Turbulence Models for Simulation of the Flow in a Rotor-Stator Cavity

Lucie Zemanová1,∗ and Pavel Rudolf1

1Victor Kaplan Dept. of Fluid Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2896/2, 616 69 Brno, Czech Republic

Abstract. Modelling of the flow in the cavities between rotor and stator in turbomachines (e.g. pumps or turbines) is a task of great interest. Correctly evaluated pressure and velocity fields enable calculation of the disk losses and therefore assessment of efficiency. It is also crucial for determination of axial thrust and thus design of the bearings. The study demonstrates abilities of various turbulence models to describe the flow in a narrow gap between rotating and stationary disks. Numerical simulations were performed in order to find out the ability of particular models to capture unstable structures appearing during specific operating conditions as well as to calculate the velocity profiles precisely. Large Eddy Simulation (LES), Scale Adaptive Simulation (SAS), Detached Eddy Simulation (DES), Reynolds stress model (RSM) and SST k − ω model were used. Obtained results were also compared with experimental measurement published by Viazzo et al. [1].

1 Introduction

The flow of a fluid between rotating and stationary disk received attention due to its applicability to many industrial and scientific problems. The investigations started at the beginning of the 20th century by study of Ekman [2] describing wind-driven ocean currents. Ekman was followed by Von Kármán [3], who obtained a similarity solution of Navier-Stokes equations for an infinite disk rotating in quiescent fluid. Bödewadt [4] followed his predecessor with study of flow near a stationary disk in rotating fluid. These findings became the background for comprehensive study of flow and boundary layers established between stationary and rotating disks.

Smith [5] noticed disturbances in form of waves or vortices dependant on Reynolds number. He carried out experiments with rotating disk boundary layer and observed fluctuations in a narrow range of Reynolds numbers below the transition to turbulence. The instability is referred to in literature as Type 1, Type B or crossflow instability. Instability was experimentally visualised and it was found out, that it appears in form of spiral vortices rotating anticlockwise. Faller [6] described the second instability occurring in lower Reynolds numbers, which is known as Type 2 or Type A instability. It also forms as spiral vortices, however it has opposite direction to Type 1 instability.

Instabilities in the vicinity of the rotating disk were first studied experimentally, i.e. [7–10], later on, numerical studies [11–13] occurred. With development of computing technology, computational fluid dynamics (CFD) became a widespread tool for flow simulation. New challenges connected with modelling of turbulence arose. Different approaches exist, nevertheless, the choice is always a compromise between the accuracy and computational resources.

From the accuracy point of view, the best results can be achieved by direct numerical simulation (DNS). The turbulence is explicitly resolved at all relevant length and timescales. From that fact results also a disadvantage, since the computational domain has to be large enough to include the largest naturally-occurring eddies while a computational grid has to be fine enough to fully resolve all dissipation scales. This leads to limited usability for relatively low Reynolds numbers, which are orders of magnitude smaller than in common industrial applications. DNS is not implemented into commercial software due to the requirement of special numerical schemes [14].

For computation of engineering problems, simpler description of the turbulence based on averaging in time is used. Instead of finding the instantaneous flow field, Reynolds-averaged Navier–Stokes (RANS) equations are solved. The set of the equation is a result of Reynolds decomposition, which divides the variables into time-averaged and fluctuating component. The concept of averaging is that it produces additional variables, therefore the system of equations is not possible to solve. This is in literature referred to as the closure problem. By supplementing additional equations (turbulence models), RANS can be solved. Many different types of turbulence models exist. Depending on the number of additional equations, we can distinguish (0-4)-equation models. The two-equation models based on Boussinesq eddy-viscosity hypothesis, such as k − ε or k − ω are the most common in practical applications [15].

Common k − ε model offers good trade-off between the accuracy and computational costs. It is based on the two additional transport equations for turbulent kinetic energy k and turbulent dissipation ε. In its basic form, it
belongs to so-called high-Reynolds models group. As the
name suggests, it is applicable to high turbulent Reynolds
numbers. Turbulent Reynolds number is given by the ra-
tio of eddy viscosity to molecular viscosity, therefore it
is a measure of the level of turbulence. In the areas with
low $Re$, where the viscous effects dominate (e.g. near wall
regions), high-Reynolds models are not appropriate and
other approach needs to be employed. The common prac-
tise it to use wall-functions, where the integration is not
performed up to the wall, but in these areas, it is replaced
by empirically obtained equations. Later, modifications of
$k - \epsilon$ which enables to solve the flow field even in the
near wall regions without the wall-functions were devel-
oped (low-Reynolds version of $k - \epsilon$ model). However,
the wall functions as well as the integration up to the wall
are based on the empirical relationships derived for bound-
ary layers developed during the steady flows over a flat
plane. Therefore, validity in case of the complex flow in-
volving boundary layer separation, pressure gradient, suc-
tion, blowing, roughness etc. is questionable [16].

Better results for solution of the flow field in a bound-
ary layer gives low-Reynolds two-equation $k - \omega$ model. It
adds two transport partial differential equations for $k$ and
$\omega$ (rate of the energy dissipation) to deal with the closure
problem. It is suitable for flows with adverse pressure gra-
dients and when the integration through the viscous sub-
layer is preferred (e.g. for solving boundary-layer transi-
tion) [17]. Modification of the model derived by Menter
[18] SST $k - \omega$ is often implemented in commercial soft-
ware due to the effective combination of high- and low-
Reynolds approach. It couples the $k-\omega$ and $k-\epsilon$ turbulence
model in a way that the $k - \omega$ is used in the region of the
boundary layers and switches to the $k - \epsilon$ in the core flow.
The next step further to describe precisely the turbulence is
Reynolds Stress Model (RSM). Unlike the eddy viscosity
models, it takes into consideration anisotropy of the turbu-
lencc. The individual components of the Reynolds stress
tensor are directly computed, therefore it is able to re-
solve more complex turbulent interactions such as strongly
swirling flows. Wall-function as well as low-Reynolds ap-
proach can be applied in connection with RSM [15].

Different approach to simplification of the Navier-
Stokes equations than Reynolds averaging in time offers
large eddy simulation (LES). Eddies of large scale are
solved directly using Navier-Stokes equations, while the
small scale (sub-grid scale) eddies are modelled. The out-
lined procedure leads to significant reduction of the com-
cputational costs compared to DNS. It is still more demand-
ing than RANS, nevertheless this approach gives results of
much better accuracy. The large eddies, which carry the
most of the turbulent energy and are responsible for the most
of the momentum transfer are captured directly. On
the other hand, sub-grid scale eddies are from their nature
more isotropic and homogeneous, therefore the modelling
is sufficient enough to obtain high accuracy solution [19].
Moreover turbulence models for isotropic turbulence can
be relatively simple. The disadvantage is a need to use
time resolution orders smaller in comparison with RANS.
Similar situation is also for the spatial resolution of the
grid. Wall functions are not possible to use and strict re-
quirements on element quality in boundary layer regions
has to be met [20].

In order to overcome the computational demands of
LES for higher Reynolds numbers, so-called hybrid mod-
els combining LES with RANS were developed. Detached
eddy simulation (DES) utilizes LES for resolving the de-
tached eddies (separated regions) far from boundaries and
RANS model for the near wall flow. It exhibits good re-
results in flows with large separation regions, cavities with
simple geometry or flows connected with acoustic prob-
lems. On the other hand, the weakness is description of
curvature streamlines and strong dependency of the re-
results on the mesh [21]. The regions of separation solved
by LES should be known prior to the simulation and
meshed in an appropriate way, as describes [22]. Slightly
different approach offers scale-adaptive simulation (SAS)
method, which brings dynamic behaviour to the model.
By introducing Von Kármán length scale into the scale-
determining equation of RANS turbulence models, auto-
matic balancing of the contributions of modelled and re-
solved parts of the turbulent stresses is enabled. As a con-
sequence, for unstable flows the model changes smoothly
from LES model through various stages of eddy-resolution
to steady RANS model [23].

For CFD simulations of the flow inside rotating cavi-
ties, different authors use various approaches to modelling
the turbulence. Chew [24] solved the velocity profile of the
flow inside the rotating cavity with high-Reynolds $k - \epsilon$
model, while authors in [25] used the low-Reynolds ap-
proach, Elena and Schiestel [26] involved RSM model. All
these studies were focused on the solution of velocity
profiles and no instabilities were mentioned. By compari-
son with measurements it was found out, that these models
underestimate the boundary layer thickness. Poncet et al.
[27] reported relatively good agreement with experiments
by RSM, while in [28] there is described successful de-
ployment of SST $k - \omega$ to rotating cavity flow problem. Lo
et al. [29] compared measurements with simulations using
LES. Apart from other turbulence models, they were able
to detect even the instabilities emerging on rotor and sta-
tor disks. From its nature, promising results with reduced
computational resources should be achieved using hybrid
LES-RANS approaches.

Taking into consideration the need to capture instabil-
itities, the structure of the flow is highly complex involving
laminar, transitional and turbulent regions. Moreover, the
turbulence is strongly inhomogeneous and anisotropic due
to rotation effects. It leads to a very challenging task for
turbulence models [30]. The aim of this study is to ex-
plre and compare the capabilities of particular models to
capture instabilities in rotor-stator cavities and their capa-
bility to describe the velocity profile precisely. Resolving
the flow in boundary layers is crucial for determination of
friction torque and efficiency, since it directly influences
the dissipation on the shroud and the hub, so-called disk
friction. The findings will be later on applied on the flow
in real shape rotor-stator cavity of a centrifugal pump and
simulations leading to the design optimization.
2 Numerical model

The present study is based on the same geometry of rotating cavity as was analysed by Viazzo et al. [1]. Different approaches to modelling the turbulence were applied on this case, namely LES, SAS, DES, RSM and $k-\omega$. Commercial software ANSYS Fluent 19.1 was used to perform the calculations.

2.1 Geometry and mesh

The analysed geometry with corresponding dimensions is shown in Fig. 1. It consists of two parallel disks of outer radius $b = 140$ mm. The disks are delimited by an inner cylinder (the hub) attached to the rotating disk and by an outer cylinder (the shroud) connected to the stator. The height of the gap is $h = 20$ mm. Due to the symmetry of boundary conditions and the resulting flow, it is possible to reduce the domain to the fraction and save computational resources, as was shown in [1].

Fig. 1. Geometry and dimensions of the domain

The flow can be characterized by the aspect ratio of the cavity $G = 5$, the curvature parameter $R_m = 1.8$ and the rotational Reynolds number $Re = 4 \times 10^5$, defined as:

\[
G = (b - a)/h
\]

\[
R_m = (b + a)/(b - a)
\]

\[
Re = \Omega b^2 / \nu
\]

For the purposes of evaluation, dimensionless axial and radial location is defined:

\[
z^* = z/h
\]

\[
r^* = (r - a)/(b - a)
\]

Fig. 2. Mesh of the domain

2.2 Boundary conditions

No-slip boundary condition was applied to all of the walls of the cavity. The shroud and the upper disk was stationary. The lower disk and the hub were set as rotating walls with constant angular velocity $\Omega = 20.4082$ rad.s$^{-1}$, corresponding to the rotational Reynolds number $Re = 4 \times 10^5$, for which the instabilities occur. Periodic boundary condition was applied to the side walls of the domain.

2.3 Computational details

The fluid inside the rotating cavity was considered incompressible. The default model for water from ANSYS library was used. The calculation was first run as a steady state with $k-\epsilon$ turbulence model. SIMPLE scheme was used for pressure-velocity coupling. First orders of accuracy were set for advection terms in all transport equations. Enhanced wall-treatment (integration up to the wall without wall functions using one-equation model in the near-wall region) was used. After achieving the convergence, pressure, turbulent kinetic energy and turbulent dissipation rate was switched to the second orders of accuracy and momentum to QUICK scheme ($3^{rd}$ order of accuracy). The results were then used as an initial condition for transient analysis. Pressure-velocity coupling algorithm was changed to PISO, which is recommended for unsteady flows. An appropriate time step was chosen with respect to the known nature of instabilities and particular model of turbulence.

2.3.1 Large eddy simulation (LES)

Based on the results previously reported in the literature, e.g. [1], [29], the time step for LES enabling to capture coherent vortical structures was set to $1 \times 10^{-5}$ s. Smagorinsky-Lilly subgrid-scale model was chosen. Second orders of accuracy with bounded central differencing method for momentum was considered.

2.3.2 Scale Adaptive Simulation (SAS)

The time step was initially set to $1 \times 10^{-5}$ s as well as in LES case. It was expected, that coarser time resolution would be sufficient, however the setting was used in order to ensure better convergence and to obtain an initial guess for Courant number.

\[
C = U\Delta t/\Delta x
\]
where \( U \) is the magnitude of velocity, \( \Delta t \) is time step and \( \Delta x \) is the minimum cell size. According to Courant-Friedrichs-Lewy (CFL) condition, Courant number should not exceed 1 for explicit numerical method. Based on that, maximum time step for obtaining correct results is \( 4 \times 10^{-4} \) s. Second orders of accuracy were applied as in the LES case. Scale adaptive simulation was used in combination with the SST turbulence model.

2.3.3 Detached Eddy Simulation (DES)

According to CFL condition, even slightly larger time step can be applied in DES, however the difference is not significant, therefore the same time step as in previous case \( \Delta t = 4 \times 10^{-4} \) s was applied. As a RANS model for resolving the parts of the domain where no separation is detected, SST \( k - \omega \) was chosen. The case was solved with the second order of accuracy. DES used SST \( k - \omega \) RANS model.

2.3.4 Reynolds stress Model (RSM)

For RSM, linear pressure-strain model was set, the second order of accuracy schemes for advection terms were applied. Time step, which ensure \( C < 1 \) is again \( \Delta t = 4 \times 10^{-4} \) s.

2.3.5 SST \( k - \omega \)

As in previous cases, appropriate \( \Delta t = 4 \times 10^{-4} \) s and the second order of accuracy was set.

3 Results and discussion

3.1 Instabilities

Coherent structures emerging in the boundary layers can be detected by Q-criterion (second invariant of velocity gradient tensor). For given rotational Reynolds number, Type II instability is expected, which can be observed as spirals in 15° angle in tangential direction, rotating in opposite direction than the disk.

![Fig. 3. Experimental visualisation of the instabilities [33]](image)

In rotor-stator cavity, Ekman boundary layer is formed on the rotor, while Bödewadt type of the boundary layer appears on the stator. Bödewadt boundary layer is less stable than Ekman and therefore, instabilities occur first on the stator side and are more pronounced. The ability of particular turbulence models to capture the instabilities is demonstrated by figures in which Q-criterion on the stator is shown.

As can be seen in Figs. 4 and 5, capability of LES to describe the instabilities was confirmed. The spiral vortices in the middle part as well as stronger unstructured turbulence near the hub described in experiment [1] is captured.

![Fig. 4. Q-criterion: LES](image)

The other turbulence models failed to detect the coherent structures in rotor-stator cavity, as document Fig. 6–9. In case of SAS, indications of vortices can be seen in near hub region, where the strongest intensity of the turbulence takes place. DES, RSM and SST \( k - \omega \) were not able to model spiral structures entirely. Even when the time step was lowered to the \( 1 \times 10^{-5} \) s as in LES case, no satisfactory results were obtained.

![Fig. 5. Radial section, Q-criterion: LES](image)

![Fig. 6. Q-criterion: SAS](image)
3.2 Velocity profiles

The flow inside the rotor-stator cavity can be characterized as a core with two thin boundary layers. The thickness of Ekman boundary layer created on the rotor is significantly lower than Bödewadt boundary layer on stator. It corresponds to the results published in [1] and [32]. The fluid is driven centrifugally outwards along the rotor, turns into axial direction and due to conservation of mass, it is forced to flow radially inwards along the stator.

The velocity of the core fluid can be divided into axial, radial and tangential directions. The axial velocity component approaches zero, therefore it is not visualised. Fig. 10 shows instantaneous radial (left) and tangential (right) velocity component.

The velocity of the core fluid can be divided into axial, radial and tangential directions. The axial velocity component approaches zero, therefore it is not visualised. Fig. 10 shows instantaneous radial (left) and tangential (right) velocity component.

The profiles were evaluated in the middle of the computational domain (mean radius of the disk cavity). Both, the axial coordinate and the velocity are dimensionless. The components of velocity were made non-dimensional according to following relationships:

\[ v_r^* = \frac{v_r}{\Omega r} \]  
\[ v_t^* = \frac{v_t}{\Omega r} \]  

Reasonable agreement between numerical predictions and experimental data was reached. All mentioned models were able to calculate velocity in the core region precisely. The closer the walls, the more the discrepancies are visible. Unfortunately, in these regions, only few measured data were obtained and the measurement was averaged over time, thus unsteady phenomena were suppressed. As mentioned earlier, LES was the only model, which was able to capture instabilities. It propagates into the velocity profile, as can be observed especially on the rotor side (\( z^* = 0 \)). This is the reason for the difference in velocity profile between experimental data and LES model towards the rotor.
In comparison with measured data, numerical models slightly underestimate the thickness of Ekman boundary layer and overestimate the thickness of Bödewadt layer. Velocity profiles in near wall region can be seen in detail in following figures. Fig. 11 shows radial and tangential velocity close to stator (Bödewadt boundary layer). In Fig. 12, similar results for rotor (Ekman boundary layer) are presented. Correct resolution of velocity profile in inner boundary layer is crucial in determination of the shear stress and consequently friction losses.

In Figs. 11 and 12, only numerical models are compared, since the closest measured point has $y^+$ out of the viscous sublayer – the main area of interest. It can be seen, that the velocity profiles computed by SAS, DES, RSM and SST $k-\omega$ models do not differ too much between each other in general. They are able to describe the profile with good accuracy in case of tangential velocity (see Figs. 11 and 12 right).

Even though the instabilities were not captured, the slope of the tangential velocity in viscous sublayer ($y^+ < 5$) is very similar to the LES model, which included the unstable nature of the boundary layer.

In buffer layer region ($5 < y^+ < 60$), the differences are most significant. Out of the buffer layer, with increasing $y^+$ the velocity profiles obtained by different models approach each other. Larger discrepancies are visible in case of radial velocity (Figs. 11 and 12 left). The shape of the velocity profile has the opposite direction on the stator and the rotor side, since the fluid flows radially inwards along the stator, whilst outwards orientation can be observed on the rotor. The slope of the curve differs in viscous sublayer as well as in buffer layer. SST $k-\omega$, DES and SAS produce similar result on the stator as well as on rotor. RSM slightly differs on the more unstable stator side. The velocity profiles computed by these models are shifted from LES profile. On the stator, they predict larger gradient of radial velocity in viscous sublayer, on the other hand, on the rotor side the gradient of radial velocity is smaller in comparison with LES. The radial velocity magnitude from SAS, DES, RSM and SST $k-\omega$ calculation is smaller in viscous sublayer and buffer layer of the stator compared to LES. On the contrary, on the rotor side, the predicted radial velocity by SAS, DES, RSM and SST $k-\omega$ is larger than LES modelled.

Fig. 11. Velocity profiles in stator boundary layer

Fig. 12. Velocity profiles in rotor boundary layer
4 Conclusions

In this paper, capabilities of LES, SAS, DES, RSM and SST $k - \omega$ turbulence models to describe the flow inside rotor-stator cavity were demonstrated. The ability to capture the vortices emerging in unstable regime was assessed. For further practical application meaning calculation of the disk friction losses and efficiency, the numerical model should be able to solve velocity profiles precisely, especially in near wall regions. Therefore, the second part of the study dealt with comparison of velocity profiles obtained by particular turbulence models. It was found out, that less computationally demanding models, i.e. SAS, DES, RSM and SST $k - \omega$, are not able to detect the instabilities at all. The vortical structure obtained by LES agrees with experimentally reported data in [1]. The model is able to capture the spiral vortices on rotor and stator as well as stronger turbulence with unstructured swirling near hub.

The radial and tangential velocity profiles computed by SAS, DES, RSM and SST $k - \omega$ are very similar. They agree with LES in the core of the fluid, however, in the near wall regions larger differences are obvious. The largest discrepancies were observed in radial component of velocity. Unfortunately, these regions are of the great interest for determination of friction losses. Also, these regions are important for the onset and further spreading of the instability.

In conclusion, the comparison between the models showed, that LES in inevitable level of modelling for such flows. Reduction of computational demands by using different turbulence model leads to significant reduction of accuracy and to the loss of the important information about instabilities.

Turbulence Models for Simulation of the Flow in a Rotor-Stator Cavity was supported by project FSI-S-17-4615.

References


