DRAFT TUBE CALCULATIONS

Thomas Staubli
Fluid Mechanics and Hydromachines
HTA Luzern
CH 6048 Horw
Switzerland

Daniel Meyer

ABSTRACT
The flow of the workshop draft tube was calculated using the commercial CFD code CFX-TASCflow. As inlet condition of the model draft tube the measured velocity distributions were used with the exception of the radial velocity component. In order to check the Reynolds number effects additional calculations were performed on a modified mesh for the prototype geometry. A series of interacting vortical structures are observed in the calculated flow fields. These structures are visualized by their core streamline and selected streamlines around that core. The most interesting features observed are the breaking down of the cone vortex rope and the formation of a strong vortex in the three-dimensional recirculation zone in the lower draft tube corner.

INTRODUCTION
The physics of the draft tube flow poses a major challenge to numerical flow calculations, since it encompasses a series of flow phenomena difficult to model:

- the inlet flow is highly three-dimensional and unsteady in nature,
- the flow separates at the rotating hub,
- the diffuser flow may cause local or global recirculation zones,
- the bent gives rise to secondary flows and potentially to local recirculation zones,
- at the outlet there is a possibility of backflow into the draft tube,
- for swirling flows and local recirculation zones the chances for occurrence of unsteady flow phenomena are high.

Separations, or rather recirculation zones, observed in draft tube flows will be generally of three-dimensional nature, that is, different streamlines for the separation point and the reattachment point can be found (Hornung, 1985). Whereas the planar separation comprises only diffusive transport of vorticity, the three-dimensional recirculation is dominated by convective vorticity transport (Ginter, Staubli, 1999).

NOMENCLATURE
The coefficients used in this paper are defined as follows:

- Pressure recovery factor $C_{pr}$
  \[ C_{pr} = \frac{\bar{p}_{out \, wall} - \bar{p}_{in \, wall}}{\rho \left( \frac{Q}{A_{inlet}} \right)^2} \]

- Energy loss coefficient $\zeta$
  \[ \zeta = \frac{\dot{E}_{inlet} - \dot{E}_{outlet}}{E_{kin \, inlet}} \]
  \[ = \frac{\int \int \left[ \left( p + \frac{1}{2} u^2 \right) u \right] dA - \int \int \left[ \left( p + \frac{1}{2} u^2 \right) u \right] dA}{\int \int \left[ \frac{p}{2} u^2 \right] dA} \]

- Efficiency $\eta$
  \[ \eta = \frac{\bar{p}_{out \, wall} - \bar{p}_{in \, wall}}{\frac{\rho}{2} \left( \frac{Q}{A_{inlet}} \right)^2 - \left( \frac{Q}{A_{outlet}} \right)^2} \]

Note: instead of using pressures averaged along the walls $\bar{p}_{in, \, out \, wall}$ the pressure recovery factor $C_{pr}$ and the efficiency $\eta$ were also calculated using area averaged pressures $\bar{p}_{in, \, out}$.

Data are given in Tab. 1.
Kinetic energy correction factors $\alpha_{\text{axial}}$ and $\alpha_{\text{swirl}}$

$$\alpha_{\text{axial}} = \frac{\int \int u_{\text{axial}}^2 \, dA}{A \bar{u}_{\text{axial}}^3}$$

$$\alpha_{\text{tangential}} = \frac{\int \int (u_{\text{tangential}} u_{\text{axial}}) \, dA}{A \bar{u}_{\text{axial}}^3}$$

Momentum correction factor $\beta$

$$\beta = \frac{\int \int u^2 \, dA}{A \bar{u}_{\text{axial}}^3}$$

Swirl intensity $S$

$$S = \frac{\int \int (ru_{\text{axial}} u_{\text{tangential}}) \, dA}{\int \int u_{\text{axial}}^2 \, dA}$$

where $r$ = radial distance from the axis, respectively the centerline.

$u, \, dA$ = vectors

$\bar{u}_{\text{axial}} = \frac{Q}{A}$

SIMULATION

The flow of the workshop drafttube was calculated using the commercial CFD code, CFX-TASCflow, version 2.8. A quasi-unsteady prediction of the flow was carried out by solving two iterations per time step. The time step was set to 0.3 s.

On the hardware side we had at our disposition a sun workstation ultra 60 with a 360MHz CPU and 512 Mbyte RAM.

For grid generation we employed ICEM CFD powermesh, version 3.3. The block structured mesh with 32 blocks is displayed in Fig. 1. A number of 493 884 elements have been totally used for discretization of the computational domain. Grid refinement was done in the inlet section and close to the walls, Fig. 2. The non-dimensional distance to the wall $y^+$ was adjusted to remain within the range of 11 and 330. The log law was used for boundary layer approximation.

As inlet condition of the model drafttube the measured velocity distributions were used. The radial component was estimated in a way that a linear angle variation in radial direction could be achieved, providing a flow congruent to the hub wall. Such flow angle variations are typically observed in probe measurements at the turbine outlet flows. Inlet turbulence intensities were chosen to be in agreement with the measured velocity fluctuations, although these fluctuations arise from blade passage periodic flow and do not correspond to statistic turbulence.

Velocities close to the walls were extrapolated from the measured data on the basis of the log law approximation. Since the integration of the measured data did not meet exactly the given experimental discharge values of test points $T(r)$ and $R(r)$ the axial velocity vectors were proportionally stretched to yield the required values.

TASCflow parameters (CFX-TASCflow User Documentation, 1995) set for the simulation:

- Discretization control parameters:
  - ISKEW = 4
  - LPAC = true

- Solution control parameters:
  - kntlin = 2
  - dtime = 0.3 s

- Fluid properties:
  - $\text{viscfl} = 0.001 \, \text{[kg/(ms)]}$
  - $\text{rhofl} = 0.999 \, \text{[kg/m}^3\text{]}$
  - $\text{cvfld and cpfld} = 4182 \, \text{[Nm/(kg K)]}$

In the course of the calculations three major causes were found to give rise to convergence problems. The first was connected to the mesh generation. Elements at the block boundaries within the bent having very small angles were found to be responsible for convergence troubles. A second
cause came from the backflow at the outlet. This problem was overcome by using "opening" (allowing in- and outflow at a constant outlet pressure) as outlet boundary condition until good convergence was achieved before then changing back to "outflow" and parameter "i.o. walls = f" as boundary condition (allowing variable pressure at the outlet). Providing radial velocity components at the inlet also proved to be an unfavorable measure with respect to convergence and slowed down the convergence rate considerably.

GLOBAL DATA RESULTS

Tab. 1 gives the data for the coefficients defined in the nomenclature section. Since authors have doubts that the inlet pressure variation resulting from the given velocity distributions are physically correctly modeled they propose to compare also the area averaged pressures and the accordingly modified coefficients $C_{pr}$ and $\eta$.

Not included in this table are calculations which were performed with reduced inlet turbulence. When reducing this turbulence by a factor of 4 the pressure recovery and efficiency rose both by about 0.01 while the losses were reduced by the same amount.

<table>
<thead>
<tr>
<th>Tab.1 Global results</th>
<th>case T(r)</th>
<th>case R(r)</th>
<th>Prototype bep</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N$</td>
<td>595</td>
<td>595</td>
<td>$N/N_P$ = 1</td>
</tr>
<tr>
<td>$Q$</td>
<td>0.535</td>
<td>0.554</td>
<td>$Q/Q_P$ = 0.1875</td>
</tr>
<tr>
<td>$H$</td>
<td>4.5</td>
<td>4.5</td>
<td>$H/H_P$ = 0.0036</td>
</tr>
<tr>
<td>Using wall pressures:</td>
<td>$C_{pr}$</td>
<td>0.697</td>
<td>0.689 $C_{pr}/C_{prP}$ = 0.94</td>
</tr>
<tr>
<td></td>
<td>$\eta$</td>
<td>0.711</td>
<td>0.702 $\eta/\eta_P$ = 0.94</td>
</tr>
<tr>
<td>Using area averaged pressures:</td>
<td>$C_{pr}$</td>
<td>0.561</td>
<td>0.522 $C_{pr}/C_{prP}$ = 0.89</td>
</tr>
<tr>
<td></td>
<td>$\eta$</td>
<td>0.571</td>
<td>0.531 $\eta/\eta_P$ = 0.89</td>
</tr>
<tr>
<td></td>
<td>$\zeta$</td>
<td>0.438</td>
<td>0.474 $\zeta/\zeta_P$ = 1.36</td>
</tr>
<tr>
<td>cross section Ia</td>
<td>$\alpha_{axial}$</td>
<td>1.032</td>
<td>1.030 $\alpha_{axial}/\alpha_{axialP}$ = 1.02</td>
</tr>
<tr>
<td></td>
<td>$\alpha_{swirl}$</td>
<td>0.074</td>
<td>0.043 $\alpha_{swirl}/\alpha_{swirlP}$ = 1.07</td>
</tr>
<tr>
<td></td>
<td>$\beta$</td>
<td>1.106</td>
<td>1.074 $\beta/\beta_P$ = 1.03</td>
</tr>
<tr>
<td></td>
<td>$S$</td>
<td>0.222</td>
<td>0.163 $S/S_P$ = 0.98</td>
</tr>
<tr>
<td>cross section III</td>
<td>$\alpha_{axial}$</td>
<td>1.812</td>
<td>1.936 $\alpha_{axial}/\alpha_{axialP}$ = 0.85</td>
</tr>
<tr>
<td></td>
<td>$\alpha_{swirl}$</td>
<td>0.200</td>
<td>0.258 $\alpha_{swirl}/\alpha_{swirlP}$ = 1.21</td>
</tr>
<tr>
<td></td>
<td>$\beta$</td>
<td>1.391</td>
<td>1.466 $\beta/\beta_P$ = 0.95</td>
</tr>
<tr>
<td></td>
<td>$S$</td>
<td>0.274</td>
<td>0.303 $S/S_P$ = 1.18</td>
</tr>
</tbody>
</table>

MODEL - PROTOTYPE

In order to check the Reynolds number effects additional calculations were performed on a modified mesh for the prototype geometry which was enlarged by a factor 11. The overall quantity of nodes was equal in both simulations and amounts to 493,884 nodes. In order to model the relatively smaller boundary layers of the prototype flow the outer points had to be located closer to the walls for the prototype.

The global quantities varied considerably (see Tab.1) and the results reflect that viscosity effects become smaller for higher Reynolds numbers. The velocity distribution at the outlet is less uniform for the prototype case, but, in contrast, the swirl component $\alpha_{swirl}$ and the swirl intensity $S$ are less pronounced for the prototype.

At first glance it is astonishing that the prototype ratios at cross section Ia are not equal to one. But since the inlet boundary layers of model and prototype were approximated slightly different (extrapolated points close to the walls were introduced for the prototype) the deviation becomes explainable.

FLOW FIELDS

A general overview on the draft tube flow downstream from the bent is given in Fig. 3. The velocity vectors show the velocity components within the planes and the colors give the velocity vector magnitudes (blue = small velocities, green = high velocities). There is no indication for a separation or recirculation along the critical upper wall in the center plane.

Fig.3 Overview on the draft tube flow: case T(r)
Fig. 4 Visualization of the tangential velocity distributions in planes along the centerline (looking downstream, shaded areas indicate upstream flow): case T(r)
The calculated draft tube flow fields are clearly dominated by the interaction of counter rotating vortices. Fig. 4 displays an overview of the flow development in various planes along the draft tube centerline for the case T(r). The cutting planes show the tangential velocity components when looking downstream. The color coding represents the magnitude of the velocity vector, that is the speed.

It was decided to show only images of flow visualization for the case T(r) (top point on the propeller curve = bep) in this paper because the general flow effects look very similar also in the case R(r) (point on the right leg of the same propeller curve = overload). The same basic features of vortex formation is observed in case of both inlet data.

In the portion of the draft tube downstream of the cone the vortex flow is dominated by the inlet swirl component, which is rotating in clockwise direction when looking downstream. The resulting cone vortex core can be clearly followed until the beginning of the bent where it is breaking down (Fig. 4, plane 1 to 6 and Fig.5). By viscosity this vortex is feeding energy to a counter rotating vortex forming on the right side in the planes 4, 5, and 6 of Fig.4. This secondary vortex is developing into one the two dominating bent vortices which are observed until the draft tube outlet section.

Fig.5 displays the vortex rope starting downstream of the cone separation. The vortex is visualized by its core and a selected number of surrounding streamlines. The color coding of the streamlines shows the time the fluid remains in the draft tube and reflects the fact that the axial velocities become very small close to the vortex core.

A second very important vortex formation starts within the separation region of the lower corner of the draft tube (Fig.6). In this recirculation zone the vortex is fed by viscous forces with energy, is then detaching and forming the clockwise rotating vortex observed in all downstream sections. The color coding of the tangential velocities in the cutting planes corresponds again to the magnitude of the velocity vector (blue = small velocities, green = high velocities).

Fig.7 shows the two vortex cores which dominate the flow field in the diffuser section of the draft tube downstream of the bent. The one on the left side rotating clockwise is the vortex of Fig.6. The counter clockwise rotating vortex observed on the right side of Fig.7 results from the secondary vortex formed in the planes 4 to 6 of Fig.4 and is additionally fed by energy by the bent flow.
Fig. 6 Vortex formation in the lower draft tube corner: case T(r)

Fig. 7 Counter rotating vortices propagating downstream from the bent: case T(r)
Pressure Fields

As mentioned before, the authors have their doubts with respect to the correctness of the prediction of the pressure distributions at the inlet cross section Ia. The lowest pressures are found at the walls, which results of course in very good values for the pressure recovery factors and efficiency. However, this pressure distribution with positive gradients towards the center as depicted in Fig.8 is in contradiction with pressure distributions typically found in swirling flows. In addition, also the streamwise curvature of the streamlines may not be made responsible for such strong radial pressure gradient leading to the conclusion that inlet pressures are not correctly predicted.

![Fig.8 Development of the static pressure distributions in the inlet section of the draft tube: case T(r)](image)

The computed static pressure distribution in the outlet cross section IV varies only slightly, confirmed by the experimental wall pressures data which also differ not more than ±100 Pa from a mean value. Comparing the local wall pressures reveals however that the distributions have their maxima and minima at different locations.

![Fig.9 Distribution of the static pressure in the outlet section of the draft tube: case T(r)](image)

CONCLUSIONS

Of course, the final conclusions can only be written after the detailed comparison with the experimental data, but the following preliminary conclusion can be drawn beforehand: namely, that the secondary flow in this draft tube test case is driven by several interacting vortical structures. Since the employed k-ε-solver is not especially adapted to model such flows the quantitative prediction may be expected to be moderate. Most interesting features observed are the breaking down of the cone vortex rope and the formation of a strong vortex (rotating clockwise as the inlet swirl) in the three-dimensional recirculation zone in the lower draft tube corner.

ACKNOWLEDGMENTS

The authors wish to express their appreciation to Dr. Peter Holbein for his invaluable advice when convergence problems procured them moments of desperation.

REFERENCES