Validation of numerical models for flow simulation in labyrinth seals

D Frączek¹ and W Wróblewski¹

¹Silesian University of Technology, Institute of Power Engineering and Turbomachinery, Gliwice Konarskiego Street 18, Poland

E-mail: daniel.fraczek@polsl.pl

Abstract. CFD results were compared with the results of experiments for the flow through the labyrinth seal. RANS turbulence models (k-epsilon, k-omega, SST and SST-SAS) were selected for the study. Steady and transient results were analyzed. ANSYS CFX was used for numerical computation. The analysis included flow through sealing section with the honeycomb land. Leakage flows and velocity profiles in the seal were compared. In addition to the comparison of computational models, the divergence of modeling and experimental results has been determined. Tips for modeling these problems were formulated.

1. Introduction

Labyrinth seals are widely used in the rotating machinery. In spite of leakages this type of sealing has many operational advantages. In design process the possible reduction in leakages is nowadays one of the main aims and therefore detailed analysis of flow phenomena in the seal is of great importance to the design of high efficient and reliable turbomachines. The flow structures in the labyrinth seal are very complex. The flow is unsteady with strong jets, flow detachment, and vortex structures. The structures modelling are very important for the evaluation of the seal effectiveness, which is directly related to the possibility of kinetic energy dissipation in seals chambers. The process of seal selection requires conducting an optimization and the numerical model of the flow to identify a solution with the greatest possible precision. CFD calculations are used to resolve the flow structures and provide the optimization of various flow channels. Numerical modelling deserves special appreciation for the design of labyrinth seals in turbines and compressors due to the high costs of the test rigs. The study [1] shows that the use of RANS turbulence models can give solutions which are incompatible with the experimental data.

Numerical calculations are used for flow simulation through the seal for many years. Some authors suggest that the seal flow can be simulated practically only with hybrid LES models [1] although, LES models require higher computational cost. LES errors presented in [1] were below approximately 5 % and predict better the flow physics, errors and scatter between RANS solutions reached levels of 23 %. Steady RANS solvers are still commonly used for seal flow simulation. Comparisons of CFD and the experiment were described in the paper [2] for seal with three fins and step. Authors obtained the relative difference of about 1.7% between the results in case of mass leakage. Straight four fins model with a honeycomb land was analyzed in [3]. The difference between CFD and experiment was at the level of 11%. Authors of the paper [4] compared results for a smooth land seal. Fidelity of the CFD results increased with the number of fins. They also obtained better results for seal land with steps.
CFD results of leakage, in case of straight seal with two fins, were inflated of about 20% comparing to the experiment results.

Complex flow with detachments or heat flow can be modeled using SST-SAS turbulence model. Comparisons [5] show that by introducing von Karman length scale, this model can be an alternative to hybrid RANS/LES in case of complex flow simulations.

The analyzed problem concerns straight-through labyrinth seals model with honeycomb land. The geometrical data were selected on base of the literature sources (occurrence in cited papers) and experimental data availability. In the paper comparison between the CFD results and results of the experiment described in [6], [7] is presented. The RANS turbulence models (k-epsilon, k omega, SST, SST-SAS), commonly used for the industrial calculations, were selected for the analysis. Depending on the selected turbulence model and parameters the independent results may occur. Therefore, the grid independence study was performed for each of the three turbulence models in case of steady-state flow. The comparison of steady and transient solutions for selected grid was presented. The results averaged over time and instantaneous distributions of the velocity field depending on the turbulence model were analyzed.

2. Test cases

For the numerical study two configurations of the straight-through labyrinth seals were chosen, which were well experimentally investigated and described in the literature. The first configuration called Test A has a honeycomb land of diameter 3.2 mm and clearance of 0.51 mm. The labyrinth seal with four tapered fins was investigated by Stocker et al. [6] among many other cases on the static test rig. The definition of geometrical parameters of the seal is presented in Figure 1 and Table 1.

![Figure 1. Geometry of the labyrinth seal Test A](image)

| Table 1. Dimensions of the seal Test A |
|--------------------------------------|
| s   | 0.51 mm |
| t   | 2.79 mm |
| SH  | 2.79 mm |
| HCD | 3.18 mm |
| HCH | 3.81 mm |

The computational domain was simplified and the pitch consist two honeycomb cells in the circumferential direction. The study performed with different pitch dimensions showed the negligible influence on the results. The domain width was equal to 5.5 mm. The computational domain has been
extended at the inlet and outlet. Inlet extension is 30 mm long. It is sufficient to form a velocity profile and to avoid the influence of the inlet vortex on the inlet boundary. The outlet channel (120 mm long) is shifted downstream to reduce the influence of the outlet jet and large recirculation zone on the outlet boundary. Boundary condition was set according to the experimental data. The inlet pressure is calculated from the pressure ratio and outlet static pressure value equal 100 kPa. The inlet turbulence intensity level was set to 5% (medium intensity with $\mu_t/\mu=10$). Epsilon, omega and k values at the boundary for turbulence equations were calculated from turbulence intensity and viscosity ratio [8]. The inlet temperature is 293 K. The air ideal gas model was assumed. The honeycomb walls and channel wall were adiabatic.

Translational periodicity was set on the left and right domain surfaces. The computational domain in the seal region consists of two parts (honeycomb part and main flow part) which are connected by the general grid interface (GGI). The O-grid was used inside the honeycomb structure and the hexahedral mesh was used in the main flow domain. The GGI connection allows the use independent meshes inside the honeycomb and main channel. The GGI connection was successfully implemented in [3], [9], [10], [11]. The mesh independence study was performed for steady state conditions on the four meshes (0.45M, 3.6M, 7.5M, and 15M). The grid was refined in the areas of honeycomb cells and near the fins. The grid cell size was changed proportionally in all directions. The basic parameter compared in the calculations is the discharge coefficient $C_D$, which is defined as:

$$
C_D = \frac{m}{m_{ideal}},
$$

$$
\dot{m}_{ideal} = \frac{p_o A}{\sqrt{T_o}} \frac{2\kappa}{\sqrt{\kappa R(\kappa-1)}} \left[ \left( \frac{1}{\pi} \right)^{\frac{2}{\kappa}} - \left( \frac{1}{\pi} \right)^{\frac{\kappa+1}{\kappa}} \right].
$$

$C_D$ describes the effectiveness of energy dissipation in the seal by comparing a mass flow rate in the seal to the mass flow rate in the isentropic flow through a single ideal nozzle (2) at the same conditions.

The mesh independence study was performed only for one pressure ratio equals 1.5. The stopping criteria for the solution were: discharge coefficient $C_D$ variation less than ±1% and the solution residuals level less than 10E-4. Figure 2 indicates that for meshes with a number of nodes higher than 3.6M there is no significant variation in the $C_D$ with increasing number of nodes. There are slight differences between results depending on turbulence models. The $C_D$ values obtained for the k-epsilon turbulence model are the highest and values for SST and k-omega turbulence model are almost the same. Values obtained for steady state solution are at least higher by about 5.5% from the results of the experiment. The mesh with 3.8M elements which will be used in further calculations is presented in Figure 3.

The geometrical data and experimental results derived from [7] were used for the second test case (Test B). Two fins straight through seal with a honeycomb land was chosen (Figure 4). Because the geometrical data in [7] were given relatively to the fin pitch it was necessary to assume the fin pitch value. It was set to 10 mm. Final seal dimensions are presented in Table 1.

Boundary conditions were set according to the conditions of experiment with assumption of adiabatic conditions. At the inlet the total pressure of 200 kPa and total temperature of 400K were assumed. The inlet turbulence intensity level was set to 5% (medium intensity). The outlet pressure was set from the pressure ratio.
The extensions of the computational domain at the inlet and outlet were similar to the Test A. Sutherland's formula was used to derive the dynamic viscosity and heat transfer viscosity of an ideal gas as a function of the temperature.

The mesh independent study was performed for 6 different meshes with nodes number from about 0.4M to 15M. Mesh independence study was performed for the pressure ratio of $\pi=1.2$. The calculations were performed with the same turbulence models as in the previous case: k-epsilon, k-omega and SST. Discharge coefficient values obtained for the different meshes and turbulence models are presented in Figure 5. The $C_D$ values for meshes with node number greater than 2M are changing very slowly. The difference of about 1.5% is between results for the mesh with 2M and 15M nodes. Meshes with the smallest number of nodes got results only about 5% higher than mesh with the highest number of nodes. The highest value of $C_D$ was obtained for SST turbulence model, the lowest for k-epsilon turbulence model. The mesh with 2M nodes was chosen for further calculations and is presented in Figure 6. The difference between experimental data and calculation results in the case with SST turbulence model was the highest and amounted about 11%.

### Table 2. Dimensions of the seal Test B

|   |   |
|---|---|
| s | 0.75 mm |
| t | 10 mm (set) |
| b | 0.5 mm |
| HCD | 1.6 mm |
| HCH | 8 mm |
| SH | 3.5 mm |
3. Results
3.1 Test case A
Discharge coefficient depending on pressure ratio for steady calculation of Test A is presented in Figure 7. The difference between CFD and experiment results increase with increasing pressure ratio. Values from the calculation increase along one trend line. There is still some difference between turbulence models for each pressure ratio. Highest $C_D$ values were calculated with k-epsilon turbulence model.
There were no available results of pressure or velocity distribution in the seal from experiment so only results between various turbulence models were selected for comparison. Pressure distribution along streamline from inlet to outlet looks similar for all turbulence models (Figure 8). The streamline is not the same in all cases, but it passes through the same seal regions clearances and chamber between fins. There are five clearly visible regions with pressure decrease connected with fins position and honeycomb meridional direction change. There are also similar pressure recovery processes after acceleration. Honeycomb meridional direction change and difference between the average values of effective clearance above each fin caused differences in pressure decrease along a streamline. An example of velocity distribution in the middle of the seal width model cross section is showed in Figure 9. Velocity profile for others turbulence models with steady calculation looks similar. Maximum velocity in cross-section occurs above third fin. The CFD models reproduce single vortices filling chamber between the fins and the honeycomb cells above fins.

Transient calculations were initialized from steady state solution. Time step was set as a function of Courant number, in the case of k-epsilon, k-omega and SST turbulence model with a max Co<5. Final time step values take a value close to 1E-7 s. Using short time step can help to show the smallest
disruption in the flow field but because it extends the time of calculation values of Courant was limited to 5. An SST-SAS turbulence model were used for calculation with a Co=1 as a control case. Simulation requires calculating over 20 000 time steps. Calculation for transient Test A CFD models were limited to the case with π=1.5.

Discharge coefficient from transient results (Figure 10) for k-omega SST and SST-SAS turbulence models decrease below steady value and start to oscillate around experiment value. Average values for k-omega, SST and SAS-SST were almost the same. Results with k-epsilon turbulence model fail to simulate unsteady flow field inside seal land, and remains equal to the steady results discharge coefficient. Flow oscillations periods are a few ranges larger than selected time step which suggests that unsteady flow condition can be simulated with larger time step.

![Figure 10. C_D(time) fluctuation Test A](image)

3.2 Test case B
Calculations for the Test B were made with a similar manner. Discharge coefficient C_D apart from pressure ratio is above results from experiment (Figure 11). Relative error (C_{DCFD}-C_{Dexp}) /C_{Dexp} between steady calculation and experiment results decrease with increase pressure ratio which is tendency reverse to Test A. Error level is on the similar level to the Test A case. The result with SST turbulence model got the highest C_D values, but a globally difference between each C_D results was small.

Pressure distribution along streamline (Figure 12) and velocity contours (Figure 13 and Figure 14) in Test B were compared for π=1.2 and various turbulence models. The pressure reduction between fins was clearly higher for k-epsilon turbulence model despite comparable values of C_D. In this case, the pressure difference is connected with a difference in velocity flow field. Maximum velocity occurs above first fin for SST (and also k-omega) turbulence model and energy dissipation takes places with different manner. Maximum value of velocity for SST turbulence model reaches about 200 m/s while for k-epsilon turbulence model maximum velocity reaches maximum about 150 m/s above second fin. Since the pressure distribution for SST and k-omega are comparable, velocity distributions were similar and figure showing velocity contour for k-omega turbulence model was omitted. The SST turbulence model is a combination of k-epsilon and k-omega turbulence models [8] solving in the region close to the wall equation of k-omega turbulence model. This indicate a strong influence of the modeling the boundary layer on the global nature of the seal flow.
There are more differences in flow distribution inside seal channel. The region with the highest value of velocity is more visible in case of SST turbulence model; also vortices inside honeycomb are clearer. These differences in flow can potentially help to show energy dissipation processes related with flow deceleration and reduction of the leakage [12].

Transient results for Test B allowed showing specific and instant flow field inside seal land. Transient calculation were performed with constraints like in Test A, k-epsilon, k-omega and SST with Co<5 and for SST-SAS Co<1. Chart from Figure 15 illustrated C_D(time). SST, k-omega and SST-SAS turbulence models showed similar tendency to Test A transient C_D fluctuation. CFD C_D values decreased and oscillate around experiment C_D results. The standard k-epsilon model did not predict flow fluctuation. Flow oscillations have a frequency with the clearly lower period than selected time step. Flow fluctuation are higher than in Test A because for a solution with fewer fins the flow is less stable and unsteady flow and pressure fluctuation are less suppressed. Maximum amplitudes were obtained for the models SST and SST-SAS.
Leakage oscillations in Figure 15 and Figure 10 are also connected with velocity and pressure field disruption. Results for Test B with SST-SAS turbulence model allowed showing very complex flow compared to the steady solution. An example of the instantaneous velocity profile (showed in Figure 16) inside seal mid-section and pressure distribution along streamline (in Figure 17) from inlet showed complex fluctuation started above fins and developed in seal chambers.

4. Summary

CFD modelling of flow through the seal can provide results which are not in full compliance with the results of the experiment. Discrepancies amounting to more than few percent could be too high to predict the seal performance. Differences between solutions can also be observed. However, it is possible to predict the global trends. The analysis can be useful in justifying the decision on the use of numerical simulation of flow through the channels.

Analyses indicate that SST and k-omega turbulence models with transient solvers are suitable for modelling compressible flows with numerous flow separation and large velocity gradients. Results with more precise flow projection can be obtained using SST-SAS turbulence model but with higher computational cost.
References

[1] J. Tyacke, R. Jefferson-Loveday and P. G. Tucker, "On the Application of LES to Seal Geometries," Flow, Turbulence and Combustion, vol. 91, no. 4, pp. 827--848, 2013.

[2] I. S. Bambang, C. K. Johan, M. J. d. C. Koen, B. K. Arjen, A. K. Gerrit and F. A. V. Joris, "GT2007-27905 Performance Evaluation of Gas Turbine Labyrinth Seals Using Computational Fluid Dynamics," in ASME Turbo Expo 2007: Power for Land, Sea, and Air, Montreal, 2007.

[3] H. H. Chougule, D. Ramerth and D. Ramachandran, "Low Leakage Designs for Rotor Teeth and Honeycomb Lands in Labyrinth Seals," ASME Turbo Expo 2008: Power for Land, Sea, and Air, no. GT2008-51024, pp. 1613-1620, 9–13 June 2008.

[4] S. Wittig, U. Schelling, S. Kim and K. Jacobsen, "Numerical Predictions and Measurements of Discharge. Coefficients in Labyrinth Seals," 87-GT-188, 1987.

[5] Y. Egorov, F. R. Menter, R. Lechner and D. Cokljat, "The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Complex Flows," Flow, Turbulence and Combustion, vol. 85, no. 1, pp. 139--165, 2010.

[6] H. L. Stocker, D. M. Cox and G. F. Holle, "Aerodynamic performance of conventional and advanced design labyrinth seals with solid-smooth abradable, and honeycomb lands," NASA-CR-135307, 1977.

[7] T. Weinberger, Einfluss geometrischer Labyrinth- und Honigwabenparameter auf das Durchfluss- und Wärmeübergangsverhalten von Labyrintherdichtungen, Stutensee, 2014.

[8] ANSYS® Academic Research, Release 15.0, Help System, CFX Theory Guide, ANSYS, Inc.

[9] W. Wróblewski, S. Dykas, K. Bochon and S. Rulik, "Optimization Of Tip Seal With Honeycomb Land In Lp Counter Rotating Gas Turbine Engine," TASK Quarterly : scientific bulletin of Academic Computer Centre in Gdansk, pp. 189-207, 2010.

[10] J. Gao, K. Zheng and Z. Wang, "Effect of Honeycomb Seals on Loss Characteristics in Shroud Cavities of an Axial Turbine," Chinese Journal of Mechanical Engineering, vol. 26, no. 1, pp. 69-77, 1 02 2013.

[11] I. Mahle, "Improving the interaction between leakage flows and main flow in a low pressure turbine," ASME Turbo Expo 2010: Power for Land, Sea, and Air, vol. 7, pp. 1177-1186, 14–18 June 2010.

[12] J. Denecke, V. Schramm, S. Kim and S. Wittig, "Influence of Rub-Grooves on Labyrinth Seal Leakage," Journal of Turbomachinery, vol. 123, pp. 387-393, April 2003.

Symbols and notes

| Symbol | Description |
|--------|-------------|
| A      | seal clearance area, m² |
| b      | fin tip thickness, m |
| C_D   | discharge coefficient |
| SH    | fin height, m |
| HCD   | honeycomb diameter, m |
| HCH   | honeycomb cell height, m |
| m     | mass flow rate, kg/s |
| p     | pressure, Pa |
| s     | clearance, m |
| t     | fin pitch, m |

R – specific gas constant, kJ/kgK
μ – dynamic viscosity
π – pressure ratio (p_0/p_out)
κ – ratio of specific heats

Index:
nom – nominal
0 – total inlet
out – outlet
t – turbulence