COMPUTATION OF UNSTEADY 3D FLOW AND TORQUE TRANSMISSION IN HYDRODYNAMIC COUPLINGS

A. Kost, N. K. Mitra, and M. Fiebig
Institut für Thermo- und Fluidodynamik
Ruhr-Universität Bochum
Bochum, Germany

ABSTRACT
For the optimized design of hydrodynamic or fluid couplings an adequate understanding of the flow field in such devices is necessary. In a hydrodynamic coupling, torque is transmitted by fluid circulation due to a speed differential between the rotating pump impeller and a matching turbine runner. Detailed studies of the unsteady 3D flow and torque transmission in fluid couplings are reported here. A finite-volume method with non-staggered variable arrangement is used to solve the unsteady Navier-Stokes equations on boundary-fitted grids and for a rotating frame of reference. The results give further insights into the physical process of torque transmission and allow important conclusions for the design of hydrodynamic couplings.

INTRODUCTION
Hydrodynamic couplings consist of a pump impeller with radial blades and a similar matching turbine runner, both within the same casing but generally with different numbers of blades (cf. Fig. 1). The casing in general is partially filled with an oil of low viscosity. The pump impeller is driven by a prime mover, such as an electrical motor or an internal combustion engine. Torque is transmitted from the impeller to the turbine by fluid circulation due to different angular velocities (slip) of the coupling halves. If there is no slip, there is no flow and consequently there will be no torque transmission. In a fluid coupling, the input and output torque must be the same, because there is no other element between the pump impeller and the turbine runner to provide a torque reaction.

FIG. 1: CROSS-SECTIONAL VIEW OF A FLUID COUPLING

Hence, the efficiency of such couplings is simply given by the ratio of output to input speeds (Langlois, 1979). Besides the friction the impact of the fluid and the rotors is the cause of losses. The impact loss results due to the non-alignment of the fluid and the rotor blades.

The circulating flow in reality is two-phase (oil/air), three-dimensional, unsteady (possibly periodic) and contains separated zones. The complexity of the flow field depends on the design of the working circuit (e.g. effective diameter of the coupling,
number and shape of the blades, axial gap between the impeller and the turbine) and dynamical parameters (oil filling, angular velocity of the impeller and speed differential). Depending on the dynamical parameters the flow may be laminar or turbulent.

Experimental data is extremely expensive to obtain due to the difficulties of measuring flow quantities in rotating narrow passages. A computational analysis is a suitable tool to analyze these flows and to optimize the design of the working circuit.

In the present study, the simulation of laminar 3D periodic flow in completely filled fluid couplings with equally pitched impellers and different shape of cross-section is reported.

**BASIC EQUATIONS**

The equations governing the unsteady 3D single-phase flow of an incompressible viscous fluid in a hydrodynamic coupling are written in non-dimensional form for a co-ordinate system rotating with constant angular velocity \( \omega_p \) (Greenspan, 1968):

\[
\nabla \cdot \mathbf{u} = 0
\]

\[
\frac{\partial \mathbf{w}}{\partial t} + \mathbf{R}_0 \cdot \nabla \mathbf{w} + 2\mathbf{k} \times \mathbf{w} = -\nabla P - E_k \nabla \times (\nabla \times \mathbf{w})
\]

(2)

Here, \( \mathbf{w} \) is the velocity vector related to the rotating frame of reference (with \( \mathbf{k} \) as the direction of positive rotation) and non-dimensionalized by the velocity scale \( U = \omega_p D \). The centrifugal acceleration and the influence of additional body forces \( \mathbf{f} \) are combined with the static pressure \( p \) to form the reduced pressure \( P \):

\[
\nabla P = \nabla p - f - k \times (k \times \mathbf{r})
\]

(3)

The dimensionless parameters appearing in eq. (2) are the Ekman number \( E_k \) and Rossby number \( R_o \) providing ratios of viscous force to Coriolis force and convective acceleration to Coriolis force, respectively:

\[
E_k = \frac{\mu}{\rho \omega_p D^2} \quad R_o = \frac{U}{\omega_p D} = s
\]

(4)

Here, \( s \) is the slip between the pump and the turbine defined as:

\[
s = 1 - \frac{\omega_T}{\omega_p}
\]

(5)

**FIG. 2:** SKETCH OF THE COMPUTATIONAL DOMAIN (A) AND RELATIVE MOVEMENT OF THE TURBINE BLADE THROUGH THE COMPUTATIONAL DOMAIN (B) (CIRCUMFERENTIAL CROSS-SECTION, \( \tau = \text{CONST.} \)) ARROW: DIRECTION OF THE RELATIVE VELOCITY OF THE TURBINE \( u_{rel,T} \)

The analysis of the flow field is limited to one pitch by assuming an equal number of blades for the pump impeller and the turbine runner and prescribing periodic boundary conditions at the circumferential surfaces of the computational domain (cf. Fig. 2a). Since the geometric form of the blades is simple, it is possible to use one grid block for calculation. The blades of the turbine runner are treated as internal obstacles moving relatively to the computational domain (cf. Fig 2b). The timestep is
determined through the grid resolution in the circumferential
direction. This procedure is suitable to avoid problems arising
from matching the flow quantities using patched or overlaid grids
that move relatively to each other (Rai, 1989). But it is not
suitable for fluid machines with complex geometric form, e.g.
fluid torque converter.

With respect to the coordinate system rotating with angular
velocity \( \omega_p \), the boundary conditions at the solid surfaces of the
pump and the turbine are:

\[
\omega = \begin{cases} 
0 & \text{pump impeller} \\
-(k \times \omega) & \text{turbine runner}
\end{cases}
\]  

(6)

METHOD OF SOLUTION

Grid generation

The computational grid is generated by different procedures.
For rectangular cross-sections of the fluid coupling (cf. Fig. 3a)
a simple algebraic method is employed. A differential equation
method has been used for the generation of more complex grids
for fluid couplings with circular cross-sections (cf. Fig. 3b).

\[
\text{FIG. 3: COMPUTATIONAL GRIDS FOR ONE HALF OF}
\text{THE FLUID COUPLING; A) RECTANGULAR CROSS-}
\text{SECTION, B) CIRCULAR CROSS-SECTION}
\]

The source terms of the Poisson-type partial differential
equations used in this method are determined iteratively so that
certain prescribed boundary conditions can be satisfied, e.g. grid
intersection angles or a specified distribution of grid lines at the
boundaries (Hilgenstock, 1988).

Discretization procedure

Using Cartesian velocity components the basic equations are
 discretized by employing a finite-volume scheme (Kost et al.,
1992). The flow domain is subdivided into a finite number of
control volumes (CV).

\[
\text{FIG. 4: GRID ARRANGEMENT AND NOMENCLATURE}
\]

All dependent variables are defined in the centerpoint P of the
CV (cf. Fig. 4 for grid arrangement and nomenclature). Integration
of the momentum equations (2) for each CV leads to a balance
equation of momentum fluxes through the CV faces and
volumetric sources. The diffusive part of the momentum fluxes
can be obtained by assuming linear variation of the variables
between adjacent grid points. Evaluation of the convection fluxes
requires discretization schemes for interpolating the variable
values at the CV faces from their nodal values.

In the present code, the convective fluxes are split into an
implicit part which is obtained by first-order upwind differencing,
and an explicit part containing the difference between the
second-order accurate central differencing scheme and the
upwind approximation. This technique originally suggested by
Khosla and Rubin (1979) is known to enhance the stability of the
iterative solution algorithm.

The resulting finite volume equation for variable \( \phi \) can then be
written in general form (Perić, 1985):

\[
\frac{\partial \phi}{\partial t} + \sum a_{ij} \frac{\partial \phi_j}{\partial x_i} + b_i = \frac{1}{\varphi} \left( \sum a_{ij} \phi_j + \frac{1 - \varphi}{\varphi} a_i \phi \right)
\]  

(7)
where the coefficients $a_{ab}$ represent the combined convection and diffusion effects and $b_{o}^{p}$ contains the discretized source terms (i.e. transient term, pressure gradient and Coriolis term) and explicitly treated parts of the convection and diffusion fluxes.

Since the equations are non-linear and strongly coupled by the convective and Coriolis term, for the convergence of the iterative solution procedure under-relaxation of variable changes $\omega_{p}$ with a factor $0 < \omega_{p} \leq 1$ is necessary in order to enhance the diagonal dominance of the coefficient matrix.

**Pressure-velocity coupling**

For incompressible flows, the convergence of the numerical method for the solution of the momentum and continuity equation depends strongly on an adequate handling of the pressure-velocity coupling (Patankar, 1980). The velocity field obtained by the solution of the momentum equations using a guessed pressure field in general does not satisfy the continuity equation. The continuity equation serves as an additional constraint for the velocity field to adjust the pressure gradient in the momentum equations.

In the present method, the coupling between pressure and velocities is achieved by the well-known SIMPLEC algorithm suggested by van Doormaal and Raithby (1984). In order to avoid an oscillatory pressure field due to the non-staggered variable arrangement a special interpolation has been used to determine the mass fluxes through CV faces from the adjacent CV-centered quantities (Rhie and Chow, 1983).

Substitution of the mass fluxes in the discretized form of the continuity equation will result in a mass imbalance $S_{m}$:

$$S_{m} = F_{m}^{+} - F_{m}^{-} + F_{m}^{*} - F_{m}^{*} - F_{m}^{*} = S_{m}$$

(8)

Flux corrections $F = F^{*}$ are needed to annihilate this imbalance. These corrections are based on corresponding velocity corrections. According to the SIMPLEC algorithm the velocity corrections at the CV faces are related to pressure corrections at the nodal points. Thus, substitution of the mass flux corrections leads to a pressure-correction equation of the final form:

$$a_{p}P_{p} = \sum a_{ab}P_{ab} - S_{m}$$

(9)

In the present method, the systems of algebraic equations (7) and (9) are solved by the SIP algorithm of Stone (1968), which is based on an incomplete LU decomposition.

**RESULTS AND DISCUSSION**

The calculations have been performed for different geometrical parameters (shape of the cross-section, number of blades $Z$, ratio of diameters $\zeta_{p} = d/D$, aspect ratio $\zeta_{g} = B/D$) and dynamical parameters ($E_k$, $R_{o}$) of the fluid coupling.

Fig. 5 displays the mean (time-averaged) velocity field relative to the rotating co-ordinate system in a meridional plane between two blades of the pump impeller. The circulating flow between the coupling halves, which is responsible for the torque transmission from the pump impeller to the turbine runner, is clearly visible. Small secondary eddies have formed in the corners of the geometry.

The corresponding flow field in an axial ("blade-to-blade") cross-section of the pump impeller is shown in Fig. 6. The velocity vectors are displayed relatively to the motion of the pump impeller, i.e. relative to angular velocity $\omega_{p}$, so that the blades of the impeller are at rest. The main flow in the pump impeller is directed radially outwards (cf. Fig. 5). Due to the rotation of the impeller there are low pressure and high velocity on suction side and high pressure and small velocity on pressure side of the blades. Near the shroud a passage vortex has formed due to the combined viscous and Coriolis effects.

Because of the cyclic passage of the turbine blades through the computational domain, the flow field in hydrodynamic couplings

$$\rightarrow S \omega_{p} D$$

**FIG. 5: CIRCULATING FLOW BETWEEN THE PUMP IMPELLER AND TURBINE RUNNER IN A MERIDIONAL PLANE OF A FLUID COUPLING WITH RECTANGULAR CROSS-SECTION ($\zeta_{p} = 0.326$, $\zeta_{g} = 0.281$, $Z = 24$, $E_k = 1.5 \cdot 10^{-5}$, $R_{o} = 0.05$)
with equal pitch of the impellers is periodic in time. Fig. 7 shows the time history of the reduced pressure at two specific points in the pump impeller and in the gap between the coupling halves.

The passing turbine blade results in a tremendous pressure variation in the gap between the coupling halves. With increasing distance from the gap, the wake is compensated so that the pressure fluctuations in the pump impeller are much smaller.

The torque transmission coefficient \( \lambda(t) \) of a fluid coupling is defined as

\[
\lambda(t) = \frac{M(t)}{\rho \omega_p^2 D^5}
\]  

and consists of a convective part due to the change of the moment of momentum experienced by the fluid between entering and leaving the impeller and a second part due to the local time variation of the velocity field in the impeller:

\[
\lambda(t) = \lambda_{\text{con}}(t) + \lambda_{\text{un}}(t)
\]

The convective part \( \lambda_{\text{con}}(t) \) can be obtained by an integration of the moment of momentum over the face \( A \) of the impeller:

\[
\lambda_{\text{con}}(t) = \int_A \left( x c_y - y c_x \right) c_z \, dA
\]

Here, \( c_x \) and \( c_y \) denote the Cartesian \( x \) and \( y \) component of the absolute velocity vector and \( c_z \) is the axial flow component.

The second part \( \lambda_{\text{un}}(t) \) only exists for unsteady flow and can be obtained by an integration of the moment of momentum over the whole volume \( V \) of the impeller:

\[
\lambda_{\text{un}}(t) = \frac{\partial U(t)}{\partial t} \quad \text{with} \quad U(t) = \int_V \left( x c_y - y c_x \right) \, dV
\]

Fig. 8 shows the time variation of the torque transmission coefficient \( \lambda(t) \) and the influence of the different parts on the total change of the moment of momentum. The influence of the unsteady part on the torque transmission is negligible for very small values of slip (cf. Fig. 8a). For higher slip between the coupling halves, the unsteadiness of the flow field has a distinct influence on the torque transmission. Notice that there is a phase shifting of the time-periodic torque transmission, i.e. with
FIG. 8: TIME-PERIODIC CHANGE OF MOMENT OF MOMENTUM IN A FLUID COUPLING. THE TOTAL CHANGE CONSISTS OF A CONVECTIVE PART AND AN UNSTEADY PART DUE TO THE LOCAL CHANGE OF THE VELOCITY FIELD.

Increasing slip the maximum and minimum of the transmitted torque are shifted to earlier times within each period due to the unsteady flow effects.

Due to the time-periodic flow the influence of the unsteady part on the change of the moment of momentum within one period $\tau$ vanishes:

$$\frac{1}{\tau} \int_{0}^{\tau} \lambda_{t}(t) dt = 0 \quad (14)$$

<table>
<thead>
<tr>
<th>$\text{Ro} = s$</th>
<th>0.05</th>
<th>0.40</th>
<th>1.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>circ.</td>
<td>1.318</td>
<td>6.865</td>
<td>9.164</td>
</tr>
<tr>
<td>rect.</td>
<td>1.100</td>
<td>5.135</td>
<td>6.017</td>
</tr>
</tbody>
</table>

**TABLE 1: TORQUE TRANSMISSION $\lambda_{m} \cdot 10^{3}$ OF FLUID COUPLINGS WITH DIFFERENT SHAPE OF CROSS-SECTION**

Fig. 9: CALCULATED OPERATING CHARACTERISTICS OF DIFFERENT FLUID COUPLINGS WITH RECTANGULAR CROSS-SECTION AND Z = 24 BLADES OF PUMP IMPELLER AND TURBINE RUNNER ($E_{k} = 1.5 \cdot 10^{-5}$).
So, the mean value of the torque transmission $\lambda_m$ can be obtained by an integration of the convective part $\lambda_{con}(t)$ over the time period $t$:

$$\lambda_m = \frac{1}{t} \int_0^t \lambda_{con}(t) \, dt \quad (15)$$

Fig. 9 shows some of the calculated operating characteristics of different fluid couplings with rectangular cross-sections. With increasing $r_d=d/D$ the torque transmission capacity of hydrodynamic couplings decreases. An increasing of the aspect ratio $r_B=B/D$ results in lower torque transmission at start-up point ($\omega_T=0$) and higher $\lambda_m$ at nominal operating point ($\omega_T/\omega_P=0.95$). This behaviour is important for light-load start-up and efficient operation of the coupling.

The influence of the meridional shape of the fluid coupling is shown in Table 1. Due to lower streamline curvature hydrodynamic couplings with circular cross-sections have higher torque transmission capacities compared to the same coupling with a rectangular meridional shape.

**CONCLUSIONS**

A finite-volume method with non-staggered variable arrangement has been developed and used to solve the Navier-Stokes equations for a rotating frame of reference on boundary fitted grids. Results for the time periodic 3D flow in fluid couplings show that the flow structure is very complex and strongly influenced by combined viscous and Coriolis effects. The operating characteristic of different fluid couplings has been obtained by an integration of the moment of momentum. The time-periodic torque transmission has been shown to be influenced by unsteady flow effects. Important conclusions for the design of the working circuit have been given.

**ACKNOWLEDGEMENTS**

This work was sponsored by the Deutsche Forschungsgemeinschaft (DFG) through special grant SFB 278 "HydrodynamischeLeistungsübertragung". The authors gratefully acknowledge this support.

**REFERENCES**


