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/CET1870127

Investigation of Heat Transfer and Flow Field Development Around a Low-Pressure Turbine Blade with the Use of Open Source CFD Tools

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

[a]Technological Educational Institute of Central Macedonia, Department of Mechanical Engineering TE, Terma Magnesias, Serres, 62124, Greece
[b]Democritus University of Thrace, Department of Production and Management Engineering, Laboratory of Fluid Mechanics and Hydrodynamic machines, Xanthi, 67100, Greece
[c]Aristotle University of Thessaloniki, Department of Mechanical Engineering, Laboratory of Fluid Mechanics and Turbomachinery, Egnatia Street, 54124, Greece
misirlis@eng.auth.gr

The present paper is focused on numerical investigations of the heat transfer and flow field development around a low pressure turbine blade (T106) with the use of computational fluid dynamics (CFD) tools and methods. The CFD computations were performed with OpenFoam open source CFD software with the use of the Shear Stress Transport (SST) low-Reynolds number turbulence model. At the first part of the present work, the CFD computations were validated in relation to available detailed isothermal experimental measurements and useful conclusions about the accuracy of the modelling approach and the flow field development were derived. At the next stage, additional CFD computations were performed for flow and thermal conditions adapted in order to reflect more closely the low-pressure turbine blade operation. In these CFD computations the interaction between the velocity and thermal boundary layers and their effect on the heat transfer was quantified through the derivation of the distribution of the local Nusselt number on the T106 surface, which was compared in relation to available correlations from open literature. Through the analysis of the CFD results it was possible to identify the regions on the low-pressure turbine surface in which decreased heat transfer performance was presented as a result of the non-optimum flow and thermal field development. Furthermore, through the CFD computations secondary flow effects resulting in operational efficiency decrease were identified. The elimination of these sources of operational decrease is planned to be the main target of future research efforts targeting the development of methodologies for the design of highly efficient turbomachinery components.

1. Introduction

As turbomachinery components have reached a high degree of efficiency, the remaining optimization margin becomes more difficult to be achieved solely with the use of conventional 1-D design approaches. The use of higher fidelity CFD numerical tools can provide significant benefits in these efforts since they can capture secondary effects and flow field development details which cannot be included in 1-D approaches and otherwise cannot be considered in the turbomachinery components design phase. Thus, modifications and non-intuitive design refinements can be incorporated in the design of turbomachinery components, in order to quantify their effect, targeting performance optimization. Furthermore, the use of CFD numerical tools can be used in order to further enhance correlations and functions applicable to 1-D approaches in order to sensitize them in relation to flow and thermal conditions and provide more realistic and accurate results.

Currently, the use of CFD tools and their capabilities are not exploited in their full extent mainly due to the associated cost related with purchase of commercial license cost and the cost for technical support or training which might be required in order to reliably apply CFD methods in practical engineering problems. In recent years, various open source freely available, CFD software have been presented which provide unrestricted CFD usage potential so as to apply CFD as a cost-effective high fidelity design optimization tool, as shown in Medina

Please cite this article as: Misirlis D., Vlahostergios Z., Salpingidou C., Yakinthos K., 2018, Investigation of heat transfer and flow field development around a low-pressure turbine blade with the use of open source cfd tools , Chemical Engineering Transactions, 70, 757-762 DOI:10.3303/CET1870127

et al. (2015). In this direction, the recent development of open source or freely available pre-processing and post-processing platforms and supplementary tools provides practical (especially for the grid creation stage in complex geometries) and relatively time-efficient workflow for all design optimization analysis stages. The use of open source software results in specific advantages and disadvantages for the user in comparison with commercially available software. Some typical advantages, as described in detail in Medina et al. (2015), include: a) no cost for usage, b) availability of source code, c) customization potential, d) open source community support and e) configurations with the use of readable text files through which linking with other third party software can be established. On the other hand, there are some disadvantages such as: a) a usually steep learning curve, b) limited documentation, c) difficulties in software setup, d) fees for detailed technical support which may apply and e) limited Graphical User Interface capabilities for pre-processing.

For all these reasons, the interest in the use of open source tools for research activities is increasing in international community, especially since these tools are becoming more user-friendly. Similar efforts are presented in the works of Horvath at al. (2009) and Gentile et al. (2016). This trend is expected to be further increased in near future as more workable and time efficient tools are created and more research works are performed with the use of open source tools, showing that open source tools can be efficiently applied for practical engineering problems. Such an effort is presented in this work through the numerical investigation of the heat transfer and flow field development around the T106 low pressure turbine blade. The numerical investigation was performed with the use of open source CFD tools and methods, using the OpenFoam version 4.0 (2016) open source CFD software, while for the turbulence modelling the Shear Stress Transport (SST) low-Reynolds number turbulence model of Menter (1994) was applied. The creation of the computational grid and the analysis of the CFD results were performed with the freely accessible Salome (2016) platform and Paraview (2016) software respectively, following also the same workflow for pre-processing, CFD computations and post-processing analyses with the use of open source tools, as also presented in Medina et al. (2015). At the first part of this work, 2D CFD computations were performed for isothermal conditions which were validated in relation to detailed experimental measurements carried out previously in the experimental facilities of the laboratory of Fluid Mechanics and Turbomachinery of Aristotle University of Thessaloniki, as presented in Sideridis et al. (2011). At the next stage, additional CFD computations, including heat transfer, were performed, for flow and thermal conditions reflecting more closely the low-pressure turbine blade real operation. The main target of the present work is to clearly present to the engineering community a useful roadmap, based only on open source tools, through which results of high accuracy can be derived and further processed in an user-friendly and time-efficient manner for applications of increased engineering interest, such as the ones related with turbine blade cooling.

2. CFD computations

2.1 Computational grid, boundary conditions and isothermal CFD computation

The CFD computations were performed following a 2D steady flow approach with the use of OpenFoam version 4.0 (2016) open source CFD software. For the turbulence modelling the Shear Stress Transport (SST) low-Reynolds number model of Menter (1994) was applied. The computational grid was performed with the freely accessible open source Salome platform software. Typical views of the computational domain with the boundary conditions types and the computational grid are presented in Figures 1 and 2.

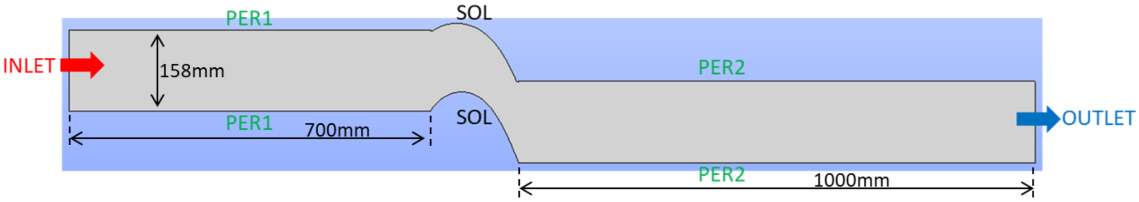

Figure 1: The computational domain with the boundary conditions types

For the CFD computations two 2D computational grids were used, the basic grid consisting of ~100000 computational points and the denser grid consisting of ~280000 computational points. As it can be seen in figure 2, a ‘structured-like’ approach was used near the T106 turbine walls in order to better resolute the regions in which boundary layers are developed and to ensure that the non-dimensional y+ values in the first cells near the walls were close to unity in order to respect the requirements set by the applied turbulence model (the maximum y+ value in the CFD computations was ~1.7 while the average one was ~0.7). At the first part of this work, CFD computations were performed for the isothermal conditions presented in the experimental measurements of Sideridis et al. (2011), corresponding to Reynolds number ~65000. These computations were corresponding to isothermal flow conditions and were used for the validation of the suitability of the CFD model.

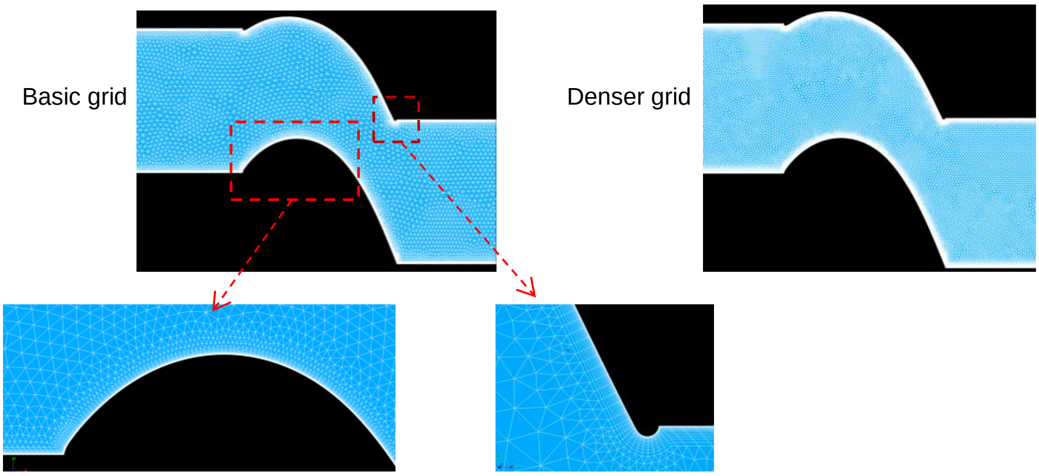

Figure 2: The computational grids for the 2D CFD model with enlarged views near the T106 turbine blade

The applied boundary conditions for the isothermal CFD computations are presented in Table 1. The selection of the turbulence dissipation length scale was based on the suggestions of Vlahostergios et al. (2012).

Table 1: Boundary conditions for the isothermal CFD computations.

Boundary condition	Details
INLET	U_x=4.312m/s, U_y=3.333m/s Turbulence intensity level=1.8%, Turbulence dissipation length scale=0.025*m* Constant static temperature at laboratory conditions
OUTLET	Constant static pressure at laboratory conditions
SOL	Adiabatic walls
PER1, PER2	Translational periodicity
OpenFOAM solver	rhoSimpleFoam
Discretization schemes	2nd order

Typical views of the CFD computations are presented in Figure 3. As it can be seen, the effect of the T106 turbine trailing edge wake is extended for a significant region downstream, affecting the velocity magnitude field. This effect creates a shift on the actual exit flow velocity angle away from the design exit velocity angle (from the velocity triangles design values) resulting to the performance deterioration of the low pressure T106 turbines.

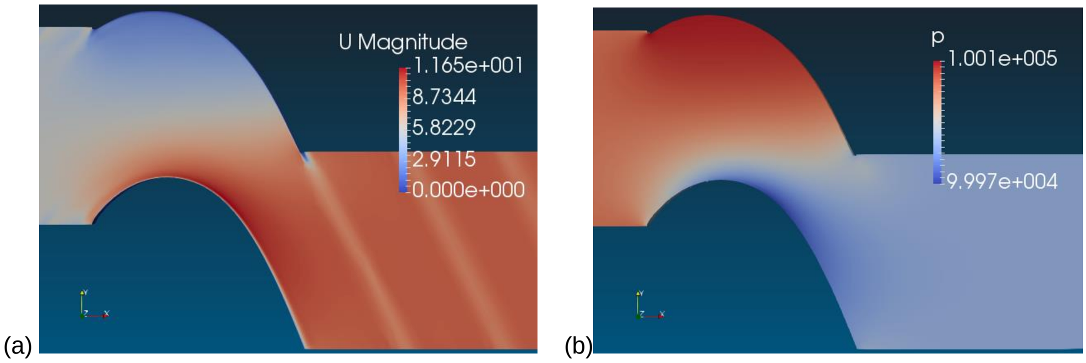

Figure 3: (a) Velocity magnitude field for the isothermal CFD computations (left). (b) Static pressure field for the isothermal CFD computations(right)

The CFD results for both computational grids are in close agreement with the experimental measurements of Sideridis et al. (2011), as digitized from the referenced work, as shown in Figure 4, where a comparison of the streamline velocity magnitude, as defined in Sideridis et al. (2011), right after the T106 trailing edge is presented. Regarding the flow velocity deviation angle, the CFD results showed a ~2° deviation angle which is also in close agreement with the 1.7° measured deviation angle in the work of Sideridis et al. (2011).

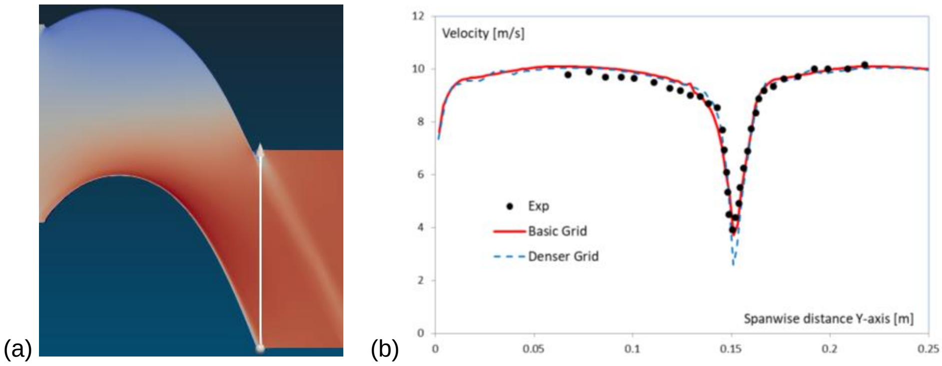

Figure 4: Streamline velocity comparison between CFD results and experimental measurements right after the T106 trailing edge (the CFD results have been repeated for better illustration with experimental measurements)

2.2 CFD computations with heat transfer

At the second part of this work, additional CFD computations were performed, this time including heat transfer, for flow and thermal conditions reflecting more closely the low-pressure turbine blade real operation, following conclusions from the work of Salpingidou et al. (2017) and also from Choi et al. (2004). The applied boundary conditions for the heat transfer CFD computations are presented in Table 2. Typical views of the CFD computations are presented in Figures 5 and 6.

Table 2: Boundary conditions for the heat transfer CFD computations.

Boundary condition	Details
INLET	U_x=23.737m/s, U_y=18.346m/s (corresponding to velocity magnitude of 30m/s and inlet flow angle of 37.7 degrees) Turbulence intensity level=10%, Turbulence dissipation length scale=0.0*25m* Static temperature = 1500K
OUTLET	100000Pa
SOL	Constant heat flux to limit average temperature at ~1250K
PER1, PER2	Translational periodicity
OpenFOAM solver	rhoSimpleFoam
Discretization schemes	2nd order

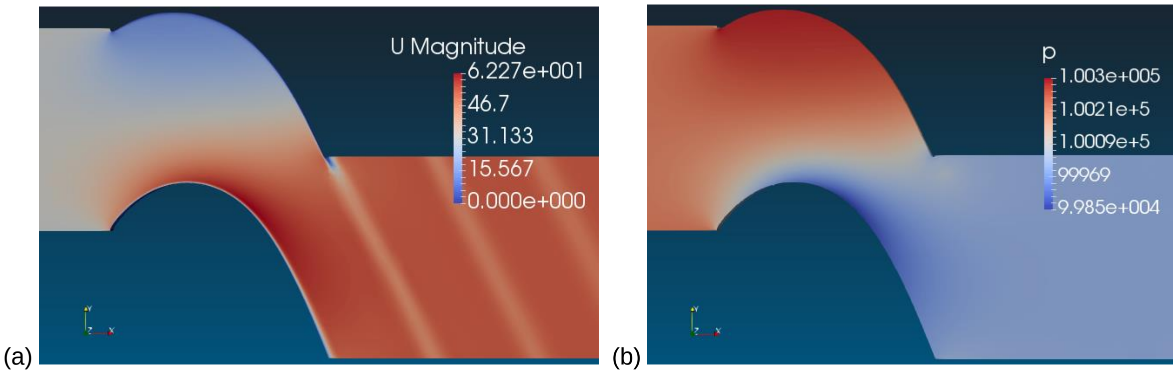

Figure 5: (a) Velocity magnitude field for the heat transfer CFD computations, lighter colour on blade surface indicates velocity boundary layer development (left). (b) Static pressure field for the heat transfer CFD computations (right)

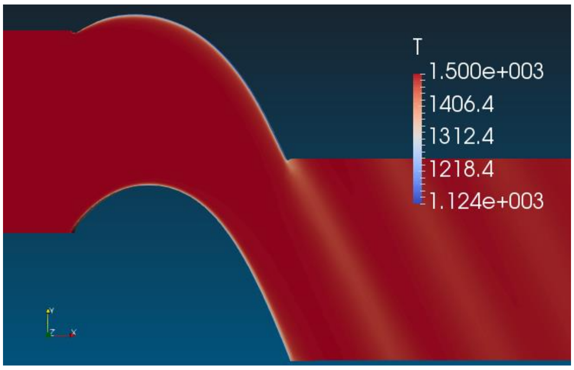

Figure 6: Static temperature field for the heat transfer CFD computations (lighter colour on blade surface indicates thermal boundary layer development)

As it can be seen, the effect of the T106 turbine trailing edge wake is still noticeable since the wake is extended for a significant region downstream, affecting the velocity magnitude and flow angle, static pressure and static temperature fields. Furthermore, the thermal and velocity boundary layers are clearly indicated showing a smooth development throughout the T106 blade surface. For the quantification of these effects on the heat transfer, the local Nusselt number distribution on the T106 blade suction side is presented in Figure 7, in comparison also in relation to open literature correlations for flat plate and turbulent flow, following the suggestions of Cunha (2018).

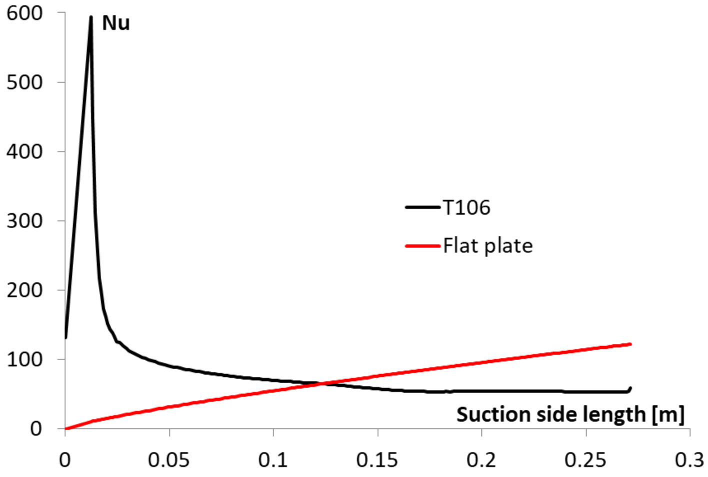

Figure 7: Nusselt number distribution on the T106 blade surface (suction side)

As it can be seen in Figure 7, significant differences on the local Nusselt number distribution are presented as a result of the T106 blade profile and the imposed pressure gradient in the streamline direction, due to the T106 cascade positioning. Furthermore, significantly increased local Nusselt number (and local heat transfer coefficient) values are presented near the T106 leading edge. In this region, a flow stagnation point is presented and the boundary layers are still very limited in height, presenting very low thermal resistance values, leading to high local Nusselt number values. These high values are reduced as the flow moves towards the trailing edge of the T106 suction side, where the boundary layers are further developed.

As a result, the average Nusselt number value of the T106 turbine blade is higher than the one calculated by the flat plate correlation by ~22%, since the latter does not include the near stagnation point effects and underestimate the heat transfer on the first half of the T106 turbine blade. Finally, the Nusselt number distribution of the T106 turbine blade is also very similar qualitatively and close quantitatively to the one presented in the work of Choi et al. (2004) for similar conditions.

3. Conclusions

At the present paper CFD computations were performed around a low pressure turbine blade (T106) with the use of open source CFD tools. The CFD computations were performed with the OpenFoam open source CFD software with the use of the Shear Stress Transport (SST) low-Reynolds number turbulence model. For the validation of the CFD results previously carried-out, detailed isothermal experimental measurements were used showing very close agreement both qualitatively and quantitatively, between CFD results and experimental measurements. At the next step, CFD computations were performed for conditions including heat transfer. These CFD computations were used for the quantification of the effect of the boundary layers development on heat transfer, through the derivation of the local Nusselt number distribution on the T106 blade surface.

Furthermore, the CFD analysis with the open source tools facilitated the accurate (and cost-efficient) identification of: i) secondary flow effects resulting in operational efficiency decrease, such as the accurate calculation of the exit flow deviation angle and ii) regions on the low-pressure T106 turbine surface with decreased heat transfer coefficients. The elimination of these sources of operational performance decrease is planned to be the main target of future research efforts. In this direction, the results and conclusions of detailed CFD computations will be fully exploited targeting the development of methodologies for the design of highly efficient turbomachinery components through the use of cost-efficient open source CFD tools.

Acknowledgments

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

References

Choi J., Teng S., Han J.C., Ladeinde F., 2004, Effect of free-stream turbulence on turbine blade heat transfer and pressure coefficients in low Reynolds number flows, International Journal of Heat and Mass Transfer 47, 3441–3452.

Cunha F. J., 4.4 Heat Transfer Analysis, <www.netl.doe.gov/File%20Library/Research/Coal/energy%20 /turbines/handbook/4-4.pdf> accessed 02.04.2018.

Gentile G., Debiagi P.E.A., Cuoci A., Frassoldati A., Faravelli T., Ranzi E., 2016, A cfd model for biomass flame-combustion analysis, Chemical Engineering Transactions, 50, 49-54.

Horvath A., Jordan C., Lukasser M., Kuttner C., Makaruk A., Harasek M., 2009, CFD simulation of bubble columns using the vof model - comparison of commercial and open source solvers with an experiment, Chemical Engineering Transactions, 18, 605-610.

Medina H., Beechook A., Saul J., Porter S., Aleksandrova S., Benjamin S., 2015, Open source Computational Fluid Dynamics using OpenFOAM, Light Aircraft Design: Methods and Tools 2015 (invited paper), Royal Aeronautical Society, General Aviation Conference, London, 1-10.

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

Open source field operation and manipulation-openfoam version 4.0, User Guide, 24th June 2016, <openfoam.org>, accessed 02.04.2018.

PARAVIEW open-source, multi-platform data analysis and visualization application, version 5.3, The ParaView Guide, <www.paraview.org/> accessed 02.04.2018.

SALOME, open source integration platform for numerical simulation, version 7.8.0, 2016, <www.salome-platform.org> accessed 02.04.2018.

Salpingidou C., Tsakmakidou D., Vlahostergios Z., Misirlis D., Flouros M., Yakinthos K., 2017, The effect of turbine blade cooling on the performance of recuperative cycles for gas turbines applications, Chemical Engineering Transactions, 61, 1027-1032.

Sideridis A., Yakinthos K., Goulas A., 2011, Turbulent kinetic energy balance measurements in the wake of a low-pressure turbine blade, International Journal of Heat and Fluid Flow 32, 212–225.

Vlahostergios Z., Sideridis A., Yakinthos K., Goulas A., 2012, Performance assessment of a non-linear eddy-viscosity turbulence model applied to the anisotropic wake flow of a low-pressure turbine blade, International Journal of Heat and Fluid Flow 38, 24–39.