International Journal of Emerging Engineering Research and Technology
Volume 4, Issue 7, July 2016, PP 14-22
ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online)

Solution Convergence for Flow through Turbine Stages (A Methodology and Approaches)

Galina Ilieva

[1]*Department Of Electromechanics, Centre for Mechanical and Aerospace Science and Technology, UBI Covilhã, Portugal*

ABSTRACT

Before new turbomachinery simulation is being started, it is very important to think carefully of what it is that should be predicted and what is the physical or computational (mathematical) phenomena that could affect the obtained results. Numerical modeling and problem solution for various geometries, design and off-design conditions, requires a number of very specific modeling and mainly convergence approaches, among others, to be estimated with taking into account of all aerodynamic particularities. Specific convergence approaches are crucial for the iteration procedure and to achieve a physically correct solution.

A complex turbine stage with application in industry is under research in current paper. Initially, the geometry modeling is carried out in Gambit, with established approaches to achieve high-quality grid; numerical modeling and calculations are fulfilled in Fluent with additionally implemented user defined codes and convergence approaches, established by the author. Numerical modeling features and various approaches to obtain solution convergence for 3D compressible, viscous and turbulent flow, through rotating machines, are under consideration and discussed in current paper. The established methodology, with its specific convergence approaches, was successfully applied to modeling and research of aerodynamic and specific flow features of flow through turbine stages to attain higher efficiency performance. Also the methodology is applicable for research of turbomachines and their exploitation in nominal and variable operating regimes, modernization and reconstruction.

Keywords: Convergence problems, High quality elements, Negative volume, Skewness, Residuals

INTRODUCTION

Flow in turbomachines is characterized by complexity and many specific aerodynamic features, caused by 3D and unsteady effects, real physical fluid properties (compressibility, turbulence effects and viscosity) and complex stator and rotor blade geometry. All aforementioned specifics contribute to mechanical and thermal stresses, secondary flows and leaks, among others, which significantly affect turbines reliability and efficiency. Precise modeling, research and analysis of fluid flow characteristics is important and very relevant task of the modern theory of thermal turbomachines.

Determination of flow parameters, considering the 3D flow characteristics, is related to solving a system of nonlinear, partial differential equations (PDEs) with derivatives on the three coordinates x, y, z and time t. Nowadays approaches are solving Reynolds-Averaged Navier-Stokes equations (RANS) system, [3, 12, 18, 27, 32, 33, 35] with appropriate turbulent model, [2, 5, 7, 14, 15, 18, 25, 28, 31]. RANS equations could be solved with 2 main solvers - Segregated Solver [4, 6, 15, 21, 27, 31] and Coupled (Implicit or Explicit) solver, [11, 16].

Flow equations can be discretized with First Order Upwind (FOU) [4,13], Second Order Upwind (SOU) [8,18,30], Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) [6, 21], etc.

For accurate prediction of complex aerodynamic phenomena, as separation, boundary layer development, effects of rotation, secondary flows and others, it is often necessary to activate specific model, even to implement additional terms to the main turbulence model. In [18, 24, 27, 29, 31, 33,

Address for correspondence:

galinaili@yahoo.com

34] the Standard k-ε model is applied, but it only gives general picture of how turbulence affects aerodynamics [22, 30, 18]. Reynolds stress turbulent model (RSM) is appropriate to model effects of additional vortices, shear stress effects over fluid particles, and rotational features, [7]. Shear Stress Model (SST) and Spallart-Allmaras (SA) [18, 30, 31] are also applicable in turbomachinery. Shear Stress turbulence model predicts location of transition; provides good agreement for numerical and test cases, [1] and has good behavior in adverse pressure gradients and separating flow. In [9, 10] is studied the capability of Large-Eddy Simulation (LES) to obtain turbulent flow specifics in turbine configurations.

A contemporary approach to stator-rotor interaction modeling is the "Mixing Planes" model, [17, 22].

An appropriate set of boundary conditions is total and static pressure, static temperature and flow direction at turbine stage inlet and static pressure and temperature at turbine stage outlet, as is shown in [11, 15, 20, 23, 31, 36].

An actual and important problem related to the process of numerical modeling and solving of 3D flow in turbine stages, is to take into account all geometry and aero-dynamic features; achieve convergence of the numerical procedure and acquire physically correct aerodynamics of flow through turbine channels. This will contribute for detailed research of turbine aggregates, working under different conditions, also to consider specific criteria and approaches for increasing efficiency and modernization of turbine aggregates.

This research is targeted to establish a numerical modeling procedure, approaches to overcome divergence problems and to obtain solution convergence for 3D compressible, viscous and turbulent flow through rotating machines.

RELATED WORK

The turbine stage under consideration, Fig. 1, is part of a double-flow low pressure turbine, working at a nuclear power plant. The configuration consists of a stator with affixed 113 blades and rotor with 127 twisted blades. Rotor blades rotate counterclockwise with 1500 rpm, extracting work from the fluid as it flows between the blades.

One important aspect of CFD simulation is the effective grid generation. Fulfilled numerous studies have shown that poorly defined elements, highly skewed elements, especially those in downstream direction, left-handed elements and disparity in elements' size lead to problems with convergence and to unrealistic results for flow parameters distribution. Thus, it is recommended a geometry reconstruction to be conducted to improve the mesh quality and avoid the presence of poorly defined finite elements [13, 19]. Multi-block hexahedral structured grid is reported as an effective mesh for curved geometries in turbomachines and is applied in our case of research, [8, 13, 19]. Cooper's discretization scheme was applied to achieve 3D discretization for the turbo-volume under consideration. It follows blades shape in radial direction, avoids negative volumes and bad quality elements. In the process of geometry modeling, different approaches to high quality grid are established and applied, [13, 19]. Boundary layer, in current modeling, consists of 16 rows of elements and has O-grid topology, which forms a continuous loop around the blade profiles. The first row of elements is with height of 0.01 m. This is very important for visualization of the process of boundary layer growth. Outside of the first cell, at a wall, is advisable to apply a growth ratio of maximum 1.25, as is proved and applied in the current research.

Due to the axisymmetrical geometry of turbine the computational domain is created only for one stage.

Water steam is imposed as a work fluid in current case. The flow is compressible, turbulent and viscous, modeled with RANS equations and solved with application of Coupled Implicit solver. Equations were discretized and solved initially with First Order Upwind discretization scheme, next the Second Order Upwind scheme was activated, to obtain more accurate solution [8, 13, 19].

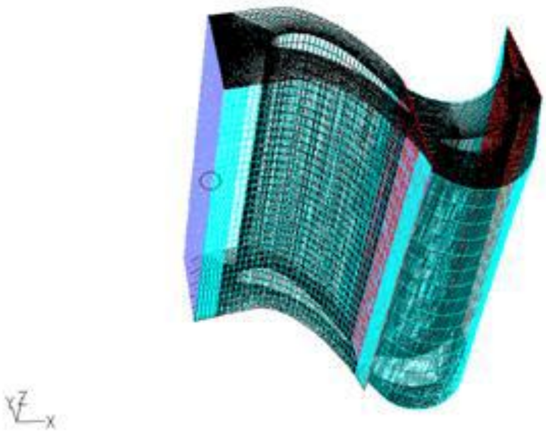


Figure1. *Merged stator and rotor volumes*.

The research, described in [13, 19], has found that there is no single turbulence model, which is suitable to all types of turbomachinery simulations; Standard k-ε and RSM models were applied to model main turbulence effects. The interaction effects between both the stator and rotor interaction were modeled with application of the so called "Mixing Planes" model.

Boundary zones and boundary values are described in details in [13, 19]. Applied boundary conditions are total and static pressure, static temperature and flow direction at turbine stage inlet and static pressure and temperature at turbine stage outlet, their values were found after 1D and 2D modeling by control sections, in radial direction, [19]. Zero flow velocities are applied to the streamed blade surfaces.

Due to the strong non-linear equations, separation and other aerodynamic effects, the solution is unstable; that is why a set of relaxation factors was implemented to remove the steep oscillations. For all under-relaxation factors, calculations were performed with initial values that were equal to the default values included in FLUENT. After a gradual decrease, the final value for each under-relaxation factor was 0.5. Convergence was obtained when the scaled residuals were in the range of 10^{-4} for all the unknown parameters; exceptions were energy and turbulence parameters, they were set to 10^{-6}.

It is advisable to apply the following logical scheme during the process of numerical modeling, [13, 19]: ●reading in Fluent a file with stage geometry and mesh data, accomplished in Gambit. ●elements quality check ●application of option "Scale" to check domain dimensions ●set up of an appropriate solver ●set up of working fluid type and its physical properties ●application of a definite turbulence model, what is the turbulence model depends on task under consideration ●activation of stator - rotor interaction model ●setting of boundary conditions by type and value, [7, 13, 19] ●application of appropriate discretization scheme ●application of relaxation factors and monitor convergence criterions ●solution initialization ●start of the iteration procedure with specific approaches to solution convergence ●activation of "Turbo Topology" option for basic turbo zones definition. ●solution results visualization ● presentation of various quantitative and qualitative flow characteristics, evaluation of energy conversion efficiency.

CONVERGENCE PROBLEMS AND APPROACHES TO THEIR SOLUTION

During the iteration process, convergence was monitored by check of residuals, statistics of forces acting on blades and check of mass flow rates through stages. Solution will stop when each of the unknown parameters meet its specified convergence criteria. In case of non-convergence, various relaxation factors must be changed; specific approaches and schemes must be applied, [19]. All approaches to convergence are related to elements' quality, fluid flow properties, turbulence model, boundary conditions, discretization schemes, etc. During the numerical modeling and solution procedure many specific approaches were settled, as discussed below.

- Once the mesh file is read in Fluent and grid quality is checked, Coupled Implicit solver is applied to solve the RANS equations set. Research on how changes of solver (from Coupled Explicit to Segregated) influence flow parameters distribution, is performed. Impossibility to get to convergence, in case of Segregated Solver is activated during the iteration process, is shown in Fig. 2. Pulsation of residuals increases close to a definite value, even though relaxation factors to a minimized value of 0.1, are decreased. In the iterative procedure, after the first some iterations, the so called "reverse flows" are reported on, [11]. The observed "reverse flows" must gradually decrease, till they disappear in the researched flow volumes. This is a guarantee for non-admission of solution errors and faster convergence of the iteration procedure. In case of activated Segregated Solver, at the beginning of the iteration procedure, convergence could not be achieved, see Fig. 3.

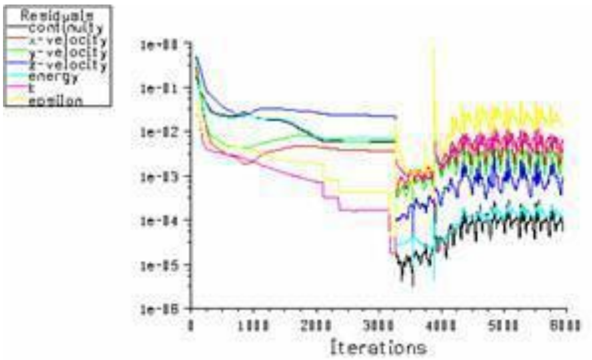


Figure2. *Impossibility for solution convergence in case of activated Segregated Solver for 3D flow*

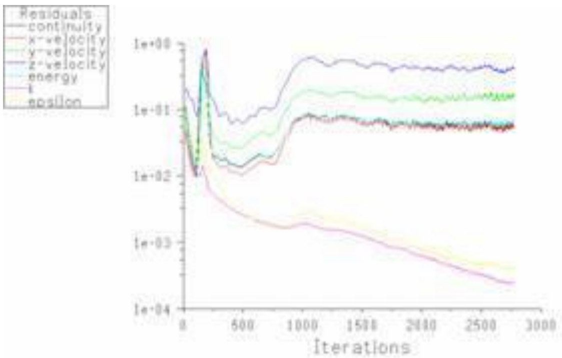


Figure3. *Impossibility to convergence when Segregated Solver set by default*

- In case of initially activated more complicated turbulence model, the following approach to convergence is applied: at first, simulations must be carried out, with activated Standard k-ε turbulent model or, in some cases, until convergence is acquired. Next, the obtained solution must be applied as a base for further computations with applied preferred turbulence model.
- If turbulence residuals are going to divergence, they must be suspended from the iteration procedure. In a close proximity of the expected convergence point, solution for turbulent parameters must be activated again.

On the basis of performed research works it's found that the aforementioned approach is applicable in case of divergence for the energy equation, Fig. 4.

- Research works have shown that number of points for the "Mixing Planes" zone, in which momentum, energy and mass exchange are realized, must be more than the default number of 10. This is for the purposes to achieve better modeling of stator-rotor interaction influence and solution accuracy. Interaction between stator and rotor blades is defined of type "rotational".

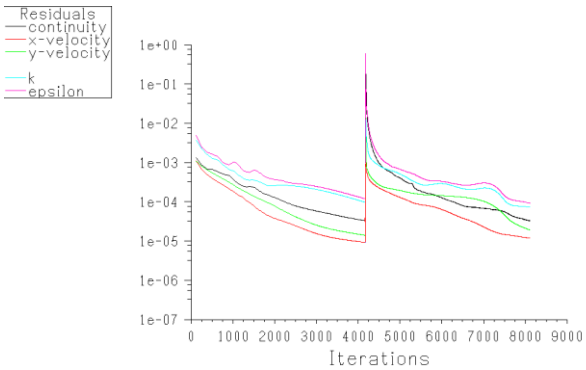
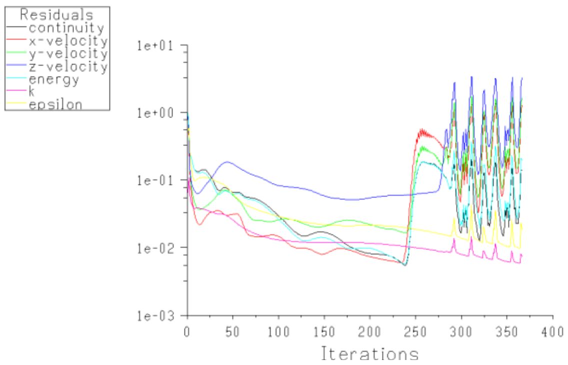


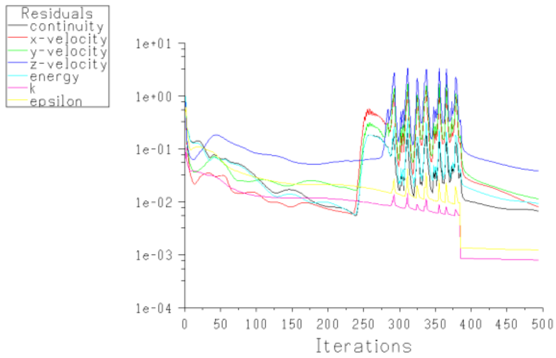
Figure4. *Suspend of energy equation during the iteration procedure.*

- Relaxation factors must be set initially to 0.5. The initially applied discretization scheme for every equation is First Order Upwind; Courant-Friedrich-Levy (CFL) number is 2. In most cases of preliminary research significant variation in the residuals progress is observed. During the iteration procedure, residuals' variation gradually vanishes and convergence is obtained, Fig. 5 (a, b).

FOU scheme, for initial discretization of equations, is defined. In the course of the calculations, a significant change in residual values is observed. There is almost negligible convergence of the iterative process even after a reduction of CFL from 2 to 1.5 and then to value of 1. After nearly 300 iterations, SOU scheme is activated. In this case, the initially reported harsh changes in residuals are followed by a rapid move towards convergence of the solution procedure. Due to the effects of viscosity, compressibility and turbulence effects, pulsations in continuity and energy equations residuals can be observed again, Fig. 5.



(a)



(b)

Figure5(a,b). *Strong variations in residuals progress, (a) progress to convergence after CFL decrease to value of 0.1, (b)*

- Rotation of rotor blades leads to presence of vortices and vortex structures, secondary flows and to variation in boundary conditions at outlet. Possible convergence problems could be avoided after application of the following approach, which was established: •calculations to determine flow parameters distribution in case of stationary rotor •set up of a very small rotational speed •after convergence is attained, one must impose a gradual increase of rotor angular velocity. In this case, each global iteration set will start based on the previously obtained solution, Fig. 6. This process will continue until full rotational velocity is defined and solution is obtained.
- Minimum CFL value that could be set is 0.10. In Fig. 7 is shown the progress of residuals to convergence after CFL was decreased to 0.10. In Fig. 8 is presented the impossibility to get convergence when low CFL value is set at the beginning of the iteration procedure. In more details, in case of observed large peaks for the residuals of the unknown parameters, there is a significant change in values of all unknown flow parameters. In order to decrease residuals' variation and to control the iteration procedure, a decrease of CFL value from 1 to 0.10, is advised. Iteration procedure continues with a value for CFL equal to 0.10. During the iteration procedure, the flow is characterized with unchangeable physical properties, rotor blades are stationary and the impact of turbulence is described with Standard k-ε model. Any unaccounted flow characteristic leads to additional convergence problems, thus change in already defined relaxation factors, is advised. In some cases, activation of new approaches for convergence, are not useful. This has a major impact on the time needed to obtain solution and leads to a lack of convergence, Fig. 8.

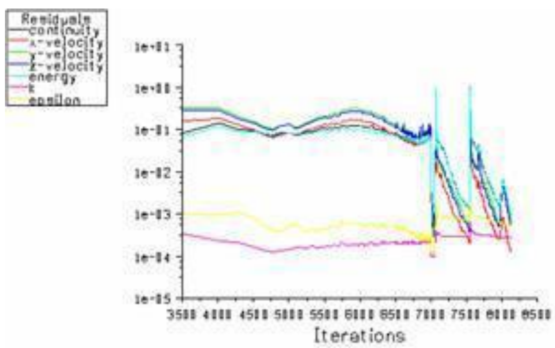


Figure6. *Start of a new iteration with increased rotational decreased value*

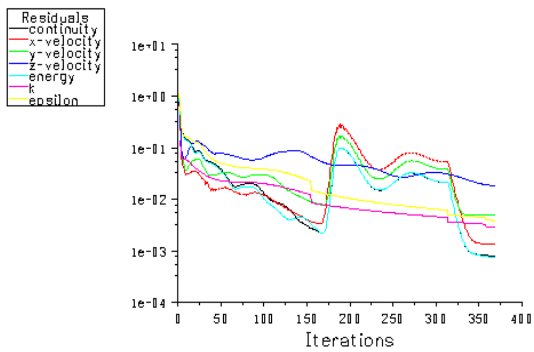


Figure7. *Progress to convergence for velocity of CFL to 0.10*

- An additional approach to obtain convergence is to activate the Residual Smoothing option for all unknown parameters, Fig. 9. Smoothing factor for flow parameters is in the range of 0.5 - 0.1 for every five iterations or for every one iteration. After a number of studies it's found that the continuous growth of residuals could be avoided through further activation of the Residual Smoothing option, in the Control Solution panel. In case of activated Residual Smoothing, a periodical smoothing and faster convergence are observed. Also, important roles have options, called Residual Smoothing and Control Solution.

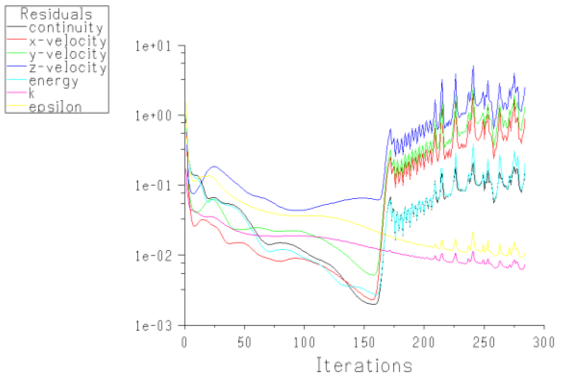


Figure8. *Impossible convergence for low value of CFL being set up in the beginning of the iteration procedure.*

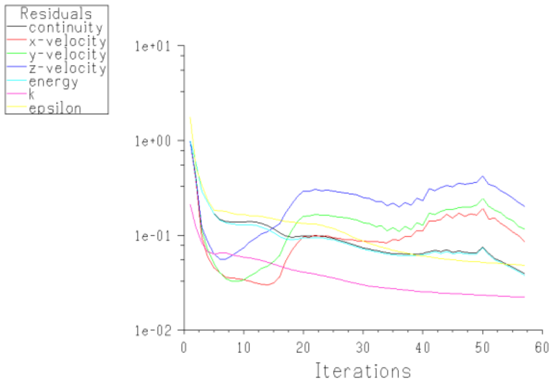
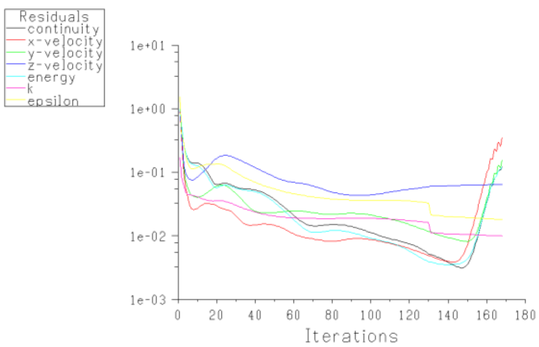
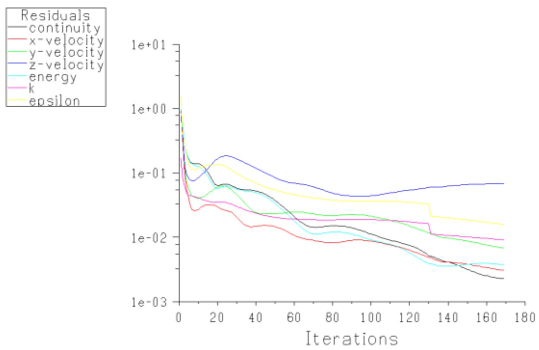


Figure9. *Progress to convergence in case Residual Smoothing option is activated.*

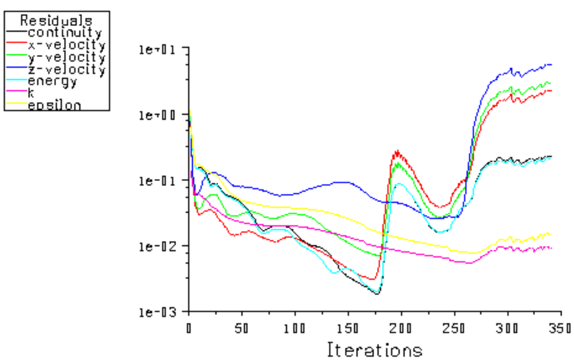


(a)

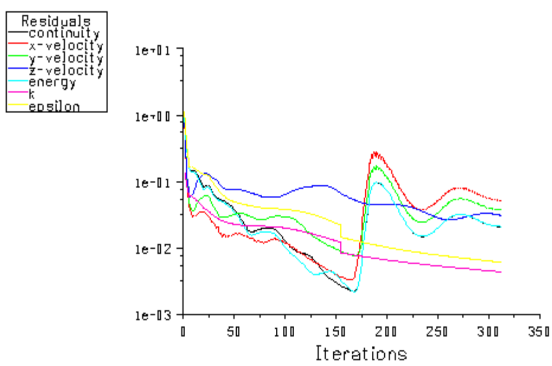


(b)

Figure10(a,b). *Change in progress of residuals in case of strong flow perturbations (a); residuals progress to convergence in case of preliminarily decreased CFL value from 1 to 0.75 is set up (b).*



(a)



(b)

Figure11(a, b). *Variations in the progress of residuals and impossibility for solution convergence, (a); progress to convergence in case of preliminarily decrease for relaxation factors (b).*

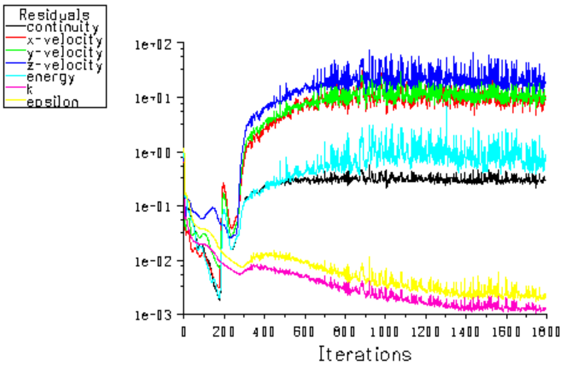


Figure12. *Impossibility to solution convergence in case of unchangeable relaxation factors during solution procedure.*

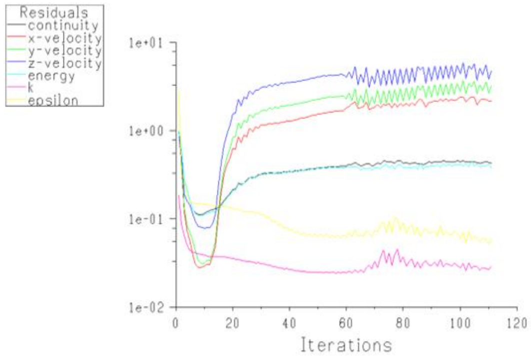


Figure13. *Divergence in case SOU and small CFL are applied at the beginning of the iteration procedure.*

- One method to obtain solution convergence is the preliminary decrease of residuals, Fig. 11(a,b). Impossibility for convergence is typical in case of invariable relaxation factors during the solution procedure, Fig 12.

In the solution process, the established and recommended scheme to achieve smoothing for observed changes in the residuals, is as described in the following lines. Before the iteration, after which strong changes in residuals for continuity and energy equations are observed, followed by the impossibility to return to convergence through described approaches above, to carry out the numerical procedure, a reduction of Residual Smoothing and Smoothing Factor values is required.

A slight increase in residuals for two consecutive iterations is observed, after application of the described approach, then residuals go to the solution point under the specified factors of convergence. For example, at the beginning of the computational procedure when were set up SOU discretization scheme, relaxation factors of 0.5 and CFL equal to 2, before iteration number 78 is reached, Residual Smoothing equal to 10 and Smoothing Factor equal to 0.50 must be defined. Before iteration number 145, to avoid the problem of significant variations in the residuals and their divergence, the CFL number is decreased from 1 to 0.75, Fig. 10 (a, b). Before iteration 245, the observed harsh change in residuals, can be avoided by a decrease of CFL number to 0.50, followed by a decrease of Residual Smoothing to 5 and decrease of Smoothing Factor to 0.30, Fig. 11(a).

• In Fig. 13 is shown an example of residuals divergence in case of initially activated SOU discretization scheme and very low Courant number. After uniform residuals are achieved and iteration procedure is close to convergence, turbulence equations are re-activated, in order to model turbulence parameters distribution and to get physically correct results, Fig.14.

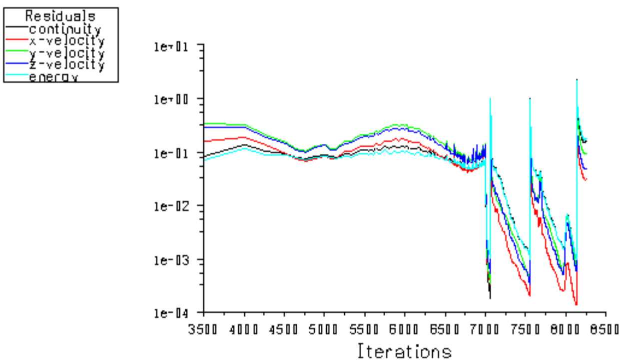


Figure14. *Process to get solution convergence in case of switched off turbulence equations during the iteration procedure.*

Also, the approach is applicable in case of convergence problems with respect to the energy equation.

CONCLUSION

A logical sequence with included specific approaches to attain high quality elements, to overcome convergence problems, and to fulfill numerical modeling of specific aerodynamic features, in 3D stages of complex geometry, was attained.

The established methodology, with its very specific convergence approaches, was successfully applied to: to obtain static pressure distribution along streamed profiles; research of aerodynamic and specific flow features at stator and rotor outlets; study on radial gap influence on flow parameters distribution; research on wet-steam conditions roughness impact over blade surfaces; study on erosion effects over turbine blades; analysis of boundary layer growth along turbine blades and predicting it's separation; also, can substitute expensive experiments.

The logical sequence with all implemented approaches to obtain convergence is very important and considered as state-of-the-art to attain: a detailed visualization for flow parameters distribution; evaluation of energy efficiency; study on thermal stresses; to fulfill stress and modal analysis in turbo aggregates and their elements; research on exploitation in nominal and variable operating regimes; modernization and reconstruction of turbomachines.

REFERENCES

[1] Florian Menter, Jorge Carregal Ferreira, Thomas Esch, Brad Konno, "SST Turbulence Model with Improved Wall Treatment for Heat Transfer Predictions in Gas Turbines", Proceedings of the International Gas Turbine Congress, IGTC2003-TS-059, 3-7 November 2003.

[2] Alessandro Corsini, "A FE Method for the Computational Fluid Dynamics of Turbomachinery", SOCRATES Teaching Staff Mobility Program, 1999-2000.

[3] Reza Aghaeitog, A. Mesgharpoor Tousi, M. Soltani, "Design and CFD analysis of centrifugal compressor for a microgasturbine", Aircraft Engineering and Aerospace Technology, Vol. 79, Issue 2, pp.137-143, 2003.

[4] Chandrakant. R. Kini, Satish Shenoy, B, N. Yagnesh Sharma, Numerical Analysis of Gas Turbine HP Stage Blade Cooling with New Cooling Duct Geometries, IJESE, Volume 05, No. 04 (02), pp.1057-1062, 2012.

[5] S. Bejanirat, L. N. Samear, Guanpeng Xu, "Evaluation of Turbulent Modeling for the Prediction of Wind Turbine Aerodynamics", AIAA-2003 - 0517.

[6] Fernández Oro, J., Argüelles Díaz, K., Santolaria Morros, C. Ballesteros, Tajadura R., "Unsteady Flow Analysis of the Stator-Rotor Interactions in an Axial Flow Fan", Proceedings of ASME FEDSM'03 4th ASME-JSME Joint Fluids Engineering Conference Honolulu, Hawaii, USA, July 6-10, 2003, FEDSM 2003-4 5394.

[7] Ansys –Fluent 12.0, User's Guide, www.fluent.com

[8] Anna Avramenko, Alexey Frolov, Jari Hämäläinen, "Simulation and Modeling of Flow in a Gas Compressor", Journal of Applied Mathematics, Vol. 2015 (2015), Article ID 682469, 7 pages, 2015. http://dx.doi.org/10.1155/2015/682469

[9] N. Gourdain, G. Wang, F. Duchaine, L. Gicquel, "Application of Large-Eddy Simulation to rotor/stator configurations", 21 Congres Francais de Mecanique Bordeaux, 26-30 August 2013.

[10] W.A. McMullan, G.J. Page, "Towards large eddy simulation of gas turbine compressors", Progress in Aerospace Sciences, Vol.52(0), pp.30-47, 2012.

[11] Vinícius Guimarães Monteiro, Edson Luiz Zaparoli, Cláudia Regina de Andrade, Rosiane Cristina de Lima, "Numerical Simulation of Performance of an Axial Turbine First Stage", J. Aerosp. Technol. Manag., Vol.4, No 2, pp. 175-184, 2012.

[12] Hong Yang, Dirk Nuernberger, Hans-Peter Kersken, "Toward Excellence in Turbomachinery Computational Fluid Dynamics: A Hybrid Structured-Unstructured Reynolds Averaged Navier-Stokes Solver", Journal of Turbomachinery, Vol. 128, No 2, pp. 390-402, 2006.

[13] Galina Ilieva, "Numerical Modeling and Research of 3D Turbine Stage, Engineering Applications of Computational Fluid Dynamics", Springer Advanced Structured Materials series, Vol. 44, pp. 103-126, 2015.

[14] Innovative Turbulence Modeling: SST Model in ANSYS-CFX", Technical Brief, ANSYS-CFX, 2005.https://support.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/techbrief/cfx-sst.pdf.

[15] Jin Yan, David Gregory-Smith, "CFD Simulations of 3-Dimensional Flow in Turbomachinery Applications", Turbomachinery Flow Prediction VIII ERCOFTAC WorkshopLac Clusaz, France, March 2000.

[16] Jochen Gier, Bertram Stubert, Bernard Brouillet, Laurent de Vito, "Interaction of Shroud Leakage Flow and Main Flow in a Three-Stage LP Turbine", Proceedings of ASME Turbo Expo

2002 June 3-6 2002, Amsterdam, The Netherlands, 2002.

[17] Jochen Gier, Sabine Ardey, Stefan Eymann, Ulf Reinmcller, Reinhard Niehuis, "Improving 3D Flow Characteristics in a Multistage LP Turbine by Means of Endwall Contouring and Airflow Design Modification Part 2: Numerical Simulation and Analisys", Proceedings of ASME TURBO EXPO 2002, June 3-6, Amsterdam, The Netherlands, 2002.

[18] Anna Avramenko, Alexey Frolov, Jari Hämäläinen, "Simulation and Modeling of Flow in a Gas Compressor", Journal of Applied Mathematics, Volume 2015 (2015), Article ID 682469, 7 pages http://dx.doi.org/10.1155/2015/682469.

[19] G. Ilieva, "Modelling, Research and Analisys of 3D Real Flow in Turbine Stages of Complex Geometry", Dissertation Thesis, Technical University-Varna, 2009.

[20] N. Sakai, T. Harada, J. Imai, "Numerical Study of Partial Admission Stages in Steam Turbine", JSME International Journal Series B Fluids and Thermal Engineering, Vol. 49, No. 2 Special Issue on International Conferences on Power and Energy,pp. 212-217, 2006.

[21] O. Uzol, D. Brzozowski, Y.C. Chow, J. Katz, C. Meneveau, "A Database of PIV Measurements Within a Turbomachinery Stage and Sample Comparisons with Unsteady RANS", Journal of Turbulence, Volume 8, № 10, 2007.

[22] http://www.cfd-online.com/Wiki/Best_practice_guidelines_for_turbomachinery_CFD

[23] Piotr Lampart, Andrzej Gardzilewicz, Sergey Yershov, Andrey Rusanov, "Investigations of Flow Characteristics of an HP Turbine Stage Including the Effect of Tip Leakage and Windage Flows using a 3D Navier-Stokes Solver with Source/Sink-Type Boundary Conditions", Proceedings of 2000 International Joint Power Generation Conference Miami Beach, Florida, July 23-26, 2000, IJPGC2000-15004.

[24] Ruprecht, A., Bauer, C., Gentner, C., Lein, G., "Parallel Computation of Stator-Rotor Interaction in an Axial Turbine", ASME PVP Conference, CFD Symposium, Boston, 1999.

[25] Sarun Benjanirat, Lakshmi N. Sankar, Guanpeng Xu, Evaluation of Turbulence Models for the Prediction of Wind Turbine Aerodynamics, AIAA-2003-0517 1, 2003.

[26] Sergey V. Yershov, Andrey Rusanov, "Numerical Simulation of 3D Viscous Turbomachinary Flow with High-Resolution ENO Scheme and Modern Turbulence Model", Conference - CFD for Turbomachinery Applications 1-3.09.2001, Gdansk Poland, 2001.

[27] Bashar, Mohammad M., "Computational and Experimental Study on Vertical Axis Wind Turbine in Search for an Efficient Design", Electronic Theses & Dissertations, Fall 2014.

[28] S. Sarkar, Peter R. Voke, Large-Eddy Simulation of Unsteady Surface Pressure Over a Low Pressure Turbine Blade Due to Interactions of Passing Wakes and Inflexional Boundary Layer, Journal of Turbomachinery, Vol. 128, No2, pp. 221-231, 2005.

[29] Toyotoka Sonoda, Toshiyuki Arima, Marcus Olhofer, Bernhard Sendhoff, Friedrich Kost, P.A. Giess, "A Study of Advanced High-Load Transonic Turbine Airfoils", Journal of Turbomachinery, *Vol.* 128, No 4, pp. 650-657, 2004.

[30] Florian Menter, "Turbulent Modeling for Turbomachinery Applications", QNET-CFD Meeting, Lucerne, 2002.

[31] Anand Kumar S Malipatil , Anantharaja M.H, "Optimization of single stage axial flow compressor for different rotational speed using CFD", International Journal of Advance Research in Science and Engineering, Vol. No.3, Issue No.9, pp.254-261,September 2014.

[32] Yang Huitao, "Investigations of Flow and Film Cooling on Turbine Blade Edge Regions", A Dissertation Thesis by Huitao Yang, August 2006.

[33] Gurdeep Singh Atwal, Sanjeev Kumar Dhama, "Computational Study on Effect of Parameters on Stress of Centrifugal Compressor Blades", International Journal of Current Engineering and Technology, Vol.5, No.3, pp.1808-1812, June 2015.

[34] V.C. Arunachalam et al, Numerical studies of the effect of impeller blade screw on centrifugal compressor performance, Proceedings of the 4th BSME-ASME International Conference on Thermal Engineering, 27-29 December 2008, Dhaka, Bangladesh.

[35] Phull Gurloveleen Singh et al, "Modelling and stress simulation of centrifugal compressor", International Journal of Advances in Thermal Sciences and Engineering, Vol.1, Issue 1, pp.99-106, June 2010.

[36] Surendran Anish et al, "Study of secondary flow modifications at impeller exit of a centrifugal compressor", Open Journal of Fluid Dynamics, Vol.2, pp. 249-256, 2012.