Numerical analysis of turbine blade cooling ducts

Citation for published version (APA):
Numerical Analysis of Turbine Blade Cooling Ducts

M.J. Noot and R.M.M. Mattheij
Eindhoven University of Technology, Dept. of Mathematics and Computing Science,
P.O. Box 513, 5600 MB Eindhoven, The Netherlands

March 2, 1998

Abstract

The cooling of turbine blades in turbines is enhanced by providing the cooling ducts with ribs, so-called turbulators. It is investigated how these ribs influence the heat transfer of the cooling air on the blades. A model is given to study this problem such that it lends itself to a numerical approach. A detailed discussion is given of the problem involved. It is shown how the ideas are implemented in a numerical code. The results of the simulations are assessed showing a practical way to test the quality of these cooling ducts.

1 Introduction

Gas turbines play an important role in aviation and industrial applications. There is a growing tendency to use higher temperatures at the inlet of the turbine to improve the efficiency of the gas turbine engine. Consequently the heat load on the turbine components increases, especially in the high pressure turbine section. This heat load is caused by the exposure to an enormous heat flux of the burnt gas from the combustion chamber. As a result the lifetime expectancy of a blade can be reduced significantly. In order to comply with modern safety standards the blades in gas turbines need to be cooled.

Usually the main objective in turbine blade cooling is to achieve maximum heat transfer while minimizing the coolant flow rate. Coolant air is routed through turbulated passages within the blade; thus it removes heat from the blade by convection. The walls of the passage have ribbed surfaces to improve the cooling efficiency of the air flow. These ribs are called
Figure 1: Turbine blade with cooling holes

turbulence promoters or turbulators (see figure 1). Here an interesting design or engineering problem occurs. How does the shape and distribution of the turbulators influence the overall heat transfer? It is known that in a turbulent flow the heat transfer is considerably higher than in a laminar flow and for turbulated holes also the area of contact surface between metal and cooling air is increased. Another factor is the amount of cooling air that is needed to achieve a certain level of heat transfer. It is clear that for turbulated holes less cooling air is needed than for smooth-walled holes.

In this paper we shall consider, for low frequency gas turbines, this cooling process in some detail and describe a model to simulate the heat exchange, resulting in actually predicting Nusselt numbers in various situations. The paper is built up as follows. In section 2 we derive the model describing this cooling process. A central equation is of course Navier-Stokes. The numerical aspects of solving them are considered in section 3. In section 4 we perform and analyse simulations for unsteady laminar flow, paying special attention to the boundary conditions. Finally, in section 5 we give the results for turbulent flow enabling us to quantify the actual heat transfer.

2 Modelling

We first consider the physical model, indicating the conditions under which our approach makes sense. Then we derive the mathematical model, i.e. the governing equations which will be used for the simulation. Then this is made dimensionless in order to assess the various contributing terms, which
is then used to derive the proper equations.

2.1 Physical modelling of cooling

Rather than modelling the blade with several cooling ducts, we will investigate the local heat transfer in the neighbourhood of just one cooling duct. Even more so, we shall restrict ourselves to only a section of the duct. We will assume the fluid to be Newtonian with the properties of a perfect gas. In order to further describe the system we have a closer look at the incompressibility and the rotation.

- Incompressibility
  The cooling air is a gas with small viscosity, forced through a narrow passage under high pressure. Let \( V \) be the characteristic velocity of the flow and let \( c \) be the speed of sound, then the Mach number is defined by
  \[
  Ma := \frac{V}{c}.
  \]
  For laminar flows \( Ma < 0.3 \) is acceptable for considering a fluid as approximately incompressible (see [4], p.581). For turbulent flows this assumption is valid for even higher Mach numbers (see [10], p.171). Our type of flow has a typical velocity of 100 m/s and since the thermal environment of a gas turbine is about 1200 K with a cooling air inlet temperature of 600 K, the speed of sound is approximately 500 m/s. Using the air temperature we thus find a Mach number of about 0.2.

- Effects due to system rotation
  The type of turbine we will be considering is, more specifically, a stationary gas turbine with a rotational frequency of 50 Hz. So we have a constant angular velocity \( \omega = \omega \mathbf{E}_1 \). From the inertial system \( \{ \mathbf{E}_1, \mathbf{E}_2, \mathbf{E}_3 \} \) a relative basis \( \{ \mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3 \} \) is obtained which is rotating along with the pipe (see figure 2). As a result additional forces will have to be introduced in the momentum equations which are now relative to the rotating frame of reference. These forces are called the centrifugal force and the Coriolis force. Due to the presence of these forces, secondary flow is induced. Thus the rotation of the pipe makes the problem essentially three dimensional.
2.2 Mathematical modelling of the cooling

Since the density $\rho$ of the fluid is taken constant, the variables that remain in our problem are the velocity $\mathbf{u}$, the pressure $p$ and the temperature $T$ of the fluid. To solve the flow problem we need five equations involving these variables. In this section we will treat the flow problem and the temperature problem separately, although the viscosity $\eta$ is the quantity coupling both problems, thus making the flow problem implicitly depending on the temperature. The general equations for an incompressible Newtonian fluid will be used (see [1], p.174). Note that we are dealing with a real fluid. However low the viscosity may be, viscous effects do play a role in this problem for the very reason that we are interested in boundary-layer effects for both temperature and velocity. Only in this way we can compute heat transfer coefficients. Together with the energy equation they will yield a complete system of equations for $\mathbf{u}$, $p$ and $T$.

The incompressibility assumption gives us the continuity equation

$$\text{div} \, \mathbf{u} = 0,$$

expressing conservation of mass. Besides conservation of mass we also have conservation of momentum

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u}, \text{grad} \, \mathbf{u}) = \mathbf{f} + \text{div} \, S,$$

where the stress tensor $S$ represents dilatation and shear in the fluid. The term $\mathbf{f}$ incorporates all external body forces on the medium, like Coriolis force, centrifugal force and gravitational force. The first two of these forces
are the result of introducing the relative frame of reference. The Coriolis force and centrifugal force contributions are given by

$$\mathbf{f} = 2\rho(\mathbf{\omega} \times \mathbf{u}) - \rho(\mathbf{\omega} \times (\mathbf{\omega} \times \mathbf{x})), \quad (2.4)$$

(see [1], p.139). We need a constitutive equation to relate the stress tensor $S$ to the variables $u$ and $p$. For a Newtonian fluid this relation is given by

$$S = -pI + 2\eta \mathbf{D}, \quad (2.5)$$

where the tensor $D$ denotes the rate of strain, i.e. $D = \frac{1}{2} (\text{grad} \ u + \text{grad} \ u^T)$. Note that $\eta$ is inhomogeneous and depending on $T$. Since we are interested in the temperature field, we have to include the energy equation in our set. For a viscous, incompressible fluid the energy equation is

$$\rho c_p \frac{DT}{Dt} = \lambda \text{div grad} \ T + \frac{Dp}{Dt} + 2\eta \text{tr}(D^T D), \quad (2.6)$$

We remark that the equations for $u$ and $p$ are nonlinear, whereas for given $u$ and $p$ and constant $\lambda$, $\eta$ and $c_p$ the energy equation is a linear equation for $T$. The nonlinearity is caused by the presence of the convective term in (2.3). Using equations (2.2) and (2.3)-(2.5) we find the unsteady Navier-Stokes equations for an incompressible flow together with the energy equation (2.6).

$$\text{div} \ u = 0, \quad (2.7a)$$

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u}, \text{grad} \ u) = 2\rho(\mathbf{\omega} \times \mathbf{u}) - \rho(\mathbf{\omega} \times (\mathbf{\omega} \times \mathbf{x})) +$$

$$- \text{grad} \ p + \text{div}(2\eta \mathbf{D}), \quad (2.7b)$$

$$\rho c_p \frac{DT}{Dt} = \lambda \text{div grad} \ T + \frac{Dp}{Dt} + 2\eta \text{tr}(D^T D), \quad (2.7c)$$

where we have used (2.4) for $\mathbf{f}$.

The computational domain for the flow problem and that for the temperature problem will not be the same. This is because we are not only interested in the temperature distribution in the cooling duct, but in parts of the metal surrounding it as well; the latter is needed to be able to compute heat transfer from the blade to the cooling duct.

### 2.3 Dimensionless numbers

In order to investigate the importance of rotational and viscous effects, the set of equations will be made dimensionless first. Since $\rho$ is constant the
centrifugal force is conservative and can be absorbed into the pressure gradient term (see [1], p.176). To this end we introduce the modified pressure $p_m$ which is defined by

$$p_m := p - \frac{1}{2} \rho \| \omega \times \mathbf{x} \|^2. \quad (2.8)$$

The introduction of $p_m$ is mainly done for notational purposes. In fact this notation only pays off when the boundary conditions involve velocity components exclusively. The momentum equations are now written as

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u}, \text{grad } \mathbf{u}) = 2\rho (\omega \times \mathbf{u}) - \text{grad } p_m + \text{div}(2\eta D). \quad (2.9)$$

We now introduce dimensionless variables by providing them with a prime.

- coordinate: $x_i = L x'_i \quad (i = 1, 2, 3)$
- velocity: $u_i = V u'_i \quad (i = 1, 2, 3)$
- time: $t = LV^{-1} t'$
- angular velocity: $\omega_i = \omega \omega'_i \quad (i = 1, 2, 3)$
- pressure: $\Delta p = \rho V^2 \Delta p'$
- temperature: $T = T_0 + T' \Delta T$
- dynamic viscosity: $\eta = \eta_0 \eta'$
- stress: $S_{ij} = \eta_0 L^{-1} V S'_{ij} \quad (i, j = 1, 2, 3)$

The quantities $L$, $V$, $\omega$, $T_0$ and $\eta_0$ are characteristic for the problem. Their values are chosen such that the dimensionless variables are $\approx 1$. The temperature $T_0$ is a reference value and can be taken equal to the inlet temperature of the cooling air. For $\Delta T$ one could take, for example, the difference in temperature between the blade and the cooling air ($T_0$). As a characteristic length scale we choose $L = D_h$, where $D_h$ is called the hydraulic diameter and is based on the narrowest cross section of the cooling duct. In our specific problem $L$ is of the order 1 mm. Let $\Phi_v$ denote the volumetric flow rate defined by $\Phi_v := \int (\mathbf{u}, \mathbf{n}) \, d\sigma$, where the surface integral is computed over a cross section of the cooling duct. For the velocity we then take

$$V := \frac{4 \Phi_v}{\pi D_h T}. \quad (2.10)$$

Because we are dealing with a ribbed surface, this definition of $V$ may not be indicative for the complexity of the flow anymore. In such a case the small scale flow pattern may fully determine this. In practice $V$ is about
75 m s\(^{-1}\). For the rotational speed \(\omega\) we find 300 rad s\(^{-1}\) as a typical value and \(10^{-5}\) kg m\(^{-1}\) s\(^{-1}\) for the viscosity at \(T = T_0\).

The relevant dimensionless numbers are given by

\[
Re = \frac{\rho_0 VL}{\eta_0} \quad \text{Reynolds number}
\]

\[
Ro = \frac{\omega L}{V} \quad \text{Rossby number}
\]

\[
Pr = \frac{\eta_0 c_p}{\lambda} \quad \text{Prandtl number}
\]

\[
Ec = \frac{V^2}{c_p \Delta T} \quad \text{Eckert number}
\]

The Reynolds number can tell us how strong convection is with respect to viscous effects. The Rossby number indicates the importance of rotational effects compared to convective effects. The Prandtl number gives an indication for the transport of heat with respect to the transport of momentum. Note that Pr only depends on the properties of the fluid. It is known that for gaseous fluids Pr hardly depends on the temperature. In dimensionless form the incompressible Navier-Stokes equations are

\[
\text{div } \mathbf{u} = 0, \quad (2.11a)
\]

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}, \text{grad } \mathbf{u}) = 2Ro(\mathbf{\omega} \times \mathbf{u}) - \text{grad } p_m + \frac{1}{Re} \text{div}(2\eta D), \quad (2.11b)
\]

\[
\frac{DT}{Dt} = \frac{1}{Re Pr} \text{div } \text{grad } T + \frac{Ec}{Re} \frac{Dp}{Dt} + \frac{Ec}{Re} 2\eta \text{tr}(D^T D), \quad (2.11c)
\]

where the prime has been omitted again for simplicity. The factor Ec/Re indicates the effect of viscous dissipation in relation to convective heat transfer. Note that the modified pressure \(p_m\) is scaled as follows

\[
p'_m = p' - \frac{1}{2}Ro^2 \| \mathbf{\omega} \times x' \|^2.
\]

The temperature is an essential variable in our problem and is determined once the velocity field is known. Due to strong convection, temperature boundary-layers will be formed along the wall. There is a strong difference between the temperature of the fluid some distance away from the wall and that of the blade. The cooling air entering the duct has a temperature of about 600 K, while the metal of the blade is at 1200 K - 1600 K near the inlet of the duct. To obtain a global quantity concerning heat transfer we will look at a further dimensionless number, namely the Nusselt number \(Nu\). Generally the Nusselt number is a function of the
geometry and all other dimensionless numbers involved, which in our case yields \( \text{Nu} = \text{Nu}(\text{Re}, \text{Ro}, \text{Pr}, \text{Ec}) \). The dependence of the heat transfer on the Reynolds number \( \text{Re} \) will be the strongest, since the Prandtl number is fairly constant and the rotational speed of the blade is assumed constant. It will even turn out that effects due to blade rotation \( (\text{Ro}) \) and the effects of viscous dissipation \( (\text{Ec}/\text{Re}) \) can be neglected, as will be discussed in the next section. Therefore for a certain turbulator configuration, a plot of \( \text{Nu} \) against \( \text{Re} \) contains all the necessary information about the heat transfer for that particular cooling duct.

The overall Nusselt number is computed by averaging the local heat transfer rate. The coefficient of heat transfer \( \alpha \) at a given cross section in the duct is implicitly defined by

\[
q_w := \alpha(T_w - T_b),
\]

where \( q_w \) is the heat flux from the wall to the fluid, \( T_w \) is the wall temperature and \( T_b \) is the bulk temperature or mass-averaged temperature. The latter is defined by

\[
T_b(z) := \frac{1}{\Phi_v} \int_{S_c} (u, n)T \, d\sigma,
\]

with \( z \) the coordinate along the axis of the duct and where the integral is taken over the cross sectional area \( S_c \) of the duct at cross section \( z \). Now there is a local value of \( \alpha(z) \) associated with each cross section \( z \). To obtain the overall Nusselt number we need to compute an average coefficient of heat transfer \( \bar{\alpha} \). Assuming a constant heat flux \( q_w \) on the wall, \( \bar{\alpha} \) can be computed by \( \bar{\alpha} = q_w/\Delta T_{ref} \), where \( \Delta T_{ref} \) is defined by

\[
\Delta T_{ref} = \frac{1}{\text{area}(A)} \int_A (T_w(z) - T_b(z)) \, d\sigma.
\]

In (2.14) \( A \) denotes the surface of the wall of the cooling duct. This results in an overall Nusselt number \( \text{Nu} \) expressed by

\[
\text{Nu} = \frac{\bar{\alpha}D_h}{\lambda}.
\]

The Nusselt number is, like the Reynolds number, based on the hydraulic diameter.
2.4 Resulting set of equations

For the dimensionless numbers we typically have the following values: \( \text{Ro} = 0.01, \text{Re} = 20000 \) and \( \text{Pr} = 0.7 \). It is clear from (2.11b) that rotational effects are negligible relative to convective effects. Therefore we neglect the Coriolis force and the centrifugal force in (2.11b). Furthermore the effect of viscous dissipation can be neglected since \( \text{Ec} = 0.01 \). One may question why \( \text{Re} \) is not neglected for the same reason as \( \text{Ro} \) is neglected. As explained in section 2.2, we need to take viscous effects into account, however small they are. Upon neglecting the rotational effects we can restrict the investigation to the case of a non-rotating pipe, which reduces the problem from 3D to an axi-symmetric 2D problem. Another simplification is to assume the viscosity to be constant, since we will only be considering a small section of the cooling duct rather than the whole passage. The latter is practically impossible since one would need an enormous amount of computing power to carry out the simulation. As a result of taking \( \eta \) constant the energy equation is decoupled from the flow equations. Now the problem can be described in two dimensions in cylinder coordinates and the set of equations becomes

\[
\begin{align*}
\text{div}\ u &= 0, \quad (2.16a) \\
\rho \frac{\partial u}{\partial t} + \rho(u, \text{grad } u) &= -\text{grad } p + \eta \text{div}(2D), \quad (2.16b) \\
\frac{\partial T}{\partial t} + (u, \text{grad } T) &= \frac{\lambda}{\rho c_p} \text{div grad } T. \quad (2.16c)
\end{align*}
\]

in full dimensions. The actual computational domain is shown in figure 3. The flow problem is computed on domain \( \Omega_1 \) and the temperature problem
on $\Omega_1 \cup \Omega_2$. Boundary conditions will be discussed in Chapter 6. Because of the complex passage shape and the high pressure difference the flow is strongly three-dimensional and turbulent. A turbulent flow can generally not be considered axisymmetric. However, for certain turbulence models axisymmetry can be assumed.

3 Numerical Aspects

3.1 Time Integration

In this section we consider the time-dependent equations. In an operator form the Navier-Stokes equations can be written as follows:

$$M \dot{u} + A u + C(u)u + B'p = F,$$

$$Bu = 0.$$

The discretization with respect to time is done by the $\theta$-method. Common values for $\theta$ are $\theta = 1$ (Euler backward scheme) and $\theta = 0.5$ (Crank-Nicolson scheme). The latter yields second-order accurate time integration opposed to the fully implicit integration scheme. However, the advantage of the Euler backward scheme is that it is unconditionally stable. The discretization scheme for the $\theta$-method is given by

$$M \frac{u^{n+1} - u^n}{\Delta t} + A(\theta u^{n+1} + (1 - \theta)u^n) + \theta C(u^{n+1})u^{n+1} +$$

$$(1 - \theta)C(u^n)u^n + B'(\theta p^{n+1} + (1 - \theta)p^n) = \theta F^{n+1} + (1 - \theta)F^n,$$

$$Bu^{n+1} = 0,$$

where $\Delta t$ is the time step size. This system of equations can be reformulated as follows

$$\left(\frac{1}{\Delta t}M + \theta(A + C(u^{n+1}))\right)u^{n+1} + \theta B'p^{n+1} =$$

$$\left(\frac{1}{\Delta t}M - (1 - \theta)(A + C(u^n))\right)u^n - (1 - \theta)B'p^n + \theta F^{n+1} + (1 - \theta)F^n,$$

$$Bu^{n+1} = 0.$$
Upon linearization according to Newton’s method we obtain

\[
\begin{align*}
\left( \frac{1}{\Delta t}M + \theta A \right) u^{n+1} &+ \theta C(u^{n+1}) u^n + \theta C(u^n) u^{n+1} + \theta B' p^{n+1} = \theta C(u^n) u^n + \\
\left( \frac{1}{\Delta t}M - (1 - \theta)(A + C(u^n)) \right) u^n &- (1 - \theta) B' p^n + \theta F^{n+1} + (1 - \theta) F^n,
\end{align*}
\]

(3.4a)\hspace{1cm} (3.4b)

\[B u^{n+1} = 0.\]

### 3.2 Efficient Solution of the Discretized System

The core of the problem is how to solve the resulting linear problem efficiently. In this section we therefore consider a numerical approach of solving system (3.4). Even in two dimensions direct solvers for medium- to large-scale problems pose severe restrictions on the number of elements that can be used in the discretization and thus on CPU-time, but even more on memory. Since finite-element matrices are inherently sparse, iterative solvers are favourable for both speed and memory usage. In the following we will focus on iterative solvers and BICGSTAB in particular. This technique is discussed in [7] and is very easy to implement. Note that we use the integrated method (cf. [6]) to solve the system of equations (see Eq. (3.4)). A preconditioner is necessary to speed up the solver. We will use incomplete LU factorization (ILU) as a preconditioner.

Since iterative methods exploit finite-element matrices being sparse, a proper matrix storage scheme is very important. Assuming a regular mesh, an internal node of the mesh will belong to about 6 elements. For the purpose of illustration we consider the \((P_2, P_1)\) Taylor-Hood element (see figure 4). This implies that a variable, representing a velocity component, at that node will have relations with 7 pressure nodes and 19 velocity nodes resulting in \(7 + 2 \times 19 = 45\) non zeros in the corresponding matrix row. For an irregular mesh this number can become as high as 66 non zeros in a matrix row. Note that this number is independent of the number of elements in the mesh.

The matrix storage scheme used is the so-called \textit{compressed row storage} scheme (CRS). Since we do not want to use a direct solver we do not store the profile of the matrix in which we allow fill-in, but rather the non zeros and their position in the matrix. For more details see [6].
The laminar flow computations have been carried out for Reynolds numbers up to 550. With a little over 5000 unknowns, the solutions are relatively cheap to obtain. This is only possible because the ejected vortices do not affect the core flow region, but remain in the cavities between the turbulators. From this we can deduce that convective inertial effects do not play an important role in the core region. Therefore the flow complexity is not really determined by the global Reynolds number, but by the unsteady flow behaviour near the turbulators where the velocities are much lower than in the middle of the cooling duct.

We have chosen to use periodic boundary conditions, thus avoiding transient effects near the inlet of the cooling duct. This makes it possible to perform the computations on a small segment of the duct. The periodicity is in the axial coordinate of the relative frame of reference (see section 2.1). Besides this imposed behaviour we observe time cycles displaying regular flow patterns. This means that there is a periodicity in time as well. We use equation (2.16b) to describe the laminar flow. In dimensionless form this yields

\[
\frac{\partial u}{\partial t} + (u, \nabla u) = -\nabla p + \frac{1}{Re} \text{div}(2D). \tag{4.1}
\]

For all laminar flow simulations we assume the viscosity to be constant (see section 2.4), thus decoupling the heat equation from the Navier-Stokes equations. Also for the heat equation we use periodic boundary conditions.

The computed Nusselt numbers are compared with the Nusselt number for
a smooth duct as found in literature.

For the numerical simulations we use the Finite Element Method (FEM), incorporating the integrated method (see section 3.2).

4.2 Periodic boundary conditions

In case the geometry of a turbulated cooling passage is without any defects, it is reasonable to assume that sufficiently far from the inlet the flow becomes periodically developed. First of all it is necessary that the cooling passage itself is geometrically periodic. Therefore we use a cosine function to describe the shape of the duct. Moreover we use periodic boundary conditions. If we divide the cooling passage into segments like in figure 5, we expect periodicity in space to occur over a small number of pipe segments. This approach enables us to perform numerical simulations with a relatively small number of finite elements. First we assume periodicity to occur already over two segments. Then we try to verify this assumption by increasing the number of segments. We choose the inlet and outlet boundaries at the position where the diameter is the smallest. We remark that this choice is arbitrary, since we enforce periodicity. In figure 6 the computational domain $\Omega$ is shown.

![Figure 5: Pipe consisting of four segments](image)

![Figure 6: Computational domain](image)
The geometry of the wall \( \Gamma \) is described by the following function

\[
r = f(z) = R_m + \frac{H}{2} \left(1 - \cos\left(\frac{2\pi z}{\ell}\right)\right).
\]

Typical dimensions for such a turbulated cooling passage can be

\[
\begin{align*}
R_m &= 1.25 \text{ mm} \quad \textit{minimal radius} \\
H &= 0.75 \text{ mm} \quad \textit{turbulator height} \\
\ell &= 3.15 \text{ mm} \quad \textit{turbulator pitch}
\end{align*}
\]

For the laminar flow simulations we do not use the definition of the Reynolds number as described in section 2.3. It turns out that in these simulations the streamlines for \( 0 \leq r \leq R_m \) are horizontal. Even when the laminar flow becomes unsteady, the core region of the cooling duct is hardly affected. The complexity of the flow is therefore characterized by the flow behaviour in the cavities between the turbulators. As a consequence it makes sense for this type of flow to define the characteristic velocity as the velocity in the axial direction at \( r = R_m \) and the characteristic length scale to be equal to the turbulator height \( H \). This Reynolds number will be denoted as \( \text{Re}_H \).

Under these conditions the pressure \( p \) can be expressed as

\[
p = -\beta z + p_r + p_0,
\]

where \( \beta \) is the pressure drop per unit length in the cooling duct, \( p_0 \) a reference pressure and \( p_r \) is called the \textit{reduced pressure}. The quantity \( p_r \) is a periodic function in \( z \) (see [3] and [8]). For notational purposes we introduce a stress tensor \( S^* \) by

\[
S^* := -p_r I + 2\eta D.
\]

Substitution of (4.3) in the momentum equation (2.16b) yields

\[
\rho \frac{\partial u}{\partial t} + \rho (u, \text{grad } u) = \text{div } S
= \text{div } S^* + f
= -\text{grad } p_r + \text{div} (2\eta D) + f,
\]

where the additional body force is given by \( f = \beta e_z \). Now \( \beta \) is the driving force for the flow. The variables \( u \) and \( p_r \) are all periodic in \( z \). We introduce the \textit{periodicity index} \( n \), which is an integer and is yet to be determined such
that

\[ u(r, z, t) = u(r, z + n \ell, t) \]  \hspace{1cm} (4.6a)
\[ p_r(r, z, t) = p_r(r, z + n \ell, t) \]  \hspace{1cm} (4.6b)

holds in the whole domain. Since we expect the periodic behaviour already to occur over a small section of the duct, we start by considering a duct consisting of two segments. If this would already yield \( n = 1 \), we are satisfied. Otherwise we will have to increase the number of segments in the computational domain and investigate the flow patterns in ducts with more segments and determine again their periodicity index. The boundary conditions for \( u \) and \( p_r \) are given in figure 7. Furthermore we need to fix the pressure \( p_r \) at one point in the computational domain to make it uniquely determined. This degree of freedom is actually expressed by the presence of the term \( p_0 \) in (4.3).

For the temperature \( T \) we have to solve a convection-diffusion problem with a prescribed temperature at the inlet and a constant heat flux \(-\lambda \frac{\partial T}{\partial n} = h (h > 0)\) on the wall \( \Gamma \) of the cooling duct. The assumption of a constant thermal load on the wall is done for reasons of simplicity. However, we need quite a lot of segments to obtain the corresponding temperature distribution for the whole cooling duct. Therefore, as for the flow problem, we assume that the temperature field becomes periodically developed sufficiently far from the inlet. Under the thermal conditions of a uniform heat flux we can then express the temperature \( T \) as follows:

\[ T = \gamma z + T^* + T_0, \quad \gamma > 0, \]  \hspace{1cm} (4.7)

where \( \gamma \) is a constant and represents the temperature rise per unit length in the duct and where \( T_0 \) is a reference temperature. This rise in temperature...
is due to the cooling air being heated up by the blade as it flows through the passage. Now the quantity $T^*$ is a periodic function of $z$ such that

$$T^*(r, z, t) = T^*(r, z + n\ell, t)$$  \hspace{1cm} (4.8)$$

holds for the whole domain $\Omega$. The boundary conditions for $T^*$ on the small segment are given in figure 8. Note that the use of a Dirichlet boundary condition for $T$ is not possible in this setup. Substitution of (4.7) in the heat equation (2.16c) results in

$$\rho c_p \left( \frac{\partial T^*}{\partial t} + (\mathbf{u}, \text{grad } T^*) \right) = \text{div}(\lambda \text{grad } T^*) + Q, \hspace{1cm} (4.9)$$

where the additional source term $Q$ is given by $Q = -\rho c_p \gamma (\mathbf{u}, \mathbf{e}_z)$. The redefinition of the temperature gives rise to the appearance of a source term in the equation. The initial condition for (4.9) obviously has to be periodic.

It is clear that the solution is determined up to a constant. Therefore $T^*$ needs to be fixed at one point in the domain. This degree of freedom is expressed by $T_0$ in (4.7). After solving $T^*$ we find $T$ by using (4.7), where the constant $T_0$ is determined by the distance of the segment to the actual inlet of the cooling duct. Because $T^*$ is determined up to a constant, we also have an additional constraint, similar to the compatibility condition for the flow problem. In this case such a condition expresses conservation of energy and is expressed by

$$\int_{\Gamma} -\lambda \frac{\partial T^*}{\partial n} \, d\sigma = \int_{\Omega} Q \, d\tau. \hspace{1cm} (4.10)$$

This yields

$$\int_{\Gamma} (h + \lambda \gamma (\mathbf{e}_z, \mathbf{n})) \, d\sigma = \int_{\Omega} -\rho c_p \gamma (\mathbf{u}, \mathbf{e}_z) \, d\tau. \hspace{1cm} (4.11)$$

16
Note that the second term in the left-hand side of equation (4.11) has a total contribution of zero to the integral, because of the geometric symmetry of the wall \( \Gamma \). As a result we find

\[
h / \gamma = - \frac{\rho c_p}{\mathcal{J}} \int_{\Omega} (u, e_z) \, d\tau.
\] (4.12)

For a given heat flux \( h \), the parameter \( \gamma \) will have to satisfy the above constraint.

### 4.3 Flow computations

To investigate the unsteady laminar flow at a fairly high Reynolds number, flow simulations have been carried out using FEM and imposing the periodic boundary conditions as discussed in the previous section. To capture the flow phenomena it is necessary to refine the grid in the boundary layer and around the turbulators where bifurcations will occur. The local refinement should resolve the large gradients in this area (see figure 9).

![Finite element mesh](image)

**Figure 9:** Finite element mesh (1156 elements, 2421 nodes)

Since we do not know beforehand at which Reynolds number the desired flow behaviour occurs, we use a continuation technique in \( \text{Re}_H \) by changing the parameter \( \beta \). We start with a steady-state initial condition \( (u = 0) \) with \( \beta = 0 \) and solve the time-dependent Navier-Stokes equations and change \( \beta \) in time according to

\[
\beta = \begin{cases} 
\frac{1}{2}(\beta_{\text{new}} + \beta_{\text{old}}) + \frac{1}{2}(\beta_{\text{old}} - \beta_{\text{new}}) \cos \left( \frac{\pi t}{t_0} \right) & (t < t_0), \\
\beta_{\text{new}} & (t \geq t_0).
\end{cases}
\] (4.13)

All parameters in (4.5) are taken equal to 1 including the dimensions of the domain. For given \( \beta \) we can compute the velocity field and the corresponding Reynolds number according to the definition of \( \text{Re}_H \) in 4.2.

It is well known that a fully implicit time integration method causes too much damping thus hindering attempts to achieve the oscillatory behaviour.
Therefore we begin our computations with an Euler backward scheme and at $Re_H = 300$ we modify the way we vary the continuation parameter $\beta$. Rather than using (4.13), we apply a stepwise change in $\beta$ of about 10% after which we continue with the implicit scheme for a while. Then we switch to a Crank-Nicolson time integration scheme and use this from then onward. In figure 11 we show the result of particle tracking for one time cycle starting at a certain time instant $t = t_0$. We can conclude that a duct consisting of two segments has a periodicity index of $n = 2$. This immediately raises the issue of the relation between the number of pipe segments considered and the resulting value of the periodicity index. It is worthwhile to investigate ducts with more than two segments as well, which we omit here (see [6]).

### 4.4 Heat transfer computations

As pointed out before we solve the energy equation and the Navier-Stokes equations separately. Heat transfer computations are carried out on the same finite-element grid as used for the flow simulations. Since the Prandtl number is of the order 1, the boundary layer for the velocity and that for the temperature have approximately the same thickness. This makes it easier to use a solution for the velocity field in the procedure for the solution of the energy equation. The simulations are carried out for $Pr = 0.7$ using the periodic boundary conditions as described before. An overall Nusselt
number is computed as described in section 2.3. Since the Prandtl number is constant, the Nusselt number only depends on the Reynolds number. We remark that it suffices to use the flow field from the domain with length $2\ell$, because $\text{Nu}$ is an integral quantity. Although for a certain Reynolds number we find slightly different flow patterns for domains of different length, the Nusselt number is expected to be the same in these cases.

In figure 13 we can observe that the Nusselt number gradually increases until $\text{Re}_H = 300$. For $\text{Re}_H > 300$ the cooling efficiency levels off and varies about a value of 0.71. A further increase of $\text{Nu}$ may only be expected at larger Reynolds numbers. Two Nusselt number definitions are used for
In section 4 we have presented the numerical results of flow simulations in the laminar flow regime. Just below \( \text{Re}_H = 550 \) the flow becomes unsteady and displays an oscillating behaviour, which is periodic in time. However, in turbine conditions the coolant air flow in the duct is known to become turbulent. A rough indication for the Reynolds number based on the hydraulic diameter in this case would be about \( 20\,000 - 30\,000 \). It is clear that we should not use direct numerical simulation (DNS) for this type of flow. The requirements on the number of grid points would be prohibitive.

### 5.1 Turbulator shape

We are interested in the cooling efficiency of turbulated ducts that are produced with the ECD technique (cf. [5]), which allows for a wide variety of turbulator shapes. In the analysis carried out in this chapter, we will consider only four different types of turbulators. These will involve on the one hand commonly used duct shapes and on the other hand a special shape which we would like to analyse in order to determine the enhancement, or
possibly, deterioration in cooling efficiency. Figure 14(b) concerns a case of a deviating turbulator shape. Here the geometry slightly differs from the specified shape as shown in figure 14(a). Another type of deviation is a missing tubulator in a sequence of turbulators, say one out of every five (see figure 14(d)). Our goal is to find a relation between the Nusselt number and the Reynolds number for each shape, thus characterizing the heat transfer for that shape. In the following the shapes, depicted in figure 14(a)-14(d) will be referred to as shape 1-4.

5.2 Flow computations

For a certain turbulator shape the simulation results are presented in figure 15. We have used the package FLUENT which employs a k-ε model. It is clear that the mean flow variables display a periodic behaviour with a periodicity index equal to one. The k-ε model is used in combination with standard wall functions. In order to be able to carry out a model validation, the physical constants are taken such that they correspond to the thermal conditions at which the experimental data have been obtained. For each turbulator shape the flow field is computed for different values of $\Phi_m$ such that the Reynolds number varies in the range 20 000 - 30 000. This Reynolds number is computed from the specified periodic mass flow rate as follows

$$Re_{D_h} = \frac{4\Phi_m}{\pi D_h \eta}. \quad (5.1)$$

For $Re_{D_h} = 25 000$ the $k$ field is shown for different turbulator shapes. The reason that a periodicity index of 2 can be observed in figure 16 for some
Figure 14: Different turbulator shapes obtained by the ECD technique

shapes, is that the geometry is not perfectly periodic itself. These geometries are based on actual photographs of such turbulators.

5.3 Heat transfer computations

In order to be able to solve the periodic heat transfer there are some special constraints. The boundary conditions for the temperature field must either be a constant heat flux or a constant wall temperature. We will opt for the latter, since this matches the experimental data best. As the fluid flows through the periodic domain, its temperature approaches that of the wall. However, the temperature can be scaled in such a way that it behaves in a periodic manner. This is possible under the assumption that eventually the temperature boundary-layer has a constant thickness. We therefore introduce the scaled temperature $\theta$, defined as follows

$$\theta = \frac{T - T_w}{T_b - T_w}$$

(5.2)

where $T_w$ is the wall temperature and $T_b$ the bulk temperature as defined by equation (2.13). Note that the periodic temperature $\theta$ obeys a periodic
condition across the segment length $\ell$:

$$
\frac{T(r, 0, t) - T_w}{T_h(0) - T_w} = \frac{T(r, 2\ell, t) - T_w}{T_h(2\ell) - T_w}. \quad (5.3)
$$

An inlet bulk temperature has to be specified to scale the temperature field. In this approach, as already mentioned above, the fluid properties are not allowed to depend on the temperature.

Local heat transfer coefficients for the individual wall cells are calculated using the temperature difference between the wall cell surface temperature and the cell average temperature of the fluid. Figure 18 shows a plot of these values belonging to the temperature distribution from figure 17(b). To have the proper interpretation of the results we have to compare them with a reference value. The most natural choice would be the values for a smooth duct. Then we can express all Nusselt numbers relative to the smooth duct. To this end we use the Dittus-Boelter correlation (see [2] p.315) which gives the following Nusselt number formula

$$
\text{Nu} = 0.023 \text{Re}_D^{0.8} \text{Pr}^{0.4}. \quad (5.4)
$$

This formula makes sense for a pipe with smooth internal surface and holds in the range $0.7 \leq \text{Pr} \leq 120$ and $2500 \leq \text{Re}_D \leq 1.24 \cdot 10^5$. In figure 19 the Nusselt number is plotted against the Reynolds number; here we have used (5.4) to quantify the Nusselt number for a smooth duct.

We conclude the following: First of all we see that a cooling duct with turbulators has a considerable higher cooling efficiency than a smooth duct. However, this efficiency may decrease dramatically if a turbulated duct has a defect like the one in figure 14(d) where out of five turbulators one is missing. Such a situation may occur when the drilling production process hampers. Furthermore it turns out that turbulator shape 2, which is considered a deviation from a specified shape, actually gives a slight enhancement in heat transfer compared to turbulator shape 1 (see figure 19). A possible explanation for this behaviour is the turbulator becoming more pronounced due to extra corrosion caused in the drilling process. This can be taken into account when tuning the drilling parameters. By a very delicate tuning of these parameters turbulator shape 3 can be obtained. The desired effect of the design of this particular shape is achieved; namely a 20% increase of cooling efficiency is found relative to turbulator shape 1. To validate the numerical results we have compared the results depicted in figure 19 with
the results in [9]. In this paper experimental data on heat transfer are presented for an even larger variety of turbulator shapes. If we compare the Nusselt numbers for turbulator shapes similar to the ones shown in figure 14, we find $100 < \mathrm{Nu}_{D_h} < 400$ like in figure 19.

Acknowledgement: The authors wish to thank Eldim b.v. for supporting this investigation and in particular ir. M. van Dijk for the many inspiring discussions.

References


Figure 15: Isolines of computed steady-state mean quantities at $\text{Re}_{D_h} = 25\,000$ for turbulator shape 1

Figure 16: Isolines of kinetic turbulent energy at $\text{Re}_{D_h} = 25\,000$, uniformly distributed
Figure 17: Computed steady-state quantities at $Re_{D_h} = 25,000$, $Pr = 0.7$ for turbulator shape 1

Figure 18: Local values for the coefficient of heat transfer along the surface [W m$^{-2}$ K$^{-1}$] versus the axial coordinate for turbulator shape 1

Figure 19: Nusselt number versus Reynolds number for different turbulator shapes