Effect of the Turbulence Modeling in the Prediction of Heat Transfer in Suction Mufflers

Tadeu Tonheiro Rodrigues
rodrigo_link@embraco.com.br

Rodrigo Link

Follow this and additional works at: http://docs.lib.purdue.edu/icec

http://docs.lib.purdue.edu/icec/2062

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html
Effect of the Turbulence Modeling in the Prediction of Heat Transfer in Suction Mufflers

Tadeu Tonheiro RODRIGUES¹, Rodrigo LINK²*

EMBRACO, Research & Development Group, Joinville, Santa Catarina, Brazil
¹Tadeu_T_Rodrigues@embraco.com.br
²Rodrigo_Link@embraco.com.br

*Corresponding author

ABSTRACT

The well-grounded knowledge of the available numerical methods is of fundamental importance to get advantage of computational aided design. Concerning compressor development, the employment of Computational Fluid Dynamics is nowadays a must, however, very little attention has been given to the importance of the turbulence modeling in the prediction of flow behavior and, moreover, the heat transfer. The present paper reports an investigation of the turbulence models influence in the heat transfer prediction of the compressor suction system, specifically the suction muffler. The importance of the correct heat transfer modeling in the suction muffler represents great importance for the computation of the volumetric efficiency, additionally, influences significantly the thermal profile and suction valve behavior. In order to present such influence, the turbulent flow and heat transfer is modeled in the suction muffler of a hermetic refrigeration compressor. Several turbulence models are employed, including standard engineering models as the \( k-\varepsilon \) and SST. The results are treated to observe the gas superheating during the flow through the suction muffler, which is performed through the heat flux mapping, outlet suction muffler temperature variations and heat transfer coefficient on muffler walls. Standard care is taken in consideration with a mesh refinement study to reduce numerical uncertainties and with the near wall treatment suitable for each model used in the investigation.

1. INTRODUCTION

Interest in compressor heat transfer has increased significantly in the last ten years, fact observed mainly by the expansion of publications in the main compressor conferences. The relevance of superheating handling in compressor design has taken great importance once much effort was employed to optimize valves dynamics, as well to reduce power losses in the suction and discharge filters, reaching high levels of manifold efficiency. Considering the current scenario, reduce the superheating represents a great source for compressor improvement.

In an early, but important, review Shiva Prasad (1998) compiled the available information about heat transfer effects in compressor performance, as well about theoretical and experimental works, where the author pointed out that much little attention was given for the phenomena applied to compressor performance design guide, except for reliability concerns. Since the above mentioned publication, several works about the subject have appeared, mainly about analysis methods, as reviewed by Ribas Jr et al. (2008), where the authors focused on descript the advantages and drawbacks of experimental, theoretical and numerical approaches. One can note when compare the two reviews, is the growing importance of computational fluid dynamics (CFD) as numerical technique. The main advantage of CFD is the possibility to virtually prototype new proposals and deal with different layouts solving all the phenomena issues with good precision. In the review of Ribas Jr et al (2008) are summarized the main achievements of the employment of CFD techniques for thermal managements. It is observed very little attention concerning turbulence modeling effects in the heat transfer prediction, most of the works are based in literature guidelines for turbulent heat transfer simulation. It is not unusual face different behavior for the numerical methods than those predicted by theory, so it is important to compare different numerical approaches in order to evaluate the sensibility of the solution.
The main goal of the present works is related with the questions raised above, and presents the results of the study of turbulence modeling effects in the turbulent flow and heat transfer simulation in the suction muffler of a hermetic compressor. The suction muffler was chosen due its importance in the compressor volumetric efficiency, which is directly affected by superheating effects. For this purpose six popular turbulence models were chosen, considering their importance in historical developments, current application in engineering simulation problems, CFD guidelines for simulation of turbulent flow and heat transfer problems and implementation in popular CFD software.

The performance of the turbulence models are measured analyzing the outlet temperature simulated confronted with experimental data. Additionally, the heat balance between the muffler boundaries and the overall calculated wall heat transfer coefficients are also presented in order to investigate the differences among the patterns of the heat exchange phenomena predicted by turbulence modeling.

2. NUMERICAL SOLUTION

2.1 Mathematical formulation

The solution of the tridimensional, steady and compressible flow of a viscous Newtonian fluid in a suction muffler of hermetic compressor is governed by the mass conservation, momentum and energy equations. The closure for the equation system is performed by a real gas equation of state for the refrigerant.

Concerning issues about turbulent flow and heat transfer simulation, Direct Numerical Simulation (DNS) and Reynolds Average Navier-Stokes Equations (RANS) procedures were employed. The DNS consists of the direct simulation of governing flow equations without turbulence modeling, which in situations of turbulent flow the convergence is slow and can leads to unreliable results, except if very dense grids are employed to solve effectively all scales of length, velocity and time. The RANS procedure is based on the Reynolds decomposition of the Navier-Stokes equations, which separates the average flow behavior from the unsteady contributions. The RANS models are divided in the groups: Eddy Viscosity and Reynolds Stress Models.

The Eddy Viscosity models are the most popular in engineering flow simulations, whose main models are: Standard k-ε, RNG k-ε, k-ω, Shear Stress Transport (SST). These models include the turbulence effects in the mean flow solution computing a turbulent viscosity which is summed to the molecular viscosity instead solve the turbulent Reynolds Stresses present in the RANS equations. The Standard k-ε (Lauder and Spalding, 1974) has proved to be suitable for most of the engineering flows, is robust and provides in the average good results, however, fails to predict swirling flows with adverse pressure gradient and is suitable for fully turbulent flows. The RNG k-ε (Yakhot et al., 1992) provides improvements to deal with the deficiencies of the standard model through modifications in the ε equation, providing better predictions for rapidly strained flows, it is also suitable for flows with low Reynolds numbers. The associated problems with the turbulence eddy dissipation ε of turbulent kinetic energy k lead to the development of the k-ω models (Wilcox, 1994), which are based on the turbulence eddy frequency ω. Models based in the ω term are well fitted to deal with low Reynolds numbers problems without special care with the wall treatment, although are strongly dependent of the free stream boundary conditions and in these cases has presented poor resolution. The SST model (Menter, 1994) employs the k-ε and the k-ω in order to take the best of both. The k-ε is applied for most of the domain far from the walls, once this model doesn’t present strong sensibility to the free stream boundary conditions, although, in the vicinity of the wall the ε equation is transformed in the ω equation for accurate near wall flow prediction.

The Reynolds Stress Models, as properly the name introduces, solve directly the six turbulent Reynolds Stresses through evaluation of a transport partial differential equation for each stress component. It is considered the second-order closure problem and, in theory, is a more general model than eddy viscosity models, being more suitable for simulation of complex swirling flows. The SSG RSM model (Speziale et al., 1991) is employed once is the most recommended due to its special treatment for the pressure strain rate term.

Due to space restrictions, depicts detailed information about the employed turbulence models is unviable, the reader is direct to traditional references as Versteeg and Malalasekera (2007), Wilcox (1994), Ferziger and Peric (2002) and the related developers articles.
2.2 Solution domain and numerical setup

The turbulent fluid flow and heat transfer process was solved by using the commercial code ANSYS CFX, which is based on the Finite Volume Method to discretize the partial differential equations of the mass conservation, momentum, energy and turbulence.

The suction muffler mesh, presented in Figure 1, was discretized employing predominately tetrahedrons. In the vicinity of the walls inflation method was prescribed generating prisms elements, in order to refining locally to deal with the heat transfer from the wall. The mesh quality was controlled keeping the skewness factor bellow recommended values.

The advection terms are discretized using a High Resolution Scheme and the resultant system of equations is solved through an algebraic multigrid method and coupled strategy. The pressure evaluation correlation is obtained from the Redlich-Kwong library available in the numerical tool.

The boundary conditions are prescribed as mass flow rate at the muffler outlet and opening for muffler inlet and Cavity I and II (Figure 1), which are connections to the compressor cavity whose peculiar functions are concerned to reliability issues as to avoid accumulation of oil inside the muffler and equalize the muffler intern and extern pressures for the starting process. The pressure and the temperature for the opening boundaries where specified based on experimental data, as well the mass flow is the average measured.

At the walls the no-slip conditions is set for velocity, for the temperature an experimentally obtained correlation for the heat transfer between the internal compressor environment and the muffler internal walls is prescribed as equivalent thermal resistance, which covers the convection outside the muffler and the conduction through the polymeric walls.

Concerning the boundary conditions for the turbulence closure, a medium intensity of 5% was prescribed at the clearances of the domain. At the walls specific treatment was employed for $\varepsilon$ and $\omega$ based models. The distribution of $\varepsilon$ near the walls is obtained indirectly by the employment of scalable wall function, which is an improvement of the empiric method proposed by Launder and Spalding (1972). A direct algebraic expression for distribution of $\omega$ near the wall is employed as automatic treatment.

The turbulent heat transfer closure is performed through the Eddy Diffusivity, obtained by the calculation of the eddy viscosity and the turbulence Prandtl number, set as 0.9. Also, at the walls specific treatment for the turbulent heat transfer closure is performed through modeling of the heat flux between the wall and the mean flow. The wall heat flux is calculated by the heat transfer coefficient, based on the non-dimensional wall temperature profile prediction and temperature difference between the wall and near wall fluid. The non-dimensional temperature profile for scalable wall functions is obtained by the log-wall relationship, while for automatic wall treatment the thermal law-of-the-wall function of Kader (1981) is employed.
3. RESULTS

3.1 Influence of mesh size

The simulations were carried out through three different meshes whose sizes are: 288755 (Mesh 1), 329388 (Mesh 2) and 517451 (Mesh 3) nodes. The main aim is to observe the influence of the mesh refinement conjugated with the turbulence modeling.

The near wall quality refinement was monitored through the $y^+$ parameter, which is the dimensionless distance of the first mesh node near the wall. According to literature guidelines, for heat transfer simulations it is recommended keep the $y^+$ value around 1, in the present results the maximum value observed was 2.3.

The parameters observed in the analysis are the outlet temperature and the overall computed wall heat transfer coefficient, which are presented for the three meshes and the six models employed, totalizing eighteen cases.

The predictions for outlet muffler temperature are presented in Figure 2, and reveals that all temperature values are in the range of 52-53.5 ºC, relating very good results compared with temperature measured in experiment, which is 52.9 ºC.

It was observed that each model provided small temperature variation between Mesh 1 and 3; with the biggest one noticed for the RNG $k$-$\varepsilon$ with 0.8ºC deviation, which leads to the conclusion the mesh refinement level used in the simulations is suitable for the current study.

The overall wall heat transfer coefficient, present in Figure 3 bellow, which is the average heat transfer coefficient by the area of the muffler walls, is quite independent with the mesh refinement. The choice of this parameter for the mesh refinement study was due to its dependence of the non-dimension temperature profile, which is dependent on the mesh density near the wall.

One can be noted the significant difference between the wall heat transfer coefficients calculated. It is important to enforce that, except in the laminar case, the wall heat transfer coefficients are modeled. The non-dimensional temperature profile influence is clear once the values calculated can be grouped based on $\varepsilon$ or $\omega$ based models, as consequence of the near wall treatment provide by the scalable and automatic models.

Conclusively, the numerical results showed the mesh independence was achieved and reasonable results are obtained with Mesh 2 for outlet temperature prediction, however, the results yielded by Mesh 3 are used in the next section once the data is available.
3.2 Energy balance on suction muffler
The results presented in section 3.1, focused specifically for the mesh refinement study, showed outlet temperature predictions very close and accurate for all models, in counterpart, the results for the overall wall heat transfer coefficient differs greatly among the approaches. In a first moment, it is supposed that high wall heat transfer coefficients differences should induce significant variations in the outlet temperature result. In order to understand the previous results, the energy balance in the suction muffler was calculated, which is presented in Figure 4, as follows:
Figure 4: Energy balance on suction muffler.

Figure 4 presents the individual contribution of each heat exchange boundary in the total heat transported at outlet boundary. It is clear that no significant differences were computed among the turbulence approaches employed, even for the laminar model, and the main contributors for the inlet gas heating are the opening holes towards the compressor cavity. Physically, these results are quite reasonable, once the muffler is manufactured with insulation material in order to avoid excessive gas superheating. Still, the great differences among the wall heat transfer coefficients are not explained.

The similar values predicted for the heat flux, among the models, indicates different temperature distributions at the wall, once the wall heat transfer coefficient is quite different for all models. It was monitored the average difference between the wall adjacent temperature and near wall fluid temperature, named as $\Delta T$ and presented in Figure 5, in order to observe the temperature field behavior.
In a similar way observed in Figure 3 for the wall heat transfer coefficient, the ΔT values are also connected with the turbulence model basis and also with the near wall flow treatment. However, the greatest difference among the ΔT values obtained is only about 4 °C, which indicates that the temperature field near to the wall is quite close for all simulated data and reinforce the small influence of the wall heat flux in the outlet temperature. However, it is clear the great influence of the turbulence closure for heat transfer simulation near to walls. Additional research is important in order the verify the reliability of the calculated wall heat transfer coefficients, in the case of use these parameters in others approaches, as lumped formulation for thermal profile.

The influence of the heating sources is evidenced observing the temperature contours and velocity vectors, on the transversal medium plane, in Figure 6 for k-ε and SST models. The mass flux from Cavities I and II is easily noted as the main heating source once the highest field temperature field value is located at these boundaries and spread towards the interior of the muffler. The small influence of the wall heat flux is indicated by the small temperature gradients near the walls, corroborating the above mentioned calculations for wall heat flux and ΔT values.

The small influence of the wall heat flux in the main temperature field leads to the conclusion that all models are suitable for simulating the turbulent flow and heat transfer in suction mufflers of hermetic compressors with low rates of heat exchange through the walls, or walls built with insulation materials.

The choice of the models now resides in the computational cost of each model, which immediately eliminates the SSG RSM, once this model employs six additional transport equations for calculate the turbulence stresses. Although the Laminar model was able to achieve good results, it is important to mention that in this study was considered only one case for the mass flow rate, which direct influences the flow patterns and turbulence effects in the heat transfer. Perform simulations with increased mass flow rate and Laminar model can leads to unstable solutions and necessity to refine even more the computational mesh and CPU cost, so the k-ε model is particularly adequate as general model start design studies, as well the RNG k-ε, k-ω and SST models, once they presents similar performance and computational cost.

![Figure 6: Temperature contours and velocity vectors for (a) k-ε and (b) SST models.](image)

4. CONCLUSIONS

In this report was presented the study of turbulence modeling effects in the steady state simulation of turbulent flow and heat transfer through suction mufflers of hermetic compressors. This study was motivated by the growing scenario in compressor development guided to thermal management, which has been carried with computational models and demands adequate choice of the available tools. It was observed good and similar results for all models chosen for the study considering the observation of the outlet muffler temperature, whose importance is connected with the compressor volumetric efficiency, and the heat balance in the muffler. Nevertheless, it was noticed high differences among the wall heat transfer coefficients, which are obtained via turbulence closure for heat transfer near walls, although the small contribution of the heat exchanged through walls do not affect the overall performance of
the models. The similarity among the models performance favors the choice of low CPU cost but robust models as the two equation eddy viscosity models, whose popular examples are the $k$-$\varepsilon$ and SST.

**NOMENCLATURE**

\begin{align*}
k & \quad \text{turbulent kinetic energy} \quad (m^2.s^{-2}) \\
\varepsilon & \quad \text{turbulence eddy dissipation} \quad (m^2.s^{-3}) \\
\omega & \quad \text{turbulence eddy frequency} \quad (s^{-1}) \\
y^+ & \quad \text{dimensionless wall distance} \quad (-) \\
\Delta T & \quad \text{temperature difference} \quad (^\circ C)
\end{align*}

**REFERENCES**


