Numerical assessment of a very high pressure ratio centrifugal impeller

O Dumitrescu1,2, V Drăgan1, I Porumbel1 and B Gherman1

1 National Research and Development Institute for Gas Turbine COMOTI, CFD Department, av. Iuliu Maniu 220D, Bucharest, Romania
2 POLITEHNICA University of Bucharest, Power Engineering Doctoral School, Splaiul Independenței, No. 313, Bucharest, Romania

E-mail: oana.dumitrescu@comoti.ro

Abstract. The paper focuses on studying the influence of mesh size, numerical discretization schemes and different turbulence models on radial impeller performances. The chosen impeller is a 14:1 pressure ratio novel design for use in the TIDE pulse detonation engine architecture. In preparation for the experimental research, numerical studies were performed for five steady-state analysis and one unsteady, single passage with a fully structured solver. A grid sensitivity study was conducted over four grid resolutions, starting from a coarse to dense mesh (0.7 mil, 1.5 mil, 5 mil and 9 mil), in order to determine the influence of mesh points, numerical stability and reduce interpolation errors. Another problem in a CFD study is the selection of a proper turbulence model, thus six RANS models were investigated: k-ε, v²-ℓ, k-ω Shear Stress Transport (both steady and unsteady), Spalart – Allmaras and an Explicit Algebraic Reynolds Stress Model. Results show that global parameters such as pressure ratio and efficiency predictions remain unchanged above 1.5 mil cells/passage, while above 5 mil cells/passage, RANS models become unstable and lead to poor convergence. The most conservative model proved to be the SST with minimalistic values for both pressure ratio and isentropic efficiency.

1. Introduction

Numerical control plays an important role in determining overall performances of a turbomachinery, but also in establishing a level of precision as high as possible between the numerical and the experimental part. One of the main advantages of numerical methods consists on obtaining an initial/final solution much faster than experimental methods, meaning that the computation time, but also the costs are much lower.

Due to flow complexity, turbulent quantities develop in the flow structures; which can be either direct computed (Large Eddy simulation-LES, Direct numerical simulation-DNS) or solved through Navier-Stokes equation with appropriate models for these quantities. Various studies focus on adverse pressure gradients prediction, corner separation and curvature to predict influences of flow turbulence characteristics over compressor performances. Gibson [1] in his comparative study of different turbulence models for a centrifugal compressor stage (impeller and vane diffuser), determined that Shear Stress Transport (SST) model is stable and offers good results for overall performance, while SA is not recommended for advanced flow predictions. Also, the comparative analysis, between numerical and experimental results, presents a discrepancy below 2%. Numerical simulations for a radial turbomachinery, using turbulence models k-ε standard, RNG and RSM (Reynolds Stress
Models) were compared with experimental data and 1D design results. For radial turbo machines, RNG proved to be a proper turbulence model, providing better results than k-ε [2].

A good agreement between SST turbulence model and experimental results was also obtained by Mao et al. [3], in the study of an annular volute-type pump. Furthermore, in the same research, the RNG k-ε model was not able to predict all vortices locations. To determine general performances of a centrifugal pump, Shah et al. [4] used URANS equations together with two-equation k-ε turbulence model, proving that the difference between the experimental and numerical data is below 10%.

Influence of Chien’s [5] k-ε and k-ω SST turbulence models on blade angle distribution, showed that SST model offers more representative results, while the difference between the two cases in total-to-static efficiency is near 14% [6]. Performances of SST, SST–CC, SAS–SST (Scale Adaptive Simulation) and SSG–RSM (Speziale-Sarkar-Gats) model were compared with standard k-ε model, analysed for a cylindrical tank agitated by a Rushton turbine [7]. Due to 3D chaotic nature of the fluid flow downstream of the impeller blade, SST and k-ε turbulence models can’t properly capture the vortices developed in that area, also near the blades a poor turbulence prediction was confirmed for k–ε and RSM models.

SST turbulence model was improved by Smirnov and Menter [8], considering also a rotational-curvature correction. Comparative with the standard SST model, the improved one show an improvement in terms of accuracy; also in terms of computational costs, SST costs are less than that of RSM.

Bourgeois [9] investigated four RANS turbulence models (SST, SST-RM, RSM-SSG and k-ε model), implemented for a centrifugal compressor stage with a tandem impeller. According to the research there are significant differences between the models: SST and SST-RM models offer wider shroud separation, while k-ε presents no flow reattachment. In terms of turbulent kinetic energy, Joshi [10] determined that k-ε model guarantees qualitative results for flows with weak recirculation, while between RNG and standard k-ε model is a significant discrepancy. Whilst, in areas with strong swirl Murthy et al. [11], found out that standard k-ε model, in the impeller region, underestimates the turbulent kinetic energy profile.

Drăgan et al. [12] analyzed the influence of mesh structure on convergence and also turbulence models influence on main parameters of the compressor; proving that the turbulence model influences the mass flow leading to a significant variation thereof. In order to calculate small scale structures the discretization schemes need to be high order and low dissipation. First order schemes comparative with higher orders offer a rapid convergence, but less accurate convergence. The accuracy of the results for a numerical simulation depends significantly on the grid resolution, and especially on y+ turbulence model. Depending on the Reynolds number and turbulence model, the value of y+ has a major impact on the results. Also, Neverov [13] determine that the most relevant parameters of the computational model are the size of the grid and outlet boundary location. Non-optimal values for them increase the converging time by 2 times or more and thereby influencing the results.

In conclusion, study show that steady and unsteady RANS simulations show generally a good agreement in the prediction of turbomachinery characteristics. However, the sources of discrepancies between numerical and experimental results can be numerous due to the limitations of CFD tools. Precision of the results is strongly affected by numerous parameters, among them are: discretization methods, grid size and turbulence model [14]. Aside from traditional RANS simulations, Large Eddy Simulations (LES) assures more accurate results, especially for off-design conditions where RANS models are not capable to converge [15].

The present paper highlights the influence of the most important factors that underlie the process of performing a numerical analysis for a turbomachinery, such as: mesh size, numerical discretization schemes and turbulence models.

2. Numerical Setup

The spatial discretization of the domain was performed with Autogrid - Numeca software, using a structured multi-block strategy. In order to establish a proper mesh structure capable to ensure case
convergence (low discretization error), a grid independence study was performed, being analyzed four grid resolutions: 0.7 mil., 1.5 mil., 5 mil. and 9 mil. For all grids, a boundary layer with $y^+$ value of one unit and the rest of the boundary layer was meshed at a 1.1:1 growth ratio in order to maintained numerical accuracy, as shown in figure 1. Also, for every grid size, six RANS models were investigated: k-$\varepsilon$, $\nu^2$-f, k-$\omega$ Shear Stress Transport (both steady and unsteady), Spalart – Allmaras and an Explicit Algebraic Reynolds Stress Model.

Besides the influence of the computational grid and the turbulence models, the paper also focuses on numerical discretization schemes, like: second order discretization schemes and central differencing scheme.

As boundary conditions, for the inlet domain: absolute total pressure (101353Pa), absolute total temperature (300K) and a mass flow of 1.65kg/s for the outlet domain. The fluid defined was Air Ideal Gas and for all solid surfaces non-slip boundary conditions was used.

The impeller is a 14:1 pressure ratio novel design for use in the TIDE pulse detonation engine architecture [12]. Therefore, the operating point of the speedline was chosen so that all models tested can converge to a feasible extent. To ensure solution convergence, residuals where monitored, along with mass flow between inlet and outlet and other parameters of interest.

3. Results and discussions

The grid dependence study focuses on pressure ratio and isentropic efficiency of the impeller. Thus, a pressure ratio grid sensitivity study is illustrated in figure 2, together with the discretization schemes used. Giving the results, grid size affects the analysis stability, also from one turbulence model to another there are significant differences in all parameters of interest.

Figure 3 presents isentropic efficiency histograms for all four RANS cases. RSM model and V2f overpredicts pressure ratio, and implicitly isentropic efficiency no matter the grid size and numerical discretization scheme. Compared with SST model, SA illustrates a slightly increase of pressure ratio, while the efficiency is slightly lower.
It is well known that SA is a single transport equation model, without wall functions, but with a good convergence rate especially for aerospace applications. Even if it is gaining more popularity in turbomachinery applications, the fluid flow through them is quite complex, bringing the model to overestimate or underestimate the parameters of interest. Therefore, other factors such as grid size and numerical discretization schemes should be taken into account in evaluating the performance of SA turbulence model.

In terms of isentropic efficiency, a lower grid size, overpredicts the results for both numerical discretization schemes; while an increase of grid size, above 1.5 mil. leads to significant differences between all models. The only acceptable difference among the results is given by a grid size of 1.5 mil; but only for SA and SST model. While RSM model overestimates the results for all grid size and numerical models, the V2f remains closer to SST model.

V2f model is similar with k-ε, but with near-wall turbulence anisotropy and non-local pressure-strain effects [16]. It is more a low Re-number, suitable for complex secondary flows and separations; especially in turbomachinery off-design simulations.

Results show that global parameters such as pressure ratio and efficiency predictions remain unchanged above 1.5 mil cells/passage, while above 5 mil cells/passage, RANS models become unstable and lead to poor convergence. Further, are presented the results for 1.5 mill cells case, for RANS and for URANS models. Figures 4 and 5 illustrates the pressure ratio and efficiency for RANS turbulence models and as numerical discretization schemes: second order discretization and central differencing schemes. RSM model predicts greater values for pressure ratio and for isentropic efficiency, comparative with the other models. Being the only model that behaves different for both numerical discretization schemes, it was concluded that for the case studied, RSM model could not properly predict overall performances.
The inappropriate behaviour of the RSM model appears also for URANS studies, showing greater value, both in pressure ratio and isentropic efficiency; data presented in figure 6 and figure 7.
Relative Mach number for all RANS cases, at 50% span, presents minor flow field differences at impeller discharge, figure 8. No recirculation areas are developed in the impeller channels, and neither flow separation which could lead to pressure losses or poor performances.

**Figure 8.** Relative Mach number contour for 1.65 kg/s mass flow (grid size 1.5 mil.) at 50% span.

Figure 9 presents the entropy generated in the impeller, in all four RANS cases. Between them there is a slight difference at the impeller discharge, given by the local characteristics of the fluid flow, more precisely by velocity and temperature fields.

**Figure 9.** Entropy (grid size 1.5 mil.).

Figure 10 presents static pressure distribution at 50% span of blade height, for main blade and also for the splitter blade. It can be observed that RSM model provides higher pressure ratio, having the maximum pressure peak on both pressure and suction side; while SST has minimum values for both sides. The only models that provides similar values are SA and V2f.
Figure 10. Static pressure distributions at 50% span for the main blades (a) and the splitter blades (b).

For 90% span only the extremes are presented, figure 11, namely RSM and SST model. At blade leading edge, the static pressure is similar for the suction side, while on the pressure side the differences are significant. Regarding the trailing edge, the situation is quite opposite: differences for the suction side and similar values for pressure side. Thus can be established that flow separation is not a problem.

Figure 11. Static pressure distributions at 90% span for the main blades (a) and the splitter blades (b), for RSM and SST turbulence model.
4. Conclusions
The current paper concerns the impact of mesh size, numerical discretization schemes and different
turbulence models on radial impeller performances. Free stream numerical diffusion was minimized
after a grid dependency test (0.7 mil, 1.5 mil, 5 mil and 9 mil), revealing the best trade-off between
resolution/numerical stability on one hand and convergence time on the other. As turbulence models,
six RANS models were investigated: k-ε, ν^2-f, k-ω Shear Stress Transport (both steady and unsteady).
Spalart–Allmaras and an Explicit Algebraic Reynolds Stress Model. Spalart-Allmaras predicted
slightly higher pressure ratio, but slightly lower efficiency when compared with the SST. On the other
hand, the EARSM predicted much greater efficiency and pressure ratio than all other models tested.
This was interpreted as a problem with the explicit algebraic formulation of the RSM, which
essentially negates the advantages of the method. Further investigation into this matter is warranted,
particularly after experimental confirmation, in order to properly assess which aspects of which
models are closer to the real flow.

5. References
[1] Gibson L, Galloway L, Kim S I and Spence S 2017 Assessment of turbulence model predictions
for a centrifugal compressor simulation Journal of the Global Power and Propulsion Society
1 142–256
[2] Aghaei tog R, Tousi A M and Tourani A 2008 Comparison of turbulence methods in CFD
analysis of compressible flows in radial turbomachines Aircraft Engineering and Aerospace
Technology: An International Journal 80(6) 657–665
[3] Mao J Y, Yuan S Q, Pei J, Zhang J F and Wang W J 2014 Applications of different turbulence
models in simulations of a large annular volute-type pump with the diffuser 27th IAHR
Symposium on Hydraulic Machinery and Systems (IAHR 2014) IOP Conf. Series: Earth and
Environmental Science 22 (2014) 022019
[4] Shah S R, Jain S V, Patel R N and Lakhera V J 2013 CFD for Centrifugal Pumps: A Review of
the State-of-the-Art Procedia Engineering 51 715-720
[5] Chien K Y 1982 Predictions of channel and boundary-layer flows with a Low-Reynolds-number
turbulence model AIAA Journal 20(1) 33–38
[6] Röyttä P, Grönman A, Jaatinen A, Turunen-Saaresti T and Backman J 2009 Effects of different
blade angle distributions on centrifugal compressor performance International Journal of
Rotating Machinery 2009 Article ID 537802 9 pages
[7] Singh H, Fletcher DF and Nijdam JJ 2011 An Assessment of Different Turbulence Models for
Predicting Flow in a Baffled Flow Stirred with a Rushton Turbine Chem. Eng. Sci. 66 5976-
5988 https://doi.org/10.1016/j.ces.2011.08.018
[8] Smirnov PE and Menter FR 2009 Sensitization of the SST turbulence model to rotation and
curvature by applying the Spalart–Shur correction term J Turbomach 2009 131: 041010-1–
041010-8
[9] Bourgeois J A, Martinuzzi R J, Savory E, Zhang C and Douglas A. A 2011 Assessment of Turbulence Model Predictions for an Aero-Engine Centrifugal Compressor J. Turbomach. 133(1) 011025 (15 pages) https://doi.org/10.1115/1.4001136
[10] Joshi J B, Nere N K, Rane C V, Murthy B N, Mathpati C S, Patwardhan A W and Ranade V
2011 CFD simulation of Stirred Tanks: Comparison of Turbulence Models. Part I: Radial Flow Impellers Canadian Journal of Chemical Engineering 89 23-82 https://doi.org/10.1002/cjce.20446
[11] Murthy B N and Joshi J B 2008 Assessment of standard k–ε, RSM and LES turbulence models
in a baffled stirred vessel agitated by various impellers Chem. Eng. Sci. 63 5462–5495
[12] Dragan V, Dumitreescu O, Malael I, Porumbel I, Gherman B and Puscasu C 2018 Turbulence model sensitivity on steady state mapping of a very high pressure ratio compressor stage ICNPAA 2018: Mathematical Problems in Engineering, Aerospace and Sciences, American University of Armenia (AUA), Yerevan, Armenia, AIP Conference Proceedings 2046(1)
[13] Neverov V V, Kozhukhov Y V, Yablokov A M and Lebedev A A 2017 Optimization of a centrifugal compressor impeller using CFD: the choice of simulation model parameters *IOP Conf. Series: Materials Science and Engineering* **232**(2017) 012037 doi:10.1088/1757-899X/232/1/012037

[14] Lakshminarayana B 1991 An assessment of computational fluid dynamic techniques in the analysis and design of turbomachinery The 1990 freeman scholar lecture *ASME J Fluids Eng* **113**(1991) 315-352

[15] McMullan W A and Page G J 2012 Towards Large Eddy Simulation of gas turbine compressors *Prog. Aerosp. Sci.* **52** 30–47

[16] Parneix S, Durbin P and Behnia M 1998 Computation of 3-D Turbulent Boundary Layers Using the V2F Model *Flow, Turbulence and Combustion* **60** 19–46 https://doi.org/10.1023/A:1009986925097