APPROXIMATIONS OF NUMERICAL ERRORS AND BOUNDARY CONDITIONS IN A DRAFT TUBE

John Bergström
Luleå University of Technology
SE-971 87 Luleå
Sweden
E-mail: John.Bergstrom@mt.luth.se

ABSTRACT
A simulation of the three-dimensional flow field in the Turbine 99 draft tube using a RST closure is reported in this paper. The best efficiency point measurements were used as boundary conditions. The report focuses on boundary conditions, grid quality, numerical error, analysis of the resulting flow field and comparison to experimental data. The incomplete measurements at the inlet made it necessary to estimate boundary conditions for the radial velocity, the Reynolds stresses and the turbulent length scale. A block-structured grid was constructed to achieve a grid of high quality, which made it necessary to reconstruct the initial surface topology of the draft tube. The numerical errors, grid and iterative errors were estimated to 3%-7% and 0.8-0.002%, respectively. Comparison with pressure measurements at the upper and lower centerline yielded satisfactory results for the finest grid (700000 grid points). Comparison with velocity measurements at a cross section further downstream yielded qualitatively good results for the main flow but poor agreement for the secondary flow. The flow field had several interesting flow phenomena including the vortex below the runner cone and a large separated region at the outlet.

INTRODUCTION
The Turbine 99 workshop is aimed to be a follow-up to the GAMM Workshop held at EPFL in Switzerland, 1989 (Sottas and Rhyning 1993) where the most advanced numerical simulations at that time were used. The goal with the Turbine 99 workshop is to assess current state-of-the art numerical simulations of the flow field in a hydraulic turbine draft tube by comparing numerical calculations with LDV and pressure measurements.

The draft tube is a scale model (1:11) of the real draft tube belonging to the Hölluforsen hydro power plant. It is situated in the Indals River and was started in 1949. The real plant has a head of 24.9 m, delivers 140 MW (three turbines), rotates with 125rpm and has a flow rate of 149.0 m$^3$/s. The model turbine has a head of 4.5 m, speed of 595 rpm, flow rate of 0.533 m$^3$/s and delivers approximately 32 kW. The geometry of the draft tube is a typical design from the 1940’s with a sharp outer corner in the bend (figure 1).

The experiments were made at Vattenfall Utveckling, Ålkarleby, Sweden (Andersson and Karlsson, 1999a and 1999b). The data supplied for the workshop were conducted at 60% load, close to the highest efficiency for the system. Two cases were measured (both at a head of 4.5 m), one at the highest efficiency point (Q=0.533 m$^3$/s) on the propeller curve and one at a higher flow rate (Q=0.544 m$^3$/s). Laser Doppler Velocity measurements at one cross section further downstream of the draft tube were conducted as well as the inlet boundary conditions. Pressure measurements at several locations at the walls were also conducted.

One engineering quantity, the pressure recovery factor, will be used, in conjunction with detailed data, to compare the measurements with the calculations. The present contribution will focus on the construction of the grid, numerical errors and the boundary conditions.

It is important that the solution from a CFD calculation is close to being grid independent. However, it is in most cases impossible to achieve a completely grid independent solution. Therefore it is necessary to estimate the errors. The errors in numerical flow simulations can be subdivided in two parts, grid errors and iterative errors. Grid errors (discretization errors) were estimated by using Richardson extrapolation to estimate the solution for a grid having infinitely many grid points. This estimation can then be used to calculate the error. These methods require the use of several grids in a sequence (at least three), (Bergström and Gebart 1999), (Ferziger and Peric, 1996).
Iterative errors can be calculated in several different ways, e.g. using the residuals or to calculate an estimate of the real iterative error, (Ferziger and Peric, 1996). In the present case the residuals were used. Because of the coupling between the iterative error and the residual, the reduction of the residuals of a certain order of magnitude implies that the error has decreased by a comparable amount, (Ferziger and Peric, 1996).

Closely connected to the convergence rate and the numerical error is the grid quality. If a poor grid is generated, this might result in poor convergence or divergence. In this case, where a block-structured grid is used, it is important how the geometry is divided into subdomains (or “blocks”). Keeping the control volume edges as orthogonal as possible was one of the problems which were encountered when trying to generate the grid for the Turbine 99 case.

One of the challenges in engineering calculations is how to treat boundary conditions at inlet and outlet. Such a problem, as in the current case, is the incompleteness of the boundary conditions, i.e. not all the necessary inlet and outlet conditions are given as input data for the calculations. This missing information must be estimated, guessed or in some way prescribed. This is a common problem when trying to simulate “real” cases. Usually we do not know all velocity components, Reynolds stresses or length scales that must be specified for a simulation. In the workshop case there is probably much more information available than what is common. More commonly the only available information would be the flow rate and the runner speed for a draft tube simulation. One way to get all the necessary information at the inlet of the draft tube would be to simulate the flow field separately in the runner. However, a runner simulation would also need inlet information that might not be so easily obtained (but probably easier than for the draft tube inlet).

Starting by describing the numerical method this report continues with the turbulence model, treatment of the boundary conditions, grid generation, numerical errors, comparison to experimental data and finally some required results for the workshop.

NUMERICAL METHOD

The code used for solving the governing equations is the parallel version of the commercial code CFX (AEA Technology, 1997). It is a finite volume based code using a structured non-staggered multi-block grid. To avoid checkerboard oscillations in pressure and velocities, the Rhie-Chow (1983) interpolation method is used. The data transfer between blocks is done by the introduction of dummy cells outside the boundary of each block. This will make each block overlap a neighbor block. The interior values in one block will become the boundary conditions for the neighbor block and vice versa (AEA Technology, 1997).

All terms in all equations are discretized using second-order centered differencing apart from the convective terms in the momentum equations. The convective terms are discretized using higher order upwind, a second-order method. The convective term in the remaining equations are discretized using hybrid differencing.

The PISO algorithm (Jang et. al, 1986) is used for the pressure-velocity coupling. This algorithm was found to give better iterative convergence rate than the SIMPLEC algorithm. The PISO algorithm (which is originally a time-marching procedure) uses one or several correction steps (additional pressure correction equations), (Jang et. al, 1986).

TURBULENCE MODEL

The turbulence model is a Reynolds stress model, (Daly and Harlow, 1970), (Rotta, 1972), (Naot et. al, 1970). It uses individual equations for the Reynolds stresses and one equation for the dissipation. The terms that must be modeled in the Reynolds stress equation are the turbulent diffusion and the pressure strain. The turbulent diffusion is modeled using a gradient-type model (Daly and Harlow, 1970). The pressure strain is written as the sum of three terms. A return to isotropy term by (Rotta, 1972), a return to isotropy of production term by (Naot et. al., 1970) and a wall reflection term (Gibson and Launder, 1978), which is omitted in the present turbulence model (AEA Technology, 1997).

For the dissipation tensor it is assumed that the turbulence is locally isotropic. A modeled $\varepsilon$-equation similar to the one used in the k-$\varepsilon$ model is used. More details about the model can be found in (Bergström, 1997) and (AEA Technology, 1997).

BOUNDARY CONDITIONS

Inlet and outlet boundary conditions

At the inlet, the measurements from the highest efficiency point on the propeller curve (case T) at the Ia(1) radius were used. At this location the axial (U) and tangential (V) velocity components were measured. Also the root mean square values of U and V, $u'$ and $v'$ and the cross correlation of U and V, $u'v'$

![Fig. 1 The Turbine 99 draft tube geometry.](image)
were measured. This is not enough information for a simulation using the current Reynolds stress model. The missing data were the radial velocity, one of the normal Reynolds stresses \( w'w' \), all Reynolds shear stresses and the turbulent dissipation which must be specified for the current turbulence model. The velocities were not measured all the way to the walls (the draft tube wall and the rotating runner cone wall) so some estimation of the velocity variation from the last measurement point up to the wall must be done.

**Radial velocity**

The radial velocity was computed assuming that the velocity vector was parallel to the walls at the runner cone wall and the outer draft tube wall. The angle of the walls at the inlet measured from a vertical plane was \( \theta = 16.8 \) degrees at the runner cone wall and \( \theta = -6.9 \) degrees at the outer draft tube wall. This angle was used to calculate the radial velocity as:

\[
\vec{u}_r = \vec{u}_{axial} \tan(\theta)
\]

where the angle \( \theta \) was assumed to vary linearly from the runner cone to the outer wall (figure 2).

**Axial and tangential velocities**

The axial and tangential velocities were measured from \( r=103.5 \) mm to \( r=235.9 \) mm so the variation close to the wall (\( r_{inner}=98.09 \) mm \( r_{outer}=236.46 \) mm) had to be estimated. At the outer draft tube wall the velocity was specified close enough to the wall to make a linear interpolation to the no-slip condition at the wall possible.

At the inner wall (the runner cone) the velocity profile was assumed to have the same boundary layer thickness as at the outer wall. Then a fourth order polynomial velocity profile was assumed close to the inner wall. This requires the value and the derivative of the velocity at the wall and at the outer edge of the boundary layer. Of these four conditions only the derivative at the wall could not be calculated from experiments. Then the derivative \( (dU/dr) \) at the inner wall was adjusted until the inner and outer boundary layer had approximately the same thickness (~ 4 mm). The final velocity profile can be seen in figure 3.

![Fig. 2 Assumed radial velocity.](image)

**Reynolds Stresses**

To calculate the Reynolds stresses from the given rms values of the velocities it was assumed that
The Reynolds stresses were then linearly interpolated from the experimental data (figure 5 shows the rms values). The \(u'\) and \(v'\) profiles are quite similar so it was assumed that \(w'=v'\), i.e. that the turbulence is approximately isotropic at the inlet of the draft tube. The only Reynolds shear stress given was \(u'v'\). This stress was about 10% of the normal stresses. Because no information about the remaining shear stresses (\(u'w'\) and \(v'w'\)) were given and the fact that the \(u'v'\) stress was small, all Reynolds shear stresses were set to zero:

\[
\overline{u'u'} = u'u' (2)
\]

Both the velocities and the Reynolds stresses were assumed to be axisymmetric at the inlet. This made it necessary to transform both these quantities depending on the azimuthal positions at the inlet of the draft tube.

\[
\overline{u'v'} = v'w' = u'w' = 0 \tag{3}
\]

**Turbulent dissipation**

The turbulent dissipation \(\varepsilon\) was calculated by assuming that

\[
\varepsilon = C_\mu \frac{k^{3/2}}{l} \tag{4}
\]

where \(k\) is the turbulent kinetic energy (which can be calculated from the normal Reynolds stresses) and \(C_\mu\) is a model constant (same as in the \(k-\varepsilon\) model \(C_\mu = 0.09\)). In order to use this relationship the turbulent length scale or integral scale \(l\) has to be specified. (Tennekes and Lumley, 1972) define \(l\) as the size of the largest eddies or the width of the flow. In this case it seemed reasonable to assume that the largest eddies were of the same size as the smallest distance between the runner blades. The Kaplan runner in the test case has 5 blades and the runner blade angle was about 25 degrees measured from a horizontal plane (Urban Anderson, Personal communication). The smallest distance at the mean radius (167 mm) was then 0.09 m.

**Wall boundaries**

In the current Reynolds stress model, a wall function approach is only used for the dissipation but not for the Reynolds stresses. The Reynolds stresses are linearly extrapolated from values in control volumes interior to the flow (Clarke and Wilkes, 1989).

**Outlet**

At the outlet all quantities were given a zero normal gradient except for velocities, which were given a constant gradient. The pressure is extrapolated from upstream values. This is equivalent to assume that the flow field is fully developed at the outlet. Initially the velocity is first given a zero normal gradient. The difference from actual mass flow rate is then calculated. Then a constant multiplied by the outward going normal is added to the velocities in order to ensure exact global mass conservation.

**GEOMETRY AND GRID GENERATION**

The basis for the 3D geometry construction for this test case was the original paper drawing of the model draft tube. This paper drawing was converted to a 3D solid model using the CAD software I-DEAS. This task caused some trouble because the real draft tube was not exactly constructed by the measures in the paper drawing. After consulting Urban Andersson at Vattenfall Utveckling, Sweden (who also conducted the boundary condition measurements for this workshop) the geometry was adjusted to be as close as possible to the real geometry. This geometry was then converted to several CAD formats. In this case the IGES format was used.

The solver used, CFX 4.2 (AEA Technology, 1997), is a block-structured code. The geometry must therefore be subdivided into several inter-connecting solids before creating the grid. The surfaces of the original model were not suited to create a mesh of high quality, a must if good convergence rate should be possible. The original model was therefore redrawn quite drastically in order to achieve a well-behaved grid.

The most critical task was to create a high quality mesh at the upper wall of the bend. The grid for this part of the geometry can be seen in figure 6. This grid is one way to achieve a high quality mesh. There are probably several other possible ways to reconstruct the geometry.
Fig. 6 The block topology at the upper wall.

If the original topology (figure 7) was used, almost zero-degree angle elements were generated at the sidewall of the draft tube.

Fig. 7 Result of grid generation when using the original surfaces.

The large errors in these badly shaped elements made it impossible to achieve a converged solution during the initial test calculations. This shows how important a good mesh is to the solution.

NUMERICAL ERRORS

The largest numerical error in Computational Fluid Dynamics is the discretization error or grid error as found out by (Bergström and Gebart, 1999). This error will be considered first. The second error, the iterative error, is usually a smaller problem than the discretization error.

Grid error

Richardson extrapolation was used to calculate the grid error as described in (Ferziger and Peric, 1996). This method expand the discretization error as the leading term in a Taylor series:

$$
\varepsilon_h = \phi_{exact} - \phi_{ah} = a_1 (\alpha h)^p + H
$$

where \( h \) is the grid cell size, \( \alpha \) is a grid refinement factor and \( a_1 \) is a coefficient that depends on the derivatives but do not depend on \( h \). Eq. (5) can be used for three grids in a sequence resulting in an expression for \( p \), the actual order (as opposed to the formal order) of the numerical scheme.

$$
\frac{\phi_{\alpha_1 h} - \phi_{\alpha_2 h}}{\phi_{\alpha_1 h} - \phi_{\alpha_2 h}} = \frac{\alpha_{1}^{p} - \alpha_{2}^{p}}{\alpha_{2}^{p} - \alpha_{1}^{p}}
$$

(6)

The three grids have to be in the asymptotic range for this analysis to work (\( h \) has to be small enough). \( p \) can then be used to calculate an approximation of the exact solution (\( \phi_{\text{extrapolated}} \)). \( \alpha \) is a grid refinement factor defined as

$$
\alpha = \left( \frac{N_1}{N_2} \right)^{1/3}
$$

(7)

where \( N_1 \) is the number of control volumes (or grid points) for the finest grid and \( N_2 \) for a coarser grid. Then the grid error can be estimated using:

$$
\varepsilon_{r,\text{approx}} = \frac{\phi_{\text{extrapolated}} - \phi_{h}}{\phi_{\text{extrapolated}}}
$$

(8)

A question remaining to be answered is what value \( \phi_h \) one should use to calculate the grid error. A natural suggestion in this case would be the pressure recovery factor that is one way to describe the efficiency of the draft tube. The pressure recovery factor is defined as:

$$
C_{pr} = \frac{P_{\text{out:wall}} - P_{\text{in:wall}}}{\frac{1}{2} \rho \left( \frac{Q}{A_w} \right)^2}
$$

(9)

where \( P_{\text{out:wall}} \) and \( P_{\text{in:wall}} \) are the mean wall pressures at the outlet and inlet. This definition is adopted because the pressure measurements were only conducted at the walls. The number of grid points, the grid refinement factor and the pressure recovery factor are presented in table 1.
Table 1. Grid sizes, grid refinement factors and pressure recovery factor. \( C_{pr} \) is based on wall pressure. \( C_{pr-s} \) is based on mean values at the inlet and outlet surfaces.

Using the values of table 1 (the \( C_{pr} \) based on wall pressures) in eq. (6) yielded a negative left-hand side. This suggests that the error decreases with alternating sign, i.e. we have oscillatory grid convergence (Celik and Karatekin, 1997). Celik and Karatekin suggest an alternative way of calculating the actual order \( p \) for this case:

\[
\frac{\phi_{\alpha_{h,b}} - \phi_{\alpha_{b,h}}}{\phi_{\alpha_{h,b}} - \phi_{\alpha_{b,h}}} = \alpha_1^2 + \alpha_2^2
\]

(10)

This equation resulted in \( p=5.668 \). This is of course not a reasonable value of \( p \) (which should be between 1 and 2 because both higher order upwind and hybrid differencing is used). This can depend on the way in which the grid is refined. The grid is not refined equally in all direction because of the necessity to have a fine grid close to the walls. If the Richardson extrapolation method should be meaningful, grids even finer than the finest used here would have to be generated. It is possible that the coarsest grid (361279 grid points) is still not fine enough.

However, it is possible that the two finest grids are in the asymptotic range and then it is still possible to estimate the grid convergence error. But then the actual order \( p \) can not be calculated. A natural suggestion would be to calculate the error using Richardson extrapolation using both \( p=1 \) and \( 2 \). In this way we will at least get an upper and lower limit for the grid convergence error (but one can not be absolutely sure that the grids really are in the asymptotic range). Richardson extrapolation can be expressed as (Bergström, 1997):

\[
\phi_{\text{extrapolated}} = \frac{\alpha_2^2 \phi_{h} - \phi_{\alpha_{b,h}}}{\alpha_2^2 - 1}
\]

(11)

Assuming \( p=2 \) yields \( \phi_{\text{extrapolated}}=0.84578 \) and \( e_{\text{approx}}=-3.06950\% \) and for \( p=1 \), \( \phi_{\text{extrapolated}}=0.81669 \) and \( e_{\text{approx}}=-6.74108\% \). To summarize: the error varies between 3-7% if \( p \) is assumed to be between 1 and 2.

A plot of \( C_{pr} \) for the three grids is shown in figure 8. The curves in this figure are obtained by solving for \( \eta \) in:

\[
\phi(\alpha) = \phi_{\text{extrapolated}} + \eta \alpha^p
\]

(12)

Fig. 8 Richardson extrapolation (\( p=1 \) and 2) applied to the result from three grids in a sequence. \( \alpha \) is the grid refinement. The box, circle and plus symbols are the results from grid 1, 2 and 3, respectively.

Eq. (7), (11) and (12) can also be used to estimate the necessary grid size for a grid error of e.g. 1%. This results in a necessary grid size of 3.9*10^6 grid points for \( p=2 \) and 222*10^6 for \( p=1 \). Unfortunately, due to available computer resources it is not possible to check if these really are the required grid sizes. However, we still do not know for sure that the grids are in the asymptotic range making it a bit hazardous to conclude anything about the grid convergence error.

**Iterative error**

The iterative convergence was controlled by the residuals of the equations. The residuals are coupled to the real iterative error. This means that a reduction in the residuals implies a reduction in the iterative error. If zero initial values are used as a starting guess then the initial error will be equal to the solution itself. E.g. if the residuals have fallen about 4 orders of magnitude, then the error should have fallen by a comparable amount, i.e. the error is about 0.01% of the solution, (Ferziger and Peric, 1996). (Ferziger and Peric, 1996) also suggest that the iterations can be stopped when the residuals for the inner iterations have decreased 1-2 orders of magnitude (Table 2) and 3-5 orders of magnitude for the outer iterations (Table 3). The convergence criterion for the outer iterations is fulfilled for all equation except for some of the Reynolds stresses. For the inner iterations the convergence criterion is fulfilled for all equation except for the pressure correction equation. However, it should be noted that table 2 only shows the average reduction in the residuals for all outer iterations.
u-velocity residuals $1.79 \times 10^1$

v-velocity residuals $1.72 \times 10^1$

w-velocity residuals $1.64 \times 10^1$

mass source residuals $5.88 \times 10^0$

$\varepsilon$ residuals $1.10 \times 10^1$

uu residuals $5.56 \times 10^1$

vv residuals $5.00 \times 10^1$

ww residuals $4.76 \times 10^1$

uv residuals $5.56 \times 10^1$

vw residuals $4.55 \times 10^1$

wu residuals $5.26 \times 10^1$

Table 2. Average reduction in residuals for the inner iterations (ratio of the initial residual and the final residual).

<table>
<thead>
<tr>
<th>Residual</th>
<th>Reduction</th>
<th>Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>u-velocity residuals</td>
<td>$1.4 \times 10^4$</td>
<td>0.007%</td>
</tr>
<tr>
<td>v-velocity residuals</td>
<td>$6.2 \times 10^3$</td>
<td>0.016%</td>
</tr>
<tr>
<td>w-velocity residuals</td>
<td>$6.4 \times 10^3$</td>
<td>0.016%</td>
</tr>
<tr>
<td>mass source residuals</td>
<td>$5.3 \times 10^4$</td>
<td>0.002%</td>
</tr>
<tr>
<td>$\varepsilon$ residuals</td>
<td>$3.0 \times 10^3$</td>
<td>0.033%</td>
</tr>
<tr>
<td>uu residuals</td>
<td>$4.9 \times 10^2$</td>
<td>0.204%</td>
</tr>
<tr>
<td>vv residuals</td>
<td>$4.4 \times 10^2$</td>
<td>0.227%</td>
</tr>
<tr>
<td>ww residuals</td>
<td>$1.2 \times 10^2$</td>
<td>0.833%</td>
</tr>
<tr>
<td>uv residuals</td>
<td>$1.3 \times 10^3$</td>
<td>0.077%</td>
</tr>
<tr>
<td>vw residuals</td>
<td>$4.2 \times 10^2$</td>
<td>0.238%</td>
</tr>
<tr>
<td>wu residuals</td>
<td>$5.8 \times 10^2$</td>
<td>0.172%</td>
</tr>
</tbody>
</table>

Table 3. Reduction in residuals of the outer iterations (ratio of residuals from second and last iterations) and error for each variable.

COMPARISON TO EXPERIMENTAL DATA

Pressure measurements

Pressure measurements were (after the workshop) available at the upper and lower centerline of the walls and at the walls at the inlet and outlet of the draft tube. Figures 9 and 10 shows the pressure for the three grids at upper and lower centerline and the measured pressure.

Fig. 9 Pressure at the upper centerline. The pressure has been normalized by dividing the pressure at all locations by the pressure at the first point (at Distance=0). Distance is the distance in meters from the inlet of the draft tube.

Fig. 10 Pressure at the lower centerline. The pressure has been normalized by dividing the pressure at all locations by the pressure at the first point (at Distance=0). Distance is the distance in meters from the inlet of the draft tube.

Figure 9 shows that grid 1 (725779 grid points) agrees quite well with experiment except for the pressure drop at Distance=0.5 which is over predicted and an area of too high pressure between Distance=1 to 2. Grid 1 shows the same
agreement for the lower centerline (fig. 10) but the pressure is too high between Distance=0.5 to 1.5.

Sorting the pressure recovery factor by increasing values results in the order: Grid 1, Grid 3 and Grid 2. The pressures at the upper and lower centerline behave in the same way. This again indicates that at least the solution from the coarsest grid is not in the asymptotic range (required for eq. 6 to work).

**Velocity measurements**

The velocity was measured at the inlet (the boundary conditions for the workshop) and at a cross section further downstream (named cross section III in the workshop) (figure 11).

![Fig. 11 Location of cross section III. A contour plot visualizes the velocity component perpendicular to the cross section.](image1)

In figure 12 the velocity component perpendicular to cross section III is plotted. The agreement is at least qualitatively close to the experiments. There is a low speed area to the left and lower left in both experiment and simulation and a high-speed area in the lower right corner.

![Fig. 12 Velocity perpendicular to cross section III. The upper part shows the CFD result from Grid 1. The gray scale is the same for both pictures.](image2)

Figure 13 shows the v-velocity component (in positive y-direction) in cross section III. For this component the agreement is not good. The experiments indicate that there is one large vortex rotating in the anti-clockwise direction but the simulation predicts two counter rotating vortices (cf. fig. 14).
Fig. 13 V-velocity (in positive y-direction to the right in the figure) at cross section III. The upper part shows the CFD result from Grid 1. The gray scale is the same for both pictures.

Table 4. Engineering quantities required for the Turbine 99 workshop.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Inlet</th>
<th>Cross section III</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha_{axial}$</td>
<td>1.0488</td>
<td>1.1763</td>
</tr>
<tr>
<td>$\alpha_{swirl}$</td>
<td>$6.1553 \times 10^{-2}$</td>
<td>$2.0140 \times 10^{-2}$</td>
</tr>
<tr>
<td>$\beta$</td>
<td>1.1026</td>
<td>1.0362</td>
</tr>
<tr>
<td>$S$</td>
<td>0.1816</td>
<td>$6.0363 \times 10^{-2}$</td>
</tr>
</tbody>
</table>

$\alpha_{axial}$ and $\alpha_{swirl}$ are kinetic energy corrections factors (Staubli and Deniz, 1994), $\beta$ is the momentum correction factor (Daugherty et. al, 1989) and S is the Swirl intensity (Seeno et. al, 1978).

The predicted pressure recovery factor was $C_{pr}=0.87174$ and the energy loss coefficient was $\zeta=0.1135$ (for the finest grid). The measured pressure recovery factor was $C_{pr}=1.122767$. The error is 22%; i.e. the simulated pressure recovery factor is 22% lower than the measured value.

**Flow field and pressure distribution**

The most striking result from the simulation is the large vortex that starts just below the runner cone and extends all the way down to the bottom wall (figure 15). This was expected, e.g. (Batchelor, 1967) shows analytically that in the case of a rotating flow inside a pipe having a inner boundary shaped as a cone, the azimuthal velocity becomes infinitely large downstream the apex of the cone.

Fig. 14 Secondary velocity in cross section III.

Engineering quantities
A number of engineering quantities were requested for the workshop. These are presented in table 4. All values are for grid 1.

Fig. 15 The vortex below the runner cone visualized by vortexlines. The vortexlines correspond to the vorticity vector as streamlines correspond to the velocity vector.
The vortex is also an area of recirculation as shown in figure 16. The velocity in the area below the cone is directed upwards towards the cone. This is because of the high rotational speed in the vortex inducing a low pressure just below the cone (figure 17). Flow visualizations with fluorescent dye showed qualitatively the same results.

![Fig. 16 The vortex visualized using two planes (x=0 and y=0) representing the velocity (m/s) in the z-direction (w). The w-velocity is positive in the vortex indicating an upward flow in this area.](image1)

![Fig. 17 The low pressure (Pa) in the area below the apex of the cone forcing the flow field upwards.](image2)

Separated regions were one of the phenomena of interest for the workshop. This simulation results in three regions close to the outlet where separation occurs, the largest one downstream the edge at the upper wall. Figure 18 shows these three regions. A closer view of the largest region is shown in figure 19.

![Fig. 18 Separated regions visualized by iso-surfaces of the x-components of the velocity (u_x=0).](image3)

![Fig. 19 The separated region at the upper wall (y=-0.33). The colors represent the pressure distribution and shows the adverse pressure gradient inducing the separation.](image4)

The separation zones are also visible at the outlet. Visualizing the flow field (figure 20) also shows that large velocities at only a small part of the outlet dominate the flow.
Fig. 20 The outlet flow field. The flow field is non-uniform with large velocities at the y>0 lower corner of the outlet. There are also low magnitude inward velocities in two regions indicating two separation zones. (Cf. figure 18).

There were no further major separation or recirculation zones found in the domain in addition to the ones already mentioned.

COMPUTER REQUIREMENTS

CFD simulations of this type undoubtedly require large computers. In this case a parallel version of the code was used on a 10 processor Silicon Graphics Onyx2 at High Performance Computing Center North (HPC2N) in Umeå, Sweden. CPU time, number of iterations and number of processors for all grids are reported in table 5.

<table>
<thead>
<tr>
<th>No. of grid points</th>
<th>No. of iter.</th>
<th>CPU time</th>
<th>processes</th>
</tr>
</thead>
<tbody>
<tr>
<td>725779</td>
<td>2000</td>
<td>9.5 hours</td>
<td>6</td>
</tr>
<tr>
<td>515752</td>
<td>2000</td>
<td>6.8 hours</td>
<td>6</td>
</tr>
<tr>
<td>361279</td>
<td>2000</td>
<td>6.2 hours</td>
<td>4</td>
</tr>
</tbody>
</table>

Table 5. Computational times for all grids.

DISCUSSION

Starting with the boundary conditions at the inlet it was necessary to make additional assumptions about the velocity variation in the boundary layer. However, the question is how important this really is. Even if the grids were large, the radial resolution of the grid was only 11 control volumes at the inlet (with smaller control volumes at the walls but not fine enough to completely resolve the boundary layer). This means that the variation of the velocity in the boundary layer in the simulation really is no better than the original measurements. The reason for using such a “small” number of control volumes at the inlet is that one have to decide were the grid should be refined (and to some extent, how the geometry is divided into subdomains). In this case it was decided that the grid should have as equal resolution as possible in all directions in order to be sure that all flow phenomena were captured. It is of course possible to guess in advance where interesting areas in the computational domain exist and create a finer grid there. However, it is possible that some unexpected phenomena arise.

The prediction of the pressure recovery factor is dependent on the pressure at the walls, which in turn is dependent on the prediction of the remaining variables close to the wall. One of the reasons for the poorly predicted pressure recovery factor can be that the wall boundary conditions for the Reynolds stresses are not defined by wall functions in the present code. Instead, the Reynolds stresses are linearly extrapolated from values in control volumes interior to the flow (Clarke and Wilkes, 1989). Using wall functions for the Reynolds stresses might improve the solution, e.g. like the work by (Perzon, 1997). A wall treatment similar to the work by (Perzon, 1997) will be included in a future project.

The creation of the geometry and grid was a major part of the work. The block-structured grid used required drastic reconstruction of the original CAD geometry. This was necessary in order to achieve a high quality grid (which is necessary to achieve good iterative convergence), especially at the upper wall where the cross section changes from being circular to rectangular. If the original CAD surfaces were used to create the subdomains, large errors in badly shaped elements at the upper wall made it impossible to achieve a converged solution during the initial test calculations. This shows how important a good mesh is to the solution.

The problem of achieving a high quality grid always appears when a block-structured grid is used in combination with a complicated geometry. There are of course some rules of thumb one can follow when creating the grid, but each time a new geometry is encountered completely new problems arise. It is possible that the use of an unstructured grid makes grid generation easier in this case. However, if a block-structured grid is the only choice, a way of improving the quality of the grid can be the use of more advanced grid generation software available (e.g. ICEM). This can help the user to create block-structured grids of higher quality, but in the end it is still up to the user to decide the topology of the grid. This can be a problem when comparing results from calculations where the same original geometry has been used, but different topologies. One at least can hope that grid refinement studies show that the topology is of secondary importance. However, this is yet to be proven.

The grid error was 3%-7% depending on the assumed order of the differencing schemes and the iterative error was approximately 0.8-0.002% depending on the variable. The errors were generally larger for the Reynolds stresses. This...
shows that the iterative error is smaller than the grid convergence error. A result also found out by (Bergström and Gebart, 1999). Following the recommendations by (Ferziger and Peric, 1996) the solution has not converged for some of the variables. This can indicate that trying to reach a steady state solution in this case is not possible. A transient simulation should be included in a future study of this case.

In this study the asymptotic range was not reached for the present grid sequence. The most obvious reason is that the grids are too coarse. It is also possible that only the coarsest grid is outside the asymptotic range. The only way to answer this question would be to create one or several larger meshes. However, it is also possible that the present grids are sufficient and that the poor iterative convergence (towards a steady state solution) for the Reynolds stresses is the reason for not reaching the asymptotic range. But as mentioned above, it might not be possible to reach a steady state solution in this case.

The comparison with pressure measurements at the upper and lower centerlines shows quite good agreement for the finest grid except for the pressure drop at the upper centerline and the area of maximum pressure at the lower centerline. The pressure drop at the upper centerline occurs at the end of the first conical section of the draft tube (this pressure drop is visible to the right in figure 17). The reason for the disagreement in this area is probably that the grid is not fine enough in this area to resolve this sharp drop. The area of high pressure at the lower centerline consists of two maxima, the first one also visible in figure 17 to the lower left and a second, more widespread area, at the lower back corner of the draft tube. Visualizations (Andersson and Karlsson, 1999a and 1999b) show that there is a quite large vortex in this corner. This vortex was not predicted in the simulation, explaining why the pressure is wrongly predicted in this area.

The pressure recovery factor was found to be 22% lower than the measured value. The reason for this (except for the reasons mentioned above) might be that the outer boundary layer at the inlet of the draft tube is not resolved enough. Applying the law of the wall and wall functions too far from the wall can result in a wrongly predicted pressure. The assumption of a radial velocity (fig. 2) which was not measured at the inlet can of course also influence the pressure distribution at the inlet.

The result from the simulation shows two main phenomena. The first is the vortex downstream the runner cone. This vortex is probably unsteady, with the core of the vortex rotating itself and parts of this vortex coming off, affecting the flow field all the way to the outlet. This makes it questionable if a steady state solution of this problem is possible to reach. However, a pure transient simulation of this case would require transient boundary conditions at the inlet and the computational time would probably be very long if high resolution in time is desirable. It is also possible to conduct a transient simulation of this case with the presently available boundary conditions. This can probably shed some light on why it is difficult to reach a steady state solution. However, one might ask if turbulence models based on RANS are suited for transient simulations. Information about the behavior of turbulence models (RANS) in unsteady flows can be found in (Leschziner, 1998). Generally one can say that turbulence models (RANS) can be used only if the unsteadiness is quasi-steady or slow but not if it is moderate or fast (Leschziner, 1998).

The second phenomenon is the large separation region at the upper wall. This region stretches out outside the outlet. Because an assumption of a fully developed flow at the outlet is assumed the region outside the draft tube should be included in the simulation. This however, will make the grid even larger and the equalization wall in the tank downstream the draft tube has to be modeled in some way. Also, the small fraction of incoming fluid into the outlet of the draft tube is probably not a large error source in the calculation.

ACKNOWLEDGMENTS

The financial support of the Swedish National Board for Industrial and Technical Development, Elforsk, and Kvaerner Turbine AB is gratefully acknowledged.

This research was conducted using the resources of High Performance Computing Center North (HPC2N).

REFERENCES


Bergström, J., (1997), Turbulence modelling and numerical accuracy for the simulation of the flow field in a curved
channel, 1997 ASME Fluids Engineering Division Summer Meeting, FEDSM’97, Vancouver, Canada, June 22-26.

Clarke D.S., and Wilkes N.S., (1989), The calculation of turbulent flows in complex geometries using a differential stress model, AERE-R 13428, Engineering Sciences Division, Harwell Laboratory, Oxfordshire OX11 0RA.


