre-processing represent an ordered sequence of data that refer to the geometry, aerodynamic loads, external forces, stresses, work, local velocity field, pressure field, temperature field, rotational speed and constraints. Any pre-processing must include a computational mesh generation, but it is the user's responsibility of how to build the most adequate mesh mapping.

2. The Steps for the Computational Mesh Generation

The steps that define the computational mesh generation procedure are:

- the selecting of the computational domain from the global physical domain,
- defining of the boundaries and of the boundary conditions,
- selecting the mesh types, which can be structured/unstructured a combination of the two previous and/or multi-block,
- expressing the node connections,
- filling the computational domain with selected elements that match best for the studied problem,
- defining the constraints (equivalent to a reduction of the number of the node degrees of freedom DOF); such an example is the *no-slip condition*, which means that the fluid remains attached to the solid surface, i.e. the fluid's velocity equals the velocity of the solid surface.

3. Defining the Computational Domain and Boundaries

The computational domain can be defined accordingly with the application's specificity. Therefore, from the physical domain, one can select/cut the computational domain as follows: (a)- a blade row profile[4] (2D), (b)- a set of blade row profiles (2D), (c)- a blade or a couple of blades (3D), (d)- the whole blade row (3D) that can rotate or not (i.e. a mobile blade row MBR[5] or a fixed blade row FBR[6]), (e)- a stage /a couple of stages/ the whole axial compressor (3D), (f) -either LPC[7], HPC[8] or both taken together, (g)- the global fluid flow path of a jet engine, which refers to the primary flow and/or the secondary flow (i.e. the by-pass flow), (h)- the air flow path of the cooling system, (i)- the air by-pass flow that is transferred by bleed slot, in order to prevent the initiation and the propagation of stall/surge phenomenon.

3.1. Defining the Computational Domain for a Blade Row Profile

For 2D applications, one may consider a blade row profile (*Fig. 2*) that is located inside the computational domain. Similar investigation can be found in [8]. Often used for the 3D applications are a couple[9] of adjacent blades, which belong to the same blade row (*Fig. 3*).

[4] For the turbomachinery applications, the blade row profile is used, while for the aircraft/ wing applications, the insulated profile is referred.

[5] MBR = Mobile Blade Row = Rotor (equivalent).

[6] FBR = Fixed Blade Row = Stator (equivalent).

[7] LPC= Low Pressure Compressor.

[8] HPC = High Pressure Compressor.

[9] The number of the selected blades equals two or three, according to a proper description of the flow alongside the passage between two adjacent blades of the same blade row.

Boundary Types For the case illustrated in *Fig. 2*, the boundary types are as follows:

1. *airfoil boundary*, which contours the specified profile,
2. *periodic boundary*, which is drawn at the middle line of the flow duct between two adjacent blade row profiles,
3. *inlet boundary*, which is located at the blade row inlet and its length equals the blade row pitch,
4. *outlet boundary*, which is located at the blade row exit and its length equals the blade row pitch,
5. *inter-block boundary*, which is positioned where two adjacent sub-domains having different types of meshing do join and/or overlap.

Boundary Conditions By imposing the boundary conditions one means to specify the geometrical, cinematic and thermodynamic parameters.

The *geometrical parameters* refer to the equations that describe the airfoil (i.e. the pressure side and the suction side), radii of the leading edge and trailing edge, chord and stagger angle, as well the pitch (equivalent the relative pitch or the solidity).

The inlet blade row velocity or the corresponding Mach number defines the *cinematic parameters*. The relative velocity (attached to a relative frame of reference) must be considered for the rotor blade row, while the absolute velocity (attached to an absolute frame of reference) must be used for the stator blade row.

By *thermodynamic parameters* one means the values of stagnation pressure and stagnation temperature, expressed at pre-defined locations.

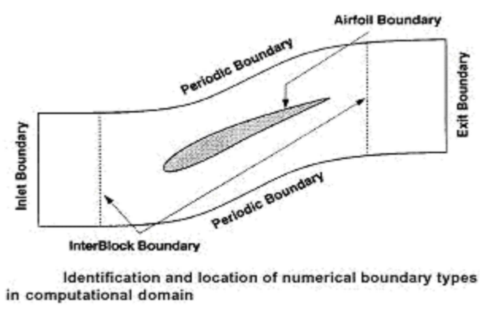


Fig. 2. The computational domain for a blade row profile – identification and location of the boundary types

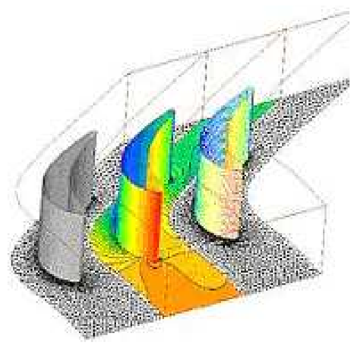


Fig. 3. Setting the boundaries and defining the computational domain for cascade blades

3.2. *Defining the Computational Domain for Stage Profiles*

Due to the fact that the topology of the computational domain represents a multiple connect field, no matter that the compressor profile or the turbine profile are considered, then the mesh mapping for them will be presented together in the next. For the compressor blades the airfoils are used, while for the turbine, the blade profiles are no longer aerodynamic. As regards the computational mesh generation, the same rules are applied.

By a stage one means a rotor and a stator taken together. The schematic diagram of an axial compressor stage is shown in *Fig. 4*. The fluid flow path is represented by a convergent duct, whose geometry is defined by the values of radii at the tip $R_V$and at the hub R_B of the blade. For the case of a turbine stage, the stator comes first (as it is a guide vane) and the rotor follows next.

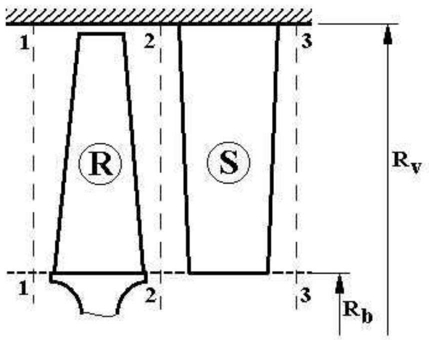


Fig. 4. The axial compressor stage

In *Fig. 5* are specified the boundary types for a turbine stage; therefore, one can notice the periodic boundaries, the stator inlet boundary as well as the rotor exit boundary and the airfoil boundary (both for the rotor and the stator). Just for the simplicity, the same denomination airfoil boundary was used both for the compressor and the turbine.

The stage profiles and velocities diagram for both cases the compressor (a) and the turbine (b) are also shown in *Fig. 5*. One can easily configure the computational domain and identify the boundary types for the compressor stage profiles by following the same pattern as for the turbine stage profiles.

3.3. *Defining the Computational Domain for Cascade Blades*

For the analysis of flow through a blade row (axial compressor or turbine, as it is shown in fig. 3) one may consider a set of cascade blades, such that to clearly point out the geometry of the passage between adjacent blades of the same blade row. A

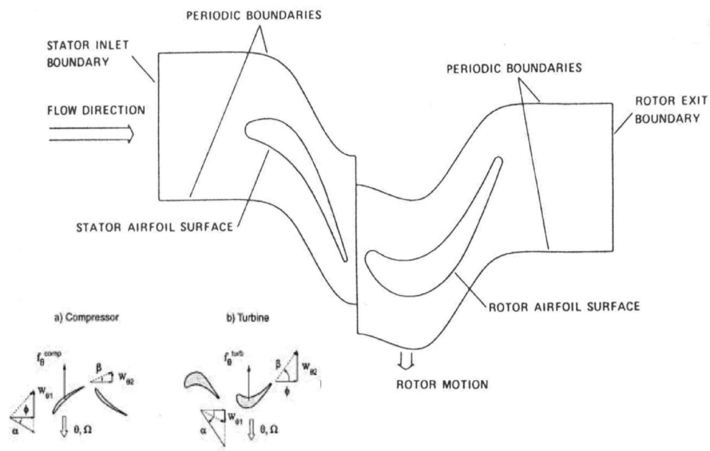


Fig. 5. Identification of boundary types in the computational domain for stage profiles (up) and the stage profiles and velocities diagram (lower) for the compressor stage (a) and the turbine stage (b)

similar procedure can be used for the axial compressor blades, with the definition of boundaries in accordance with *Fig. 2* and *Fig. 5*.

4. Mesh Mapping

For every given problem, the specified computational domain can be mesh mapped. For this purpose, a previous procedure of mesh generation must be initiated. The types of meshing are (1)- the structured, (2)- the unstructured, (3)- a combination of structured and unstructured and (4) the multi-block technique (i.e. the generation of sub-structures).

4.1. Structured Meshes

The structured meshes are mostly used due to the fact of being very simple to be built. The topology of the structured meshes can be of type C (*Fig. 6a*), type H (*Fig. 6b*) and type O (*Fig. 6c*). Often, the structured meshes are also referred as *grids*. In *Fig. 6* are shown different definitions of the computational domain for the blade row profiles.

Therefore, an H grid for a subsonic axial compressor airfoil (having 80*20 nodes) with a focus over the leading edge area, is exposed in *Fig. 7*.

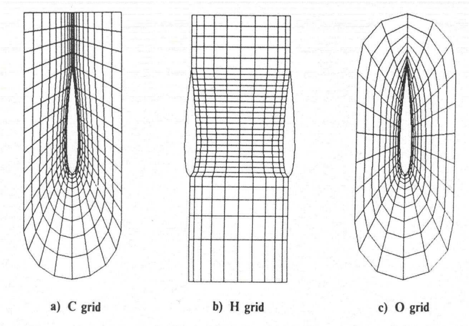


Fig. 6. Topology types of structured meshes (grids): (a) C, (b) H, (c) O

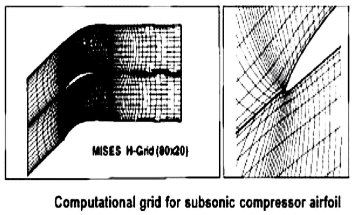
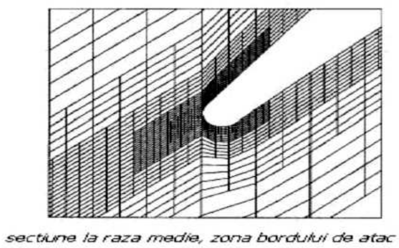
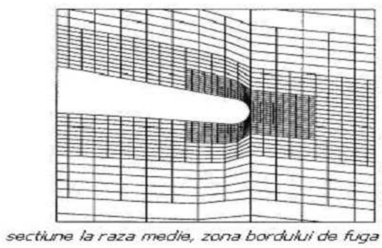


Fig. 7. An H grid (80x20 nodes) for a subsonic axial compressor airfoil; mesh mapping details for the leading edge

Fully details about mesh mapping around the leading edge of a subsonic axial compressor airfoil (*Fig. 8a*) and around the trailing edge of the same airfoil (*Fig. 8b*) by using an H grid are given in *Fig. 8*.



a) the leading edge area



b) the trailing edge area

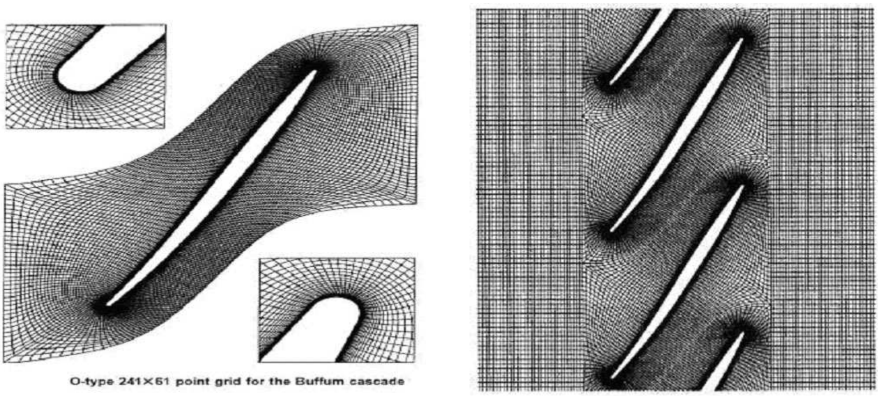
Fig. 8. Details on airfoil mesh mapping by using an H grid

For a fan airfoil, the computational domain can be mesh mapped with an O-type grid having 241*61 nodes, as *Fig. 9* shows. For the leading edge and the trailing edge the details with regards the mesh mapping are also given (*Fig. 9a*).

4.2. Unstructured Meshes

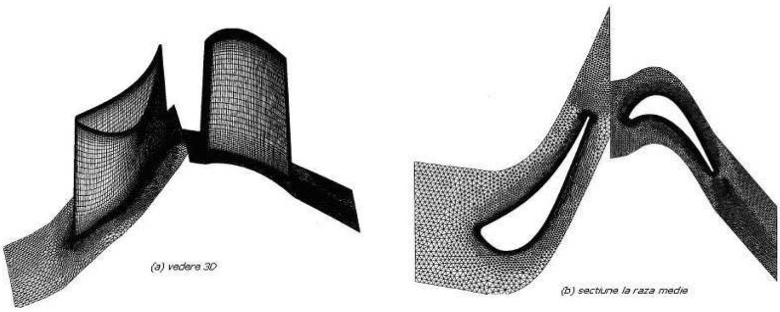
An example of unstructured meshing for the turbine stage blades (guide vane[10] and rotor) is displayed in *Fig. 10*.

[10] Guide vane = stator (synonym).



a)blading profile b) fan blade row

Fig. 9. O-type 241*61 point grid for the Buffum cascade. Fully details for the leading edge and the trailing edge.



a)3D view b) cross section at 50 % span

Fig. 10. Unstructured meshing for the stator and rotor turbine blade

4.3. Combination of Structured and Unstructured Meshes

A combination of structured and unstructured meshing is usually done such that in the vicinity of the airfoil, the structured mesh mapping is used, while for the rest of the computational domain, the unstructured mesh mapping is considered.

4.4. Multi-block Technique

The *multi-block technique*, which is also referred as the *multi-domain technique* or *the sub-structuring principle*, represents a practical idea and consists in the transformation of the physical domain in some simpler sub domains, such that to simplify the calculations. According to its definition, the multi-block technique (or the sub-structuring principle) states that any physical domain can be split in as many computational sub-domains as necessary, which can be mapped with distinct types of meshes.

The mesh mapping of the computational domain is obtained with the aid of an initial global parent mesh and a set of child meshes that can be smoothened by local refinement.

As regards the multi-block structure, the processing begins with a single *global parent mesh*, which is comprised of uniform cells and covers the entire computational domain. The first level of refinement subdivides the global parent mesh into sub-domain meshes. These segmented domains, relative to the global parent mesh, are called *child meshes*.

The only rule that must be followed in refining a mesh level is that the boundary mesh lines of a child mesh must be coincident with a mesh line of the parent mesh. Every subsequent level of mesh refinement is done in a similar manner. The child meshes on the previous mesh level are called parent meshes on their mesh level. From within these meshes, new child meshes are defined within the boundary of the parent mesh.

5. Conclusions

The intention of this paper is to present some of the most important topics with regard to the mesh generation with applications in axial turbomachinery. Therefore, cases of interest like the blading profiles (which can belong to either a rotating or to a fixed blade row), and the axial stage (a single-stage or a multi-stage axial compressor, fan or axial turbine) are focused. In order to improve the computational accuracy, there were shown several options, as: the use of the multi-block technique; different meshing in the vicinity of the profile and far away; special pattern of meshing near the leading edge and the trailing edge, *Fig. 7, 8, 9a* ; different meshing inside the cascade and outside of it; the local or global mesh refinement by increasing the mesh density.

The steps for the computational mesh generation are described and fully details with regards the defining of the computational domain and the identification of the boundaries are given. Detailed explanations regarding the boundary types and conditions are pointed out for the case of the blade row profile, as well as for the stage profile and the cascade blades, for both the axial compressor and turbine (*Fig. 2, 3* and 5). The mesh mapping can be done by using structured grids, unstructured meshes (taken apart or together, as a combination of them) (*Fig. 6, 7, 8, 9, 10*) or the multi-block technique. In accordance with the flow characteristics, i.e. inviscid or viscous, one can choose from a coarse mesh up to fine and more refined meshes. The mesh refinement allows to increasing the computational accuracy for complex flow processes.

At present, the computational mesh generation techniques have reached their fully maturity. Still, the future directions in this domain are expressed by the optimization of the generating process, the simplifying of the generating itself, the increasing of the mesh quality and the last but not the least, the block decomposing process, which allows the parallel execution of the computational codes.

References

[1] *Applied Computational Aerodynamics*, **125**, Progress in Astronautics and Aeronautics, editor Henne P. A., John Wiley & sons, 1990.
[2] *ANSYS,* http://www.ansys.com
[3] FLETCHER, C. A. J., *Computational Techniques for Fluid Dynamics 2*, Springer-Verlag Berlin Heidelberg, 1988.
[4] *Fluent, Gambit*: http://www.fluent.com/software/gambit/gambit.htm
[5] SBARDELLA, L. – IMREGUN, M., *Linearized Unsteady Viscous Turbomachinery Flows Using Hybrid Grids*, ASME Journal of Turbomachinery, July 2001, **123**, pp. 568–582.
[6] MADAVAN, N. K. – RAI M. M., *Computational Analysis of Rotor-Stator Interaction in Turbomachinery Using Zonal Techniques,* John Wiley & Sons, 1990, pp. 481-529.
[7] WEBER, S. – PLATZER, M. F., *, A Navier–Stokes Analysis of the Stall Flutter Characteristics of the Buffum Cascade,* ASME Journal of Turbomachinery, October 2000, **122**, pp. 769–776.
[8] VERESS, Á. – SÁNTA, I., *A 2D Mathematical Model on Transonic Axial Compressor Rotor Flow*, Periodica Polytechnica Transportation Engineering, 2002 **30/1-2**, pp. 53–68.
[9] http://www.pp.bme.hu/tr/index.html
[10] SPAKOVSZKY, Z. S., *Analysis of Aerodynamically Induced Whirling Forces in Axial Flow Compressors,* ASME Journal of Turbomachinery, October 2000, **122**, pp. 761–768.
[11] http://media.nasaexplores.com
[12] http://www.gasturbine.pwp.blueyrider
[13] http://www.pratt-whitney.com
[14] http://www.rolls-royce.com