Simulation of an Air Flow between Adiabatic and Heated Cooling Fins

AUTHOR(S)
Juhaveikko Ala-Juusela

ABSTRACT
The flow between the cooling fins of the ABB HXR 400L electric motor was simulated numerically by calculating one fin space and its surroundings. FINFLO flow solver, developed at Helsinki University of Technology, was used in the simulations. For the turbulence modelling Chien’s low-Reynolds number $k-\varepsilon$ models were utilized, since in this case $k-\omega$ model did not seem to work properly.

MAIN RESULT
A suitable grid with 750 000 cells was introduced. Different variations of a computational domain and boundary conditions were tested and they affected significantly to the computational results. The results were compared to measurements made by a 'cold' motor, i.e. only the fan was rotated. The calculated velocity was quite near the measured values at the first half of the fin, but differed significantly towards the end of the fin. Also the simulated flow seems to stay between the fins too well.

PAGES
32

KEY WORDS
FINFLO, $k-\varepsilon$-model, $k-\omega$-model, cooling fin

APPROVED BY
Timo Siikonen 26th February 2004
# Contents

Nomenclature 2

1 Introduction 5

2 Flow Equations 5

3 Turbulence Modelling 7
   3.1 $k - \varepsilon$ Turbulence Model .................................................. 7
   3.2 $k - \omega$ Turbulence Model ............................................................. 8

4 Computational Domains and Grids 10

5 Results 14
   5.1 Adiabatic Simulation ................................................................. 14
   5.2 Simulation of a Heated Fin ............................................................ 20

6 Discussion 26

A Velocity distributions of the Adiabatic case 28

B Convergence Histories of the Heated Vertical Case 30

C Velocity and temperature distributions of the Heated cases 31
Nomenclature

\begin{align*}
A & \quad \text{Jacobian matrix } \frac{\partial \hat{F}}{\partial U} \text{ in the } i\text{-direction; cell face area} \\
C & \quad \text{constant} \\
CFL & \quad \text{Courant number} \\
C_p & \quad \text{pressure coefficient} \\
E & \quad \text{total energy} \\
F, G, H & \quad \text{flux vectors in the } x-, y\text{- and } z\text{-directions} \\
Pr & \quad \text{Prandtl number} \\
Q & \quad \text{source term vector} \\
Re & \quad \text{Reynolds number} \\
T & \quad \text{temperature} \\
U & \quad \text{vector of conservative variables} \\
c_p & \quad \text{specific heat at a constant pressure} \\
c_v & \quad \text{specific heat at a constant volume} \\
e & \quad \text{specific internal energy} \\
k & \quad \text{turbulent kinetic energy} \\
\dot{m} & \quad \text{mass flow} \\
p & \quad \text{pressure} \\
\vec{q} & \quad \text{heat flux} \\
t & \quad \text{time} \\
u, \upsilon, w & \quad \text{velocity components in the } x-, y\text{- and } z\text{-directions} \\
\vec{v} & \quad \text{velocity vector} \\
y^+ & \quad \text{dimensionless distance from the wall } (= yu_+ / \nu) \\
\beta, \beta^* & \quad \text{artificial compressibility coefficient, model constants in the } k - \omega \text{ model} \\
\delta_{ij} & \quad \text{Kronecker’s delta} \\
\epsilon & \quad \text{dissipation of the kinetic energy of the turbulence} \\
\theta & \quad \text{temperature difference } (= T - T_\infty) \\
\mu & \quad \text{dynamic viscosity} \\
\nu & \quad \text{kinematic viscosity} \\
\phi & \quad \text{scalar} \\
\rho & \quad \text{density} \\
\tau & \quad \text{shear stress} \\
\omega & \quad \text{specific dissipation rate of turbulent kinetic energy, } \varepsilon / (\beta^* k)
\end{align*}

Superscripts

\begin{align*}
T & \quad \text{transposition} \\
l & \quad \text{left side} \\
r & \quad \text{right side} \\
' & \quad \text{fluctuating component}
\end{align*}
Subscripts

\[ T \quad \text{turbulent} \]
\[ i, j, k \quad i-, j- \text{-} k\text{-component} \]
\[ n \quad \text{normal component} \]
1 Introduction

In the present work a flow between the cooling fins of the ABB HXR 400L electric motor is simulated numerically. This is done by calculating one fin space and its surroundings and utilizing symmetry boundary conditions between the fins. The calculations are performed using the $k - \varepsilon$ model. In this case $k - \omega$ model did not give converged results. The case is complicated, because the inlet flow in a real motor coming from the fan, creates a secondary flow which cannot be fixed as a normal inlet without measuring the flow velocities. The fin space is located on a side of the motor, so the centerline of the fin from a root to a tip is in a horizontal position. Since the fins can also be heated, the buoyance must be taken care of. In this case the periodic boundary condition is used to allow buoyance driven air flow up freely.

In order to study the effect of a buoyancy a separated simulation was made with the centerline of the fins in a vertical direction as if the fins would be on the top of the motor.

In the following, the governing equations and turbulence modelling are firstly described. Next, the computational domain and the grid are depicted and, finally, the results of the simulation are presented and compared with the measurements.

2 Flow Equations

The Reynolds-averaged Navier-Stokes equations, and the equations for the kinetic energy ($k$) and dissipation ($\varepsilon$) of turbulence can be written in the following form

$$\frac{\partial U}{\partial t} + \frac{\partial (F - F_w)}{\partial x} + \frac{\partial (G - G_w)}{\partial y} + \frac{\partial (H - H_w)}{\partial z} = Q$$  \hspace{1cm} (1)

where the unknowns are $U = (\rho, \rho u, \rho v, \rho w, E, \rho k, \rho \varepsilon)^T$. The last equation can be replaced by that of $\omega$. The inviscid fluxes are

$$F = \begin{pmatrix}
p
\rho u + \rho v^2 + p + \frac{2}{3} \rho k \\
\rho w
\rho w u
\rho w u + (E + p + \frac{2}{3} \rho k) u
\rho u k
\rho u e
\end{pmatrix} \quad G = \begin{pmatrix}
pw
pw
pw
pw
pw
\end{pmatrix} \quad H = \begin{pmatrix}
pw
pw
pw
pw
\end{pmatrix} \quad (2)$$

where $\rho$ is the density, the velocity vector by using Cartesian components is $\vec{V} = u \hat{i} + v \hat{j} + w \hat{k}$, $p$ is the pressure, $k$ is the turbulent kinetic energy and $\varepsilon$ its dissipation, and the total energy $E$ is defined as

$$E = \rho e + \frac{1}{2} \rho \vec{V} \cdot \vec{V} + \rho k$$  \hspace{1cm} (3)
where $e$ is the specific internal energy. The viscous fluxes are

$$
F_v = \begin{pmatrix}
0 \\
\tau_{xx} & \tau_{xy} & \tau_{xz} \\
\tau_{yx} & \tau_{yy} & \tau_{yz} \\
\tau_{zx} & \tau_{zy} & \tau_{zz}
\end{pmatrix}
\begin{pmatrix}
0 \\
\frac{u\tau_{xx} + v\tau_{xy} + w\tau_{xz}}{} - q_x \\
\frac{\mu_k(\partial k/\partial x)}{} \\
\frac{\mu_e(\partial e/\partial x)}{}
\end{pmatrix}
\begin{pmatrix}
0 \\
\frac{u\tau_{yx} + v\tau_{yy} + w\tau_{yz}}{} - q_y \\
\frac{\mu_k(\partial k/\partial y)}{} \\
\frac{\mu_e(\partial e/\partial y)}{}
\end{pmatrix}
G_v =

H_v =

Here the stress tensor, $\tau_{ij}$, includes laminar and turbulent components. The fluid is assumed to be Newtonian and, therefore, the laminar stresses are modelled by using Stokes hypothesis. The Reynolds stresses $\rho u_i^j u_j^i$ are included to the stress tensor $\tau_{ij}$.

$$
\tau_{ij} = \mu \left[ \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3}(\nabla \cdot \mathbf{V})\delta_{ij} \right] - \rho u_i^j u_j^i + \frac{2}{3}\rho k\delta_{ij}
$$

For the Reynolds stresses the Boussinesq's approximation

$$
-\rho u_i^j u_j^i = \mu_T \left[ \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3}(\nabla \cdot \mathbf{V})\delta_{ij} \right] - \frac{2}{3}\rho k\delta_{ij}
$$

is utilized. Here $\mu_T$ is a turbulent viscosity coefficient, which is calculated by using a turbulence model, and $\delta_{ij}$ is the Kronecker's delta. In the momentum and energy equations, the kinetic energy contribution $2/3\rho k\delta_{ij}$ has been connected with pressure and appears in the convective fluxes, whereas the diffusive part is connected with the viscous fluxes. The viscous stresses contains a laminar and a turbulent part. Respectively, the heat flux can be written as

$$
q' = -(\lambda + \lambda_T)\nabla T = -\left(\mu \frac{c_p}{Pr} + \mu_T \frac{c_p}{Pr_T}\right)\nabla T
$$

where $\lambda$ is a molecular and $\lambda_T$ a turbulent thermal conductivity coefficient and $Pr$ is a laminar and $Pr_T$ a turbulent Prandtl number, respectively, and $c_p$ is a specific heat at constant pressure. The diffusion of turbulence variables is modelled as

$$
\mu_k \nabla k = \left(\mu + \frac{\mu_T}{\sigma_k}\right) \nabla k
$$

$$
\mu_e \nabla e = \left(\mu + \frac{\mu_T}{\sigma_e}\right) \nabla e
$$
where $\sigma_k$ and $\sigma_\epsilon$ are turbulent Schmidt’s number of $k$ and $\epsilon$. The pressure is calculated from an equation of state $p = p(\rho, \epsilon)$, which, with a perfect gas assumption utilized in this study, is written as

$$p = (\gamma - 1)(E - \frac{\rho \bar{V} \cdot \bar{V}}{2} - \rho k) = (\gamma - 1)\rho e$$

(10)

where $\gamma$ is the ratio of specific heats $c_p/c_v$. Since this case is essentially incompressible, pressure differences are solved instead of pressure.

The components of the source term $Q$ are non-zero in buoyance terms and in turbulence model equations. The buoyance terms are

$$Q = \begin{pmatrix}
-(\rho - \rho_0)g_i \\
-(\rho - \rho_0)g_j \\
-(\rho - \rho_0)g_k \\
-(\rho - \rho_0)(g_iu + g_jv + g_kw)
\end{pmatrix}$$

(11)

where $\rho_0$ is the reference density and $\bar{g}$ is gravitational acceleration.

In the present study an artificial compressibility approach is used to determine the pressure. The flux calculation is a simplified version of the approximate Riemann-solver utilized for compressible flows [1]. It should be noted that in this approach the artificial sound speed affects the solution, but the effect is of the second-order and is not visible as the grid is refined. The solution method is described in [2].

3 Turbulence Modelling

3.1 $k - \epsilon$ Turbulence Model

As mentioned, turbulent stresses resulting from the Reynolds averaging of the momentum equation are modelled by using Boussinesq’s approximation (6). The turbulent viscosity coefficient $\mu_T$ is determined by using Chien’s [3] low-Reynolds number $k - \epsilon$ model from the formula

$$\mu_T = c_\mu \rho \frac{k^2}{\epsilon}$$

(12)

where $c_\mu$ is a empirical coefficient. The source term of Chien’s model is

$$Q = \begin{pmatrix}
P - \rho \epsilon - 2\mu \frac{k}{y_n^2} \\
c_1 \frac{\epsilon}{k} P - c_2 \frac{\rho \epsilon^2}{k} - 2\mu \frac{\epsilon}{y_n^2} e^{-y^+ / 2}
\end{pmatrix}$$

(13)

where $y_n$ is the normal distance from the wall, and the dimensionless distance $y^+$ is defined by

$$y^+ = y_n \frac{\rho n \tau_w}{\mu} = y_n \sqrt{\frac{\rho n \tau_w}{\mu}} \approx y_n \left[ \frac{\rho |\nabla \times \bar{V}|}{\mu} \right]_{w}^{1/2}$$

(14)
Here $u_r$ is friction velocity and $\tau_w$ is shear stress on the wall, and the connection between them is $u_r = \sqrt{\frac{\tau_w}{\rho}}$. The unknown production of the turbulent kinetic energy is modelled using Boussinesq’s approximation (6)

$$
P = -\rho u_i u_j \frac{\partial u_i}{\partial x_j}
$$

$$
= \left[ \mu_T \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) - \frac{2}{3} \delta_{ij} \rho \frac{\partial k}{\partial x_j} \right] \frac{\partial u_i}{\partial x_j}
$$

(15)

The turbulence model presented above contains empirical coefficients. Those are given by [4]

$$
c_1 = 1.44 \quad \sigma_k = 1.0
$$

$$
c_2 = 1.92(1 - 0.22e^{-\frac{\rho \mu}{30 \sigma}}) \quad \sigma_\epsilon = 1.3
$$

(16)

where the turbulence Reynolds number is defined as

$$
Re_T = \frac{\rho k^2}{\mu \epsilon}
$$

(17)

Chien’s model is very robust, but it has several shortcomings. It usually overestimates the turbulence level and is not performing well in a case of an increasing pressure gradient.

### 3.2 $k - \omega$ Turbulence Model

In order to improve the near-wall behaviour of a $k - \epsilon$ model, a mixture of the $k - \epsilon$ and $k - \omega$ models, known as Menter’s $k - \omega$ SST model [5, 6, 7], has gained increasing popularity in recent years. Menter’s $k - \omega$ SST-model is a two-equation turbulence model where the $k - \omega$-model is utilized in a boundary layer and outside of that the turbulence is modelled with the $k - \epsilon$-model. However, the $\epsilon$-equation is transferred into the $\omega$-equation in order to allow a smooth change between the models. In the SST-model the turbulent stress is limited in a boundary layer in order to avoid unrealistic strain-rates, which are typical to the Boussinesq eddy viscosity models. The equations for $k$ and $\omega$ are

$$
\rho \frac{\partial k}{\partial t} + \rho u_j \frac{\partial k}{\partial x_j} = P - \beta^k \rho k \omega
$$

(18)

$$
+ \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]
$$

$$
\frac{\partial \omega}{\partial t} + \rho u_j \frac{\partial \omega}{\partial x_j} = \frac{\gamma \mu}{\mu_T} P
$$

(19)

$$
+ \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right]
$$

$$
+ 2 \rho \frac{1 - F_1}{\sigma_{\omega} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
$$
The model coefficients in Eqs. (18) and (19) are obtained from
\[
(\sigma_k \sigma_\omega \beta)^T = F_1 (\sigma_k \sigma_\omega \beta)_1^T + (1 - F_1) (\sigma_k \sigma_\omega \beta)_2^T
\]  
(20)
with the following values
\[
\begin{align*}
\sigma_{k1} &= 1.176 & \sigma_{\omega1} &= 2.0 & \beta_1 &= 0.075 \\
\sigma_{k2} &= 1.0 & \sigma_{\omega2} &= 1.168 & \beta_2 &= 0.0828
\end{align*}
\]
Coefficients \( \kappa \) and \( \beta^* \) have constant values 0.41 and 0.09. Coefficient \( \gamma \) is calculated from
\[
\gamma = \frac{\beta}{\beta^*} - \frac{\kappa^2}{\sigma_\omega \beta^*}
\]  
(21)
Term \( P \) in Eqs. (18) and (19) is a production of the turbulent kinetic energy and it is calculated from Eq. (15). The last term in the \( \omega \)-equation originates from the transformed \( \epsilon \)-equation and it is called a cross-diffusion term. The switching function which governs the choice between the \( \omega \)- and the \( \epsilon \)-equations is
\[
F_1 = \tanh (\Gamma^4)
\]  
(22)
where
\[
\Gamma = \min \left( \max \left( \frac{\sqrt{\kappa}}{\beta^* \omega d} ; \frac{500 \nu}{\omega d^2} \right) ; \frac{4 \rho \sigma_\omega^2 k}{C_D k \omega d^2} \right)
\]  
(23)
The first term is turbulent length scale divided with the distance from the walls. This ratio is around 2.5 in a logarithmic layer and approach zero in a outer layer. The second term has a value of \( \geq 1 \) only in a viscous sublayer. The meaning of the third term is to ensure stable behaviour of \( F_1 \) when the value of \( \omega \) in the free stream is small. \( C_D k \omega \) is the positive part of the cross diffusion term
\[
C_D k \omega = \max \left( \frac{2 \rho \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}}{\sigma_\omega^2 \omega \frac{\partial x_j}{\partial x_j}} ; C_D k \omega \min \right)
\]  
(24)
where \( C_D k \omega \min \) is lower limit of the cross diffusion term.
In the original SST-model the eddy viscosity \( \mu_T \) is defined as
\[
\mu_T = \frac{a_1 \rho k}{\max (a_1 \omega ; |\Omega| |F_2 F_3|)}
\]  
(25)
where \( a_1 = 0.31 \) and \( |\Omega| = \sqrt{2 |\Omega_{ij} \Omega_{ij}|} \), with \( \Omega_{ij} \) being the vorticity tensor
\[
\Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)
\]  
(26)
Above term \( F_2 \) is a switching function that disables the SST limitation outside of the boundary layers. Function \( F_2 \) works like function \( F_1 \) except its value remains as one farther in the outer boundary layer. It is defined as
\[
F_2 = \tanh \left( \Gamma^2 \right)
\]  
(27)
where

\[ \Gamma_2 = \max \left( \frac{2\sqrt{k}}{\beta \omega d} \cdot \frac{500 \nu}{\omega d^2} \right) \]  

(28)

In Eq. (25) the lower limit of the nominator is based on Bradshaw’s assumption, on which the turbulent shear stress in the boundary layer depends on \( k \) as follows:

\[ |\rho u'^{\prime} v'^{\prime}| = a_1 \rho k \]  

(29)

Thus the traditional Kolmogorov-Prandtl-expression \( \mu_T = \rho k / \omega \) is used to the limit

\[ \mu_T = \frac{|\rho u'^{\prime} v'^{\prime}|}{|\Omega_{ij}|} = \frac{a_1 \rho k}{|\Omega_{ij}|} \]  

(30)

This is called the SST-limitation of \( \mu_T \). SST-limitation improves significantly the behavior of the model in boundary layers that has unfavorable pressure gradient, in which cases the traditional model clearly overestimates the turbulent viscosity.

The meaning of function \( F_3 \) is to prevent an activation of the SST-limitation near the rough walls [7, 8]. Function \( F_3 \) is defined as

\[ F_3 = 1 - \tanh \left[ \left( \frac{150 \nu}{\omega d^2} \right)^4 \right] \]  

(31)

where \( d \) is a distance from the walls.

4 Computational Domains and Grids

Dimensions of the fin and the fin gap are given in Figs. 1 and 2. The fin is 1116 mm long and 60 mm high. Before and after the fins there is a 12 mm long plate. The original geometry contained roundings at the ends of the fin but those were left out of the grid because the grid structure would have got too complicated and the effect of the roundings is probably insignificant on the flow field. The angle of the sides is 6°.
Fig. 2: Dimensions of the fin space.

The velocity distribution shown in Fig. 3 is given as an inlet condition. The distribution is constant in the direction of the left wall of the grid. The distribution is adjusted so that the simulated flowfield 20 mm from the inlet matches the measured values. The pressure is extrapolated from the computational domain. On the solid walls the velocities and the kinetic energy of the turbulence are set to zero. Zero gradients are assumed at the outlet and pressure is given. Outlet boundary conditions are also used in farfield boundaries, where the flow can be either into the domain or out of it. Periodic boundary condition is used for the upper and the lower boundaries. Although the case is not symmetric, the periodic boundary gives a good approximation for the flow coming from the other fin spaces. Otherwise one would have to calculate a half of the motor. A sketch of the boundary conditions is shown in Fig. 4.

Fig. 3: Velocity distribution at the inlet.
Fig. 4: Boundary conditions.

The computational grid is shown in Fig. 5. The grid ranges 0.2 m from the fins into the horizontal direction. The constant velocity boundary, where the velocity is taken from the measurements, is located 0.5 m behind the inlet in order to avoid problems in the boundary treatment. The grid structure between the fins and in front of the fin ends is depicted in Fig. 6, and the surface grid is shown in Fig. 7. Grid dimensions are available in Table 1.

Fig. 5: Computational grid, every second grid line is drawn.
Table 1: Dimensions of the grid.

<table>
<thead>
<tr>
<th>Explanation</th>
<th>Block</th>
<th>i</th>
<th>j</th>
<th>k</th>
<th>Σ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fin head (upper)</td>
<td>1</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Before the fin gap</td>
<td>2</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Before the fin gap</td>
<td>3</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Fin head (lower)</td>
<td>4</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Fin gap</td>
<td>5</td>
<td>80</td>
<td>32</td>
<td>48</td>
<td>122 880</td>
</tr>
<tr>
<td>Fin gap</td>
<td>6</td>
<td>80</td>
<td>32</td>
<td>48</td>
<td>122 880</td>
</tr>
<tr>
<td>Fin end (upper)</td>
<td>7</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>After the fin gap</td>
<td>8</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>After the fin gap</td>
<td>9</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Fin end (lower)</td>
<td>10</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Surrounding space</td>
<td>11</td>
<td>128</td>
<td>8</td>
<td>24</td>
<td>24 576</td>
</tr>
<tr>
<td>Surrounding space</td>
<td>12</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Surrounding space</td>
<td>13</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Surrounding space</td>
<td>14</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Surrounding space</td>
<td>15</td>
<td>24</td>
<td>32</td>
<td>48</td>
<td>36 864</td>
</tr>
<tr>
<td>Surrounding space</td>
<td>16</td>
<td>16</td>
<td>8</td>
<td>24</td>
<td>3 072</td>
</tr>
<tr>
<td>Surrounding space</td>
<td>17</td>
<td>24</td>
<td>8</td>
<td>24</td>
<td>4 608</td>
</tr>
<tr>
<td>Inlet</td>
<td>18</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Inlet</td>
<td>19</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Inlet</td>
<td>20</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Inlet</td>
<td>21</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Outlet</td>
<td>22</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Outlet</td>
<td>23</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Outlet</td>
<td>24</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Outlet</td>
<td>25</td>
<td>2</td>
<td>32</td>
<td>48</td>
<td>3 072</td>
</tr>
<tr>
<td>Outlet</td>
<td>26</td>
<td>2</td>
<td>8</td>
<td>24</td>
<td>384</td>
</tr>
<tr>
<td>Secondary flow inlet</td>
<td>27</td>
<td>2</td>
<td>8</td>
<td>24</td>
<td>384</td>
</tr>
<tr>
<td>Farfield boundary</td>
<td>28</td>
<td>17</td>
<td>8</td>
<td>2</td>
<td>256</td>
</tr>
<tr>
<td>Farfield boundary</td>
<td>29</td>
<td>128</td>
<td>8</td>
<td>2</td>
<td>2 048</td>
</tr>
<tr>
<td>Farfield boundary</td>
<td>30</td>
<td>24</td>
<td>8</td>
<td>2</td>
<td>384</td>
</tr>
</tbody>
</table>

Total 748 416
5 Results

5.1 Adiabatic Simulation

The $k - \epsilon$ model was used in the simulation. The calculation time was 287 hours with a Silicon Graphics Origin 2000 computer using one processor. The simulation was started on the third grid level and the result was interpolated on the finer levels until the first level was reached. The convergence histories on the finest level are shown in Figures 8, 9 and 10. The density residual and the momentum residuals are converged within 2 000 cycles or more likely already in the coarser grid level results. The budget of the turbulent kinetic energy is changing evenly until it starts to oscillate after 8 000 cycles.
Fig. 8: Convergence history of $\|\Delta \rho\|_2$ (left) and $\|\Delta \rho u\|_2$ (right) as calculated with the $k-\varepsilon$-turbulence model.

Fig. 9: Convergence history of $\|\Delta \rho u\|_2$ (left) and the total mass (right) as calculated with the $k-\varepsilon$-turbulence model.

The mass flux between the fins as scaled with the inlet mass flux is shown in Fig. 11. The fin starts at the location 0.012 m and is in its full width at the location 0.046 m. The mass flow decreases rapidly between these points, but after that a vortex brings more mass to the finspace. Then the mass flow between the fins slowly decreases until at the end of the fins it starts to increase rapidly. After the fins mass flow suddenly increases and the there is strange fluctuation.

An overview of the flow in the computational domain can be seen in Fig. 12 where streamlines coloured by the velocity distribution are shown. The flow near the fins is somewhat vortical and near the end of the fins there are large vortices on the farfield boundary side of the domain. The velocity distribution at a horizontal plane in the middle of the grid and vertical planes normal to the flow direction are shown in Fig. 13. From the velocity distribution it can be seen how the flow stays between the fins and does not spread into the surroundings. The corresponding distributions of the turbulent kinetic energy
Fig. 10: Convergence history of the budget of the kinetic energy of turbulence (left) and turbulent fluctuation velocity over the main velocity (right) as calculated with the $k-\varepsilon$-turbulence model.

Fig. 11: Mass flux between the fins as scaled with the inlet mass flux. Vertical lines show the places of the ends of the fins.

and the turbulent viscosity are shown in Fig. 14. The turbulent kinetic energy is mainly produced just outside the fin space as well as after the fins. Turbulent viscosity is created when the flow behind the fins encounters the ambient air. There is also a massive growth of the turbulent viscosity inside the vortices.

Comparison between the simulated and measured velocities 200 mm before the inlet and 350 mm after the inlet is shown in Fig. 15. The velocity distribution shows that the jet is not spreading enough from the fin space to the surroundings.

The flow field in the fin space is measured by rotating the fan with a separated motor [9]. That means that the fins are not warmer than the surrounding air. The measurements were made at the locations of 10 mm, 100 mm, 350 mm
and 1050 mm from the edge of the casing of the fan, which is located at the station $x = 20$ mm in the computational grid. The measurements are made at each location mentioned above at a centerline between the fins and at four vertical sweeps between the fins: 5 mm, 15 mm, 30 mm and 55 mm from the bottom of the finspace. The locations of the distributions are depicted in Fig. 16. Simulated distributions of the $u$-velocity are shown in Fig. 17 and inter-
Fig. 14: Distribution of the turbulent kinetic energy (left) and distribution of the turbulent viscosity (right) at the center plane of the computational domain as calculated with $k-\varepsilon$ model.

Fig. 15: Simulated and measured velocities 200 mm before the inlet and 350 mm after the inlet.

Polulated distributions of the measured values [9] are shown in Fig. 18. Similar figures for the $w$-velocity are shown in Appendix A, Figs. 30 and 31. Measured and simulated $w$-velocity distributions at the centerline of the fins are compared in Fig. 19 and vertical sweeps in Appendix A, Figs 32 - 35. These figures also show that the calculated velocity at the end of the fin is too high. Furthermore, the overall agreement with the experiments is poor.
Fig. 16: Locations of the distributions in $z$- and $y$-directions.

Fig. 17: Velocity distribution at locations 10 mm, 100 mm, 350 mm and 1050 mm from the fan cover.
Fig. 18: Measured velocity distribution at locations 10 mm, 100 mm, 350 mm and 1050 mm from the fan cover.

Fig. 19: Velocity distribution at the centerline of the finspace.

5.2 Simulation of a Heated Fin

In this case the finspace was located in a horizontal and also in a vertical position. The heat flux used was 2000 W/m², which is too high, but the correct value was not available as the simulations were started. The $k - \epsilon$ model was used also in these simulations. The calculation times were 140 hours each with a Silicon Graphics Origin 2000 computer using one processor. The simulations were started on the third grid level and the result were interpolated on the finer levels until the first level was reached. The convergence histories on the finest level are shown in Figs. 20, 21 and 22 for the horizontal case and in Figs.
36, 37 and 38 in Appendix B, for the vertical case. The density residual and the momentum residuals are converged within 1 500 cycles. After that a flow is oscillatory and the budgets at the turbulence variables are not stabilizing.

\[ \| \Delta \rho \|_2 \text{ (left)} \quad \| \Delta pu \|_2 \text{ (right)} \]

**Fig. 20:** Convergence history of $\| \Delta \rho \|_2$ (left) and $\| \Delta pu \|_2$ (right) in case of a heated horizontal fin.

\[ \text{Averaged pressure (N/m)} \]

**Fig. 21:** Convergence history of $\| \Delta pu \|_2$ (left) and the total mass (right) as calculated fins in horizontal direction.

An overview of the flow in the computational domain in heated horizontal and vertical case can be seen in Fig. 23 where streamlines coloured by the velocity distribution are shown. Also in these cases the flow near the fins is somewhat vortical and near the end of the fins there are large vortices on the farfield boundary side of the domain.

The mass flux between the fins as scaled with the inlet mass flux is shown in Fig. 24. The mass flow behaves differently from the adiabatic case as the mass flow is decreasing rapidly at the end of the fin and there is no fluctuation after the fin. The mass flow at the end of the fin is around 60% of the inlet value as it is around 80% in the adiabatic case.
Fig. 22: Convergence history of the budget of the kinetic energy of turbulence (left) and turbulent fluctuation velocity over the main velocity (right) as calculated fins in a horizontal direction.

Fig. 23: Streamlines colored with the velocity in case of heated horizontal (left) and vertical (right) fins.
**Fig. 24**: Mass flux between the fins as scaled with the inlet mass flux. Vertical lines show the places of the ends of the fins.

The velocity distributions of the horizontal case at a horizontal plane in the middle of the grid and vertical planes normal to the flow direction are shown in Fig. 25. The corresponding distributions of the turbulent kinetic energy and the turbulent viscosity are shown in Fig. 26. Corresponding distributions of the vertical case is shown in Appendix D, Figs. 39 and 40. The distributions of the both cases are quite similar.

**Fig. 25**: Velocity distributions at the center plane of the computational domain (left) and in cross sections (right) as calculated fins in a horizontal direction.

Also in this case it can be seen that the flow stays between the fins and does not spread into the surroundings. The turbulent kinetic energy is mainly produced just outside the fin space as well as after the fins and the distribution is similar to the adiabatic case. Turbulent viscosity is created when the flow encounters the ambient air. There is also a high-level turbulent viscosity area behind the fins, that is much larger than in the adiabatic case.
Fig. 26: Distribution of the turbulent kinetic energy (left) and distribution of the turbulent viscosity (right) at the center plane of the computational domain as calculated fins in a horizontal direction.

Comparison between the simulated and the measured (a cold motor) velocities 200 mm before inlet and 350 mm after inlet is shown in Fig. 27. The velocity distribution shows also in this case that the jet is not spreading enough from the finspace to the surroundings.

![Graph](image)

**Fig. 27:** Simulated and measured velocities 200 mm before the inlet and 300 mm after the inlet.

Measured and simulated \( u \)-velocity distributions at the centerline between the fins are compared in Fig. 28 and vertical sweeps in Appendix D, see Figs. 41 - 44. The distributions are quite similar to the adiabatic case except at the last station, 1050 mm, where the velocity is quite near the measured one in this case.
Fig. 28: Velocity distribution at the centerline of the heated finspace.

Temperature distributions on the surfaces of the fins and at the center plane of the computational domain of the horizontal case are shown in Fig. 29 and the corresponding distributions for the vertical case in Appendix D, Fig. 45. There is no measured data of the temperatures, so the results are just qualitative. The temperature of the ambient air remains low and the temperature rise is concentrated between the fins.

The average surface temperature is 366 K in both cases and the average heat transfer coefficient

$$\bar{h} = \frac{q}{A_s(T_\infty - T_s)} = 27.4 \text{W/m}^2\text{K}$$  \hspace{1cm} (32)

where $q$ is the amount of transferred heat, $A_s$ area of the surface, $T_\infty$ temperature of the ambient air and $T_s$ is the surface temperature.

Fig. 29: Temperature distributions on the surfaces of the fins (left) and at the center plane of the computational domain (right) as calculated fins in a horizontal direction.
6 Discussion

A flow between the cooling fins of an electric motor has been simulated. This case was particularly difficult, because one of the boundaries must allow air flow into the domain. Also periodic boundary conditions were used over large areas. Five different computational domains and several boundary conditions were tested during the project. The selected computational domain and boundary conditions affected significantly to the simulation result. For example, larger surrounding space resulted in a large vortex which finally caused the air 1.5 m from the motor to move at a speed of 10 m/s, which is clearly unphysical. The original idea was to let the inlet flow aspirate secondary air freely from the farfield boundary. This approach led to excessive secondary air speeds and was replaced with a given constant velocity profile, which was estimated from the measurements.

The residuals of the solution are still oscillating at the end of the simulations. This means that the solutions are not converged. However, the solution between the fins is probably quite stable with the $k - \varepsilon$ model. Simulations with the $k-\omega$-turbulence model did not converge at all. The overall agreement with the experiments is poor. The flow stays between the fins and does not spread enough into the ambient air as it did in the experiments. The behaviour of the boundary conditions in the outlet boundaries is a bit questionable as there are peculiar suction type phenomena on the outlets. The effect of the boundary treatment should be further studied. The spreading of the jet into a free space and also near a solid wall are the basic cases to test the boundary conditions. This could be done as a 2D calculation similar to the study of Rautahêimo [10].

It seems that a simulation of a single finspace is not working properly and, therefore, a half of the motor should be modelled in order to get reliable results. This kind of simulations need much more computer capacity and, consequently, the density of the grid must be decreased near the walls at the cost of the accuracy of the results. The computing capacity is fortunately developing so fast that accurate simulations with workstations are soon feasible even in the present application.
References


A  Velocity distributions of the Adiabatic case

**Fig. 30:** Velocity distribution at locations 10 mm, 100 mm, 350 mm and 1050 mm from the fan cover.

**Fig. 31:** Measured velocity distributions at locations 10 mm, 100 mm, 350 mm and 1050 mm from the fan cover.
Fig. 32: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 10 mm from the beginning of the fin.

Fig. 33: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 100 mm from the beginning of the fin.

Fig. 34: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 350 mm from the beginning of the fin.

Fig. 35: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 1050 mm from the beginning of the fin.
B Convergence Histories of the Heated Vertical Case

**Fig. 36:** Convergence history of $\|\Delta \rho \|_2$ (left) and $\|\Delta \rho u \|_2$ (right) as calculated fins in horizontal direction.

**Fig. 37:** Convergence of $\|\Delta \rho u \|_2$ (left) and the total mass (right) as calculated fins in a vertical direction.

**Fig. 38:** Convergence history of the budget of the kinetic energy of turbulence (left) and turbulent fluctuation velocity over the main velocity (right) as calculated fins in vertical direction.
C Velocity and temperature distributions of the Heated cases.

Fig. 39: Velocity distribution (left) at the center plane of the computational domain and in cross sections as calculated fins in a vertical direction.

Fig. 40: Distribution of the turbulent kinetic energy (left) and distribution of the turbulent viscosity (right) at the center plane of the computational domain as calculated fins in a vertical direction.
Fig. 41: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 10 mm from the beginning of the fin.

Fig. 42: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 100 mm from the beginning of the fin.

Fig. 43: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 350 mm from the beginning of the fin.

Fig. 44: Velocity distributions in y-direction 5, 15, 30 and 55 mm from the bottom of the fin gap, at station 1050 mm from the beginning of the fin.
Fig. 45: Temperature distributions on the surfaces of the fins (left) and at the center plane of the computational domain (right) as calculated fins in a vertical direction.