CHEMICAL ENGINEERING TRANSACTIONS
VOL. 70, 2018

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Timothy G. Walmsley, Petar S. Varbanov, Rongxin Su, Jiří J. Klemeš

ISBN 978-88-95608-67-9; **ISSN** 2283-9216

DOI: 10.3303/CET1870117

Design and CFD Modelling of a Low Pressure Turbine for Aeroengines

Christina Salpingidou[a], Charalampos Deralas[a], Dimitrios Misirlis[b*], Zinon Vlahostergios[c], Kyros Yakinthos[a]

[a]Aristotle University of Thessaloniki, Laboratory of Fluid Mechanics and Turbomachinery, Egnatia Street Thessaloniki, Greece
[b]Technological Educational Institute of Central Macedonia, Department of Mechanical Engineering TE, Terma Magnesias, Serres, 62124, Greece
[c]Democritus University of Thrace, Laboratory of Fluid Mechanics and Hydrodynamic machines, Xanthi, Greece
misirlis@eng.auth.gr

During the last decades, the reduction of aeroengines fuel consumption is becoming an issue of high importance, due to economic and environmental reasons. One of the main parameters that can severely affect the fuel consumption and the whole performance of the engine is the efficiency of its turbomachinery components. In this work, a detail design study of a low-pressure turbine (LPT), which is integrated in a turbofan engine, is presented. The LPT is designed based on a thermodynamic cycle analysis of the turbofan engine, based on specific requirements such as: mass flow rate, inlet pressure, temperature, power etc, targeting always to an aerodynamic design of increased efficiency. In the first part, the development of a computational tool (0D analysis) based on the book of Saravanamuttoo et al. (2001) on gas turbine theory and the work by Kacker and Okapuu (1982) on axial-flow turbines, capable to calculate the main geometrical and thermodynamic characteristics of the turbine (e.g. inlet and outlet flow and blade angles, aspect ratio, pitch to chord ratio, root and tip radius, pressures, temperatures and densities at each stage) is presented. In order to have a proper evaluation of the design methodology, a 3D CAD of all the turbine stages is created and detailed 3D CFD computations are performed. The CFD analysis and the results are presented in the second part of this work. The ANSYS commercial software is used for the design, meshing and prediction of the flow field through the turbine. Additionally, grid independency study is conducted also, in order to determine and conclude to the size of grid that will provide grid-independent results. Finally, the results of the 0D analysis are compared with those of the 3D CFD computations. A very close agreement is achieved, with a deviation in efficiency values less than 1 %. Thus, it can be concluded that the 0D computer tool can be used as a first low-computational-cost and accurate presizing tool for the calculation of the engine turbine performance.

1. Introduction

The low-pressure-turbine (LPT) is considered one critical part for the proper and efficient operation of an aero engine. Many researchers are trying to optimize the LPT (Burton et al., 2007) and suggest methodologies for an efficient way for optimization of blade profiles (Trigg et al., 1999). The aim of this work is the conceptual aerodynamic design of a low pressure axial turbine and it is accomplished in two stages. During the first stage a 0D computer code is developed for the calculation of the geometrical characteristics of the blade, the thermodynamic performance and the velocity triangles, for specific inlet conditions and power requirements, aiming always the maximization of LPT efficiency. At the end of this stage, a design which leads to increased efficiency is determined. The main goal and motivation of this work is to show that the 0D methodology is accurate and can lead to an LPT design in a very short time.

During the second stage the calculated geometrical parameters are used for the design of the turbine CAD model in order to carry out a Computational Fluid Dynamics (CFD) analysis. The commercial software ANSYS Turbogrid (ANSYS, 2016) is used for the mesh generation and the commercial CFD software ANSYS CFX

Please cite this article as: Salpingidou C., Deralas C., Misirlis D., Vlahostergios Z., Yakinthos K., 2018, Design and cfd modelling of a low pressure turbine for aeroengines , Chemical Engineering Transactions, 70, 697-702 DOI:10.3303/CET1870117

(ANSYS, 2016) for the modeling of the turbine. A grid independence study is conducted for deciding the sufficient number of computational nodes. Proper comparative values were selected in order to conclude on the deviation between the 0D code and the CFD results and thus, estimate the accuracy of the 0D code.

2. Computational tool based on 0D model

For the conceptual design of the LPT an in-house tool is developed. The main steps of the procedure are shown in Figure 1.

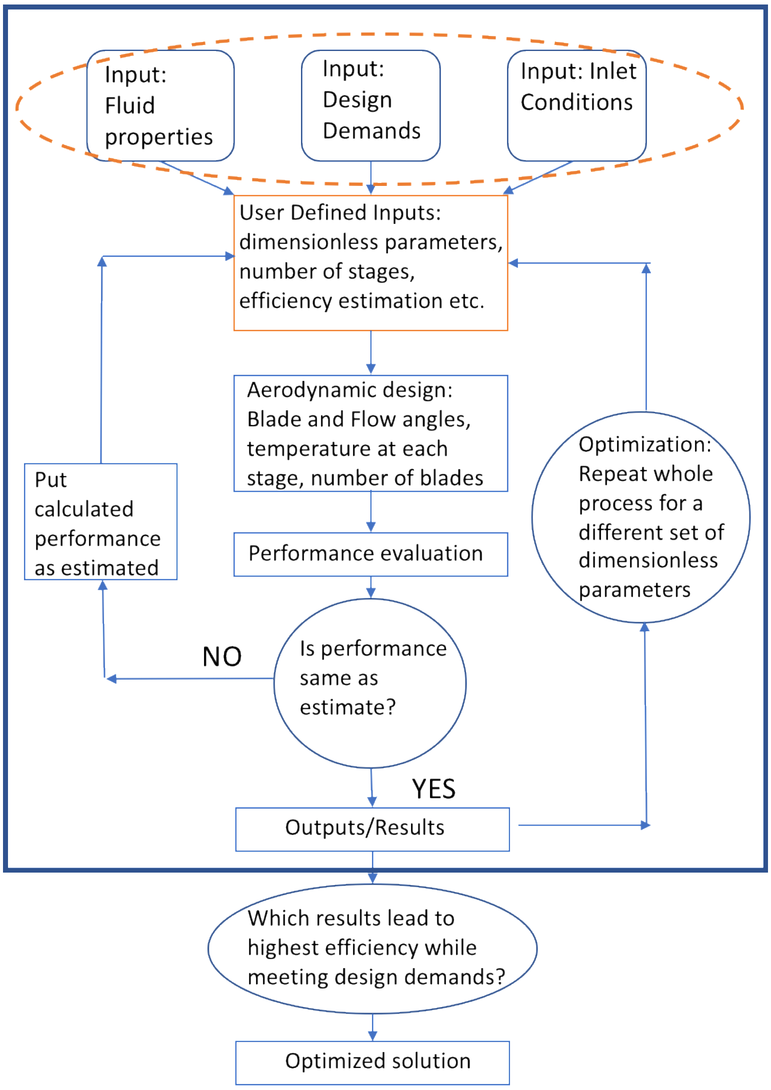


Figure 1: Conceptual design algorithm

Three dimensionless parameters are most commonly used in turbine design mostly affecting the main design of the blades. The first one is called stage loading coefficient or temperature drop coefficient, ψ, and expresses the work capacity of the stage and is given by Eq(1). Parameter C_α is the velocity in the stationary frame and U is the blade linear velocity. Symbols α and β are used for the angles of the velocity triangles. The index 1 is used for the stator entry, 2 for the stator outlet and 3 for the rotor outlet. C_p is the specific heat capacity and ΔT_0 is the total temperature difference.

$$\psi=\frac{2C_p\Delta T_{0s}}{U^2}=2\frac{C_\alpha}{U}(\tan\beta_2+\tan\beta_3) \qquad (1)$$

Another parameter is the degree of reaction, Λ, which represents the ratio of static enthalpy or temperature drop in the rotor to the total enthalpy (or temperature) drop in the stage:

$$\Lambda=\frac{T_2-T_3}{T_1-T_3} \tag{2}$$

For constant axial velocity through the turbine, Eq(1) can be transformed to Eq(3):

$$\Lambda=\frac{C_\alpha}{2U}(\tan\beta_3-\tan\beta_2) \tag{3}$$

Finally, the last parameter is called flow coefficient and it represents the ratio of the axial flow velocity to the blade velocity:

$$\varphi=\frac{C_\alpha}{U} \tag{4}$$

Combining Eq(1) to Eq(4), the angles of the velocity triangles α and β can easily be calculated if these 3 dimensionless parameters are initially defined. These dimensionless parameters constitute the main user defined input. Moreover, providing the linear blade velocity U, all the other velocities can be calculated. Considering the power required for a stage, W_s, the first law of thermodynamics, and an estimation of the stage efficiency, η_s, and of the temperature loss coefficient for the nozzle blades, λ_N the temperature and pressure values can be calculated.
The next step is the calculation of the density values and annulus area at stator and rotor inlet and exit. The density values can be easily calculated from the static pressure and temperature values obtained above with equation of state while the annulus areas can be calculated with continuity equation
The heights (h) and radii of root (r_r), meanline (r_m), and tip (r_t), of the blades can be calculated from the following Equations, where N show the revolutions per minute and A the blade area:

$$r_m=\frac{U}{2\pi N} \tag{5}$$

$$h=\frac{AN}{U} \tag{6}$$

$$\frac{r_t}{r_r}=\frac{r_m+\frac{h}{2}}{r_m-\frac{h}{2}} \tag{7}$$

For the selection of pitch to chord ratio, s/c, and aspect ratio, h/c, of the blades, already existing literature data are used Saravanamuttoo et al. (2001). Since the main geometrical characteristics and the velocity triangles of each blade have been calculated the blade angles should also be calculated. A choice must be made for the incidence ($i=\alpha_{in}-\alpha'_{in}$), the deviation ($\delta=\alpha_{out}-\alpha'_{out}$) angles, the stagger angle and maximum airfoil thickness to chord ratio. For determining the incidence, various correlations exist. An empirical correlation is given by Eq(8), Kenny (2005) :

$$i=0.175\alpha_{in}-1.5 \tag{8}$$

where α_{in} is the inlet flow angle, α'_{in} is the inlet blade angle, α_{out} is the outlet flow angle and α'_{out} is the outlet blade angle.
For the estimation of the blade stagger angle and the maximum airfoil thickness to chord ratio, graphs given from Kacker and Okapuu (1982) can be used. The selection of appropriate deviation angle is also important, since it is directly related with the performance of the turbine. The formula for the deviation angle calculation is given from Zhu and Sjolander (2005), Eq(9), where Φ is the stagger angle and t_{max} is the maximum thickness of airfoil and is valid from Mach numbers up to 0.6.

$$\delta=\frac{\left(s/c\right)^{0.05}(\alpha_{in}+\alpha_{out})^{0.63}\cos^2(\Phi)\left(t_{max}/c\right)^{0.29}}{30+0.01(\alpha'_{in})^{2.07}\tanh\frac{Re}{200000}} \tag{9}$$

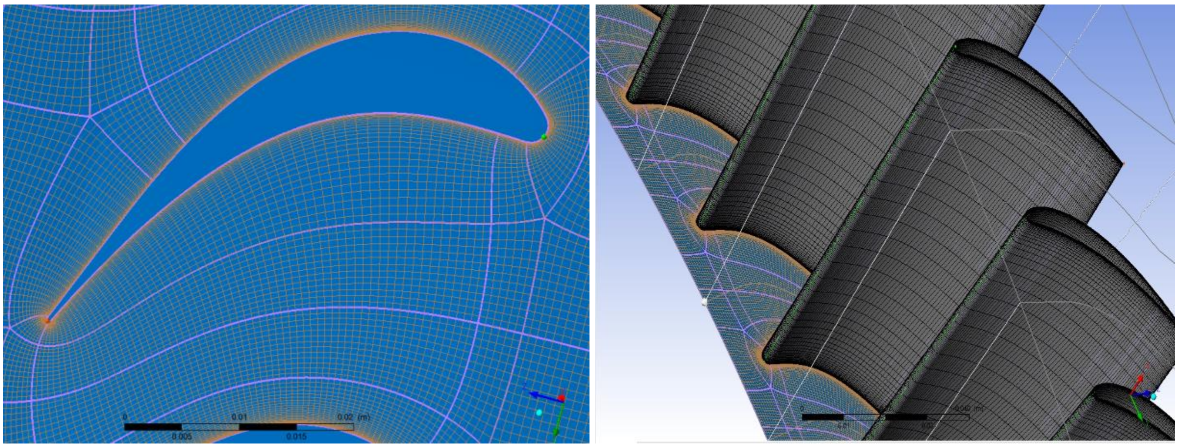
Up to this point, the basic elements of the airfoil design are complete, and a calculation of the LPT stage efficiency is needed, so that the previous calculations, based on an estimated efficiency, can be performed again based on the new calculated efficiency. For the efficiency calculation, a method outlined by Kacker and Okapuu (1982) is adopted.

3. CFD model

In order to obtain results from a CFD analysis the next steps are followed:

1. Turbine stages design with the use of ANSYS BladeGen
2. Mesh creation with the use of ANSYS TurboGrid
3. Grid independence study
4. CFD simulation with the uses of ANSYS CFX

The first step is the creation of the blade CAD model and the periodic flowpath, as shown in Figure 2. Three computational grids are considered in an initial simulation only for one stage, in order to determine the sufficient number of computational elements. More specifically, for the stator there are three grids consisting of 300×10^3, 600×10^3 and 1.2×10^6 elements. For the rotor, the three grids consist of 450×10^3, 900×10^3 and 1.8×10^6 elements respectively. After a comparison between the CFD results using three study, the final selected grid consists of 600×10^3 elements for the stator and 900×10^3 elements for the rotor. These values are selected due to the fact that an increase of the mesh quality beyond the aforementioned values lead to insignificant changes in the flow simulation results (less than 0.2 %), meaning that using this element number a grid independent solution is obtained.



(a) (b)

Figure 2: (a) Mesh of fourth stage rotor. (b) Mesh of fourth stage rotor, 3D view

Steady state simulations are performed. The total pressure inlet (322 kPa) and the mass flow at the outlet (17 kg/s) are provided, as boundary conditions. Since high accuracy in the solution was the scope of this work, the chosen turbulence model for this simulation was the Shear stress transport model of Menter (1994). This is a two-equation eddy viscosity model that combines a k-ε model away from the wall boundaries and a k-ω model in the near wall regions. The SST model was designed to give highly accurate predictions of the onset and the dimensions of flow separation under adverse pressure gradients, with the proper modelling of the behaviour of the turbulent shear stress transport by the introduction of appropriate eddy-viscosity limiters. This lead to a major improvement regarding flow separation prediction. The superior performance of this model has been demonstrated in a large number of validation studies, (Costa Rocha et al., 2016)

For the computations, the turbulence intensity is defined at 5 %. Simulations with very complex and. turbulence-generating components upstream will have a very high incoming turbulence level. For example, a high-pressure turbine just downstream of a turbulence generating combustor might have incoming turbulence levels up to 20 %. Fans and low-pressure compressors with not many components upstream might have as low incoming turbulence level as 1 %. For components between these two extreme examples, like high-pressure compressors or as this example, low-pressure turbines, a turbulence level of around 5 % seems realistic. The turbulence length-scale is often even more difficult to estimate than the turbulence level. The best way to make a realistic assumption is by using the geometrical characteristics of the upstream components. The incoming turbulence length-scale can be estimated as say the thickness of upstream blades or somewhere between 2 % and 20 % of the incoming channel height. In the present case, the length scale is considered equal to the mean maximum blade thickness at the mean radius.

The mesh of the stator is modeled as a stationary domain and the mesh of the rotor as a rotating domain. The interface-interaction between these two domains is performed using a general connection interface model. This is a powerful model and can be used (and is actually necessary) to apply both a frame change and a flow path change/pitch change (when both domains do not present an exact periodic flow path as aforementioned) in the interaction between the stator and the rotor. When a general connection interface is used, the frame change/mixing model and the pitch change must be specified. The Stage model (also known as the Mixing-Plane model) as a frame change model is used. It performs a circumferential averaging of the fluxes through bands on the interface. Steady-state solutions are then obtained in each reference frame. This model enables steady-state predictions to be obtained for multi-stage machines. The stage averaging at the frame change interface incurs a one-time mixing loss. This loss is equivalent to assuming that the physical mixing supplied by the relative motion between components is sufficiently large to cause any upstream velocity profile to mix out prior to entering the downstream machine component. The Stage model usually requires more computational time than other models to converge.

4. CFD results

Figures 3 summarizes some of the most important CFD results. Figure 3 shows the pressure distribution and. The CFD results are in very close agreement with those of the 0D methodology. The pressure distribution in Figure 3 is similar with the results of the 0D methodology.

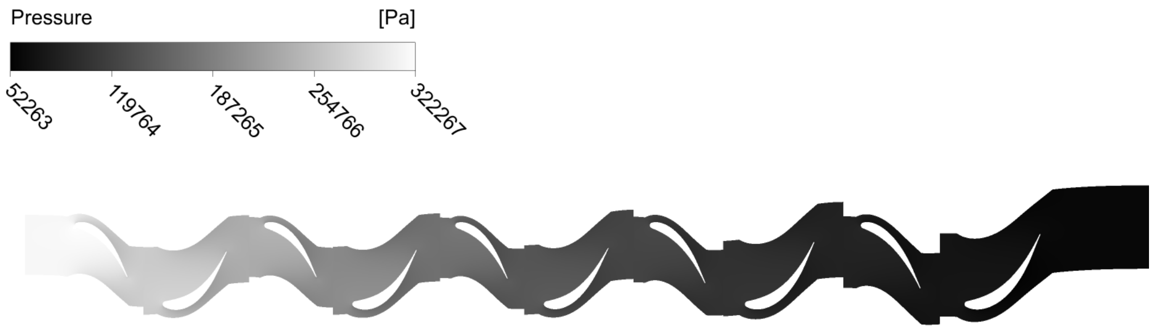
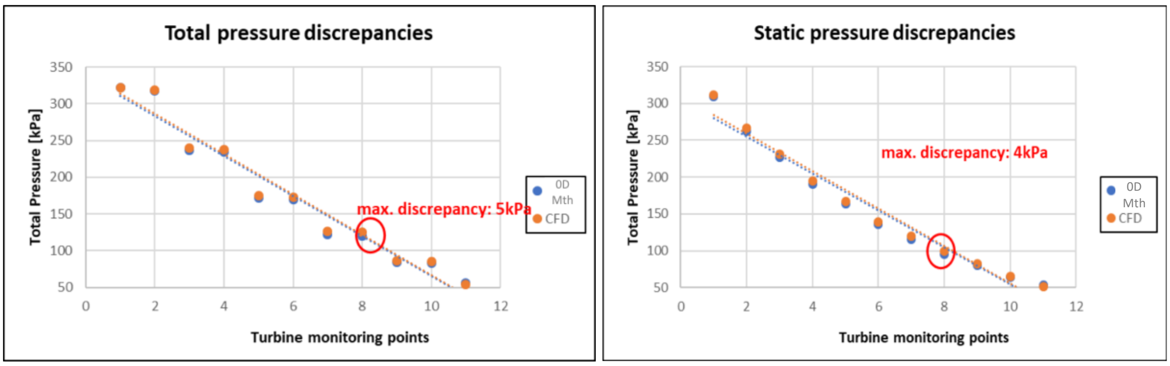


Figure 3: Pressure profile



(a) (b)

Figure 4: (a) Total pressure discrepancies between 0D code and CFD results. (b) Total pressure discrepancies between 0D code and CFD results

Finally, Figure 4 provide a comparison of the CFD results and the 0D computational tool, the approximation line are added in order to show the deviation between the two results. As it can be seen, the CFD results are in good agreement with the zero-dimensional tool. Discrepancies could be contributed to the fact that the 0D analysis cannot predict the velocity component in the third dimension and also the selection and modification of the total area size brings a significant change to the flow field. The modification of areas according to the blade height is not entirely accurate and also does not take into account the blockage imposed by the boundary layer.

Sutherland's formula for the calculation of the viscosities could also bring minor changes, since there is a small variation of the viscosity values taken from the table to the viscosity values calculated by the formula.

5. Conclusions

The main conclusions are the following.

- The 0D methodology is time efficient and can lead to results in very short time.
- The CFD results are in very good agreement, with a deviation less than 5 % with those of the 0D methodology.
- Therefore, the 0D results can be used for the conceptual design of an LPT.

Acknowledgments

Christina Salpingidou would like to thank Alexander S. Onassis Public Benefit Foundation for the scholarship.

References

ANSYS® Academic research, 2016, Release 16.2, ANSYS, Inc., < www.ansys.com/products/fluids/ansys-fluent>, accessed 08/06/2018.

Burton S.A., Prahash C., Machnaim J., 2007, Multistage low pressure turbine airfoil shape optimization using the C3 Process, 48th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference 2007, doi:10.2514/6.2007-1914.

Kacker S.C., Okapuu U., 1982. A mean line prediction method for axial flow turbine efficiency, Journal of Engineering for Power, 104, 111-119.

Kenny S., 2005, Development of a multi-disciplinary design tool for axial flow turbines, Mater Thesis, University of Carleton, Institute for Mechanical and Aerospace Engineering, Ontario, Canada.

Menter F.R., 1994. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32, 1598-1605,.

Costa Rocha P.A., Barbosa Rocha. H. H., Moura Carneieo F. O., Vieira da Silva. M. E., Freitas de Andrade C., 2016, A case study on the calibration of the k–ω SST (Shear stress transport) turbulence model for small scale wind turbines designed with cambered and symmetrical airfoils, Energy, 97, 144-150.

Saravanamuttoo H.I.H., Rogers G.F.C., Cohen H., 2001. Gas turbine theory, 5th edition, Prentice Hall, Great Britain.

Trigg M.A., Tubby G.R., Sheard A.G., 1999, Automatic genetic optimization approach to two-dimensional blade profile design for steam turbines, Journal of Turbomachinery, 121, 11-17.

Zhu J., Sjolander S.A., 2005, Improved profile loss and deviation correlations for axial-turbine blade rows, ASME Turbo Expo 2005: Power for Land, Sea, and Air, 6, 783-792.