Analysis of Fluid Flow and Heat Transfer in Industrial Fluid Couplings

Zixiang SUN*, John Chew*, Neil Fomison** and David Edwards**

*Fluids Research Centre, School of Engineering, University of Surrey, Guildford,
Surrey GU2 7XH, UK

**Fluid Systems, Trent Hall 1 (SINA-73), Rolls-Royce plc, PO Box 31, Derby DE24
8BJ, UK

Abstract

Computational fluid dynamics (CFD) solutions are presented for unsteady flow and heat transfer in model fluid couplings. Factors investigated include the effects of coupling size, cooling throughflow, vane numbers and angled vanes. Predictions of torque characteristics are consistent with previously published experimental data and an elementary analysis. In this initial study, only single phase solutions are presented, although these results do confirm that cavitation and/or air entrapment can be significant in practice. Angling of the vanes at 20° to the axial direction is found to give a large increase in torque at low slip running conditions. However, pressure variations within the coupling are also increased and so the angled vane geometry will be more susceptible to cavitation.

Key Words: Fluid coupling, CFD analysis, Heat transfer, Torque characteristics

1. INTRODUCTION

Fluid couplings are used in engineering applications due to their unique features in flexible transmission of shaft torque between a pair of driving and driven shafts. The principle of a fluid coupling and a model wheel geometry proposed for an industrial application are shown in Figures 1-a and 1-b, respectively. A pair of wheels rotate about the coupling axis at different speeds, with a small axial gap between them. The chamber is filled with low viscosity oil, and a circulation is established in the coupling circuit due to the differential rotation. This leads to exchange of angular momentum between the two wheels. The driving wheel works as a pump, imparting angular momentum to the working fluid, while the driven wheel works as a turbine, receiving the angular momentum from the fluid. Therefore, a flexible transfer of shaft torque is realized from the driving pump impeller to the driven turbine runner through the working fluid without mechanical contact.

Research work on fluid couplings is reported in the open literature as early as the 1930's. As described by Sinclair [1, 2], the early emphasis was on experimental investigations. Later in the 1960's and 70's, a number of studies were published including the development of elementary 1D and 2D forced vortex flow models for prediction of fluid coupling performance, see, for example, Qualman and Egbert [3], Wallace et al. [4] and Whitfield et al. [5]. Experimental studies and design developments have continued up to the present day. The flow inside the couplings is complex, especially under partially-filled or cavitating conditions. Different flow regimes occur depending on the conditions and so prediction of coupling performance is difficult.

Several useful experimental studies are reported in Japanese journals and Germany theses. In 1967, Ishihara et al. [6] give results for the effects of geometrical

parameters on the coupling performance. Flow visualization is reported in 1980 by Uchiyama et al. [7], highlighting the importance of speed ratio and filling fraction on flow and performance. The torque-speed characteristics have been shown to be subject to instability and to be sensitive to small changes in conditions by Uchiyama et al. [8], Takenaga et al. [9] and Matsui et al. [10]. These conclusions are confirmed for various coupling geometries and operating conditions by other experimental studies, such as those by Middelmann [11], Wiennholt [12] and Kurokawa et al. [13].

A literature survey has revealed few publications on numerical investigations of the internal flow in fluid couplings. Almost all those that are available are from the Ruhr University research group and its research partners. A laminar flow simulation was reported in 1994 by Kost et al. [14], and Bai et al. [15] reported a turbulent flow simulation. The flows were assumed to be single-phase and incompressible in a one-pitch domain for a model fluid coupling with a rectangular cross-section geometry. Turbulent flow modelling was reported to give better predictions, and most of the flow simulations thereafter assumed turbulent flow using the k - ϵ model (Formanski et al. [16], Huitenga et al. [17] and Bai et al. [18]). Strong unsteadiness and the forced vortex flow were correctly reproduced by the CFD calculations. CFD solutions were also used in improving start-up behaviour of a fluid coupling by Huitenga and Mitra [19] and in exploring torque output enhancement by employing angled vanes by Bai et al. [20].

The present investigation is aimed at understanding various factors, which affect performance of industrial fluid couplings. The investigated factors are scale effects, speed ratio between the driving and driven wheels, cooling throughflow, number of vanes and angled vanes. The numerical investigation employs the k - ϵ model of turbulence and the sliding plane technique to model unsteady flow assuming single-

phase flow. Two extreme filling conditions, fully-filled (oil-filled) and completely empty (air-filled), were examined. The CFD data obtained are compared with available measurements from publications. As some engineering applications may require the coupling to run empty at high rotation speed, heat transfer was also investigated to get some insight into the possible cooling requirements. As far as authors are aware, this is the first publication addressing effects of coupling size, coolant throughflow and heat transfer.

2. MODEL DESCRIPTION

2.1 The Fluid Coupling

The industrial fluid couplings under investigation have a circular cross section to minimise the wetted area, as shown in Figure 1. The two wheels are geometrically identical and arranged in a symmetrical way. The inner and outer radii of the fluid coupling are depicted as R_i and R_o , respectively. As a result, the hydraulic diameter of the coupling circuit d can be expressed as $d = R_o - R_i$, and the mean pitch diameter of the coupling D is equal to $D = R_o + R_i$.

Two couplings are investigated in the present study. Except where stated otherwise, the vanes employed in both couplings are thin radial plates. Angled vanes were investigated in some calculations. Geometrical similarity approximately holds between the two couplings, with the large coupling being around 1.4 times bigger in size than the small one, with the diameter ratio of d/D being maintained around $d/D=0.38$ for both the small and large couplings. The small axial gap h between the two wheels was set to be 0.54% and 0.77% of the mean pitch diameters for the large and small couplings, respectively.

The speed ratio between the two wheels is defined as $SR = \omega_t / \omega_p$, where ω_p and ω_t denote the angular speeds of pump impeller and turbine runner, respectively. Although the technical term slip ratio $1 - SR$ is also commonly used in the engineering literature, the speed ratio SR is generally used in this paper for convenience. Obviously, the speed ratio SR should normally be between 0 and 1, with $SR = 0$ describing the stall condition, where the turbine runner is stationary. At this condition, the induced circulation and shaft torque generated are expected to reach their maxima. $SR = 1$ designates a condition of synchronous rotation, i.e. $\omega_t = \omega_p$, where “solid-body” rotation of the fluid is expected and consequently both the induced circulation and shaft torque generated reduce to zero.

The speed ratio investigated in the present CFD investigation covers a range from $SR = 0$ to $SR = 0.8$. The speed of wheel rotation investigated was up to 2000 rpm (33.3 Hz) for the fully-filled state (oil-filled) and 30,000 rpm (500 Hz) for the empty state (air-filled), respectively. The corresponding rotational Reynolds number $Re = D_o^2 \omega_p / \nu = D^2 (1 + d/D)^2 \omega_p / \nu$ was in a range between 0.25×10^6 and 4.40×10^6 , where D_o stands for the outer diameter of the coupling and ν denotes kinematic viscosity of the working fluid, oil or air, at wall temperature and operating pressure conditions. Accordingly, the flow inside the couplings was assumed fully-turbulent in all the calculations.

2.2 1D Model Analysis

For a quick estimation of the coupling performance, an elementary one-dimensional vortex flow model similar to those proposed by Qualman and Egbert and Wallace et al. was developed in the present investigation. In the 1D model, the flow in the coupling is assumed incompressible and axisymmetric, and a uniform

distribution of the induced velocity across the radius of the coupling circuit is considered, as shown in Figure 2-a. V_{ind} stands for the induced velocity. R_m represents the neutral radial position, or the centre of the induced circulating flow. By applying mass conservation for the induced circulating flow, R_m can be determined as follows.

$$\dot{m} = \rho V_{ind} \pi (R_o^2 - R_m^2) = \rho V_{ind} \pi (R_m^2 - R_i^2) \quad (1)$$

where $\dot{m}$ = induced mass flow rate, ρ = density of the working fluid

Hence

$$R_m = \sqrt{\frac{R_i^2 + R_o^2}{2}} \quad (2)$$

R_1 and R_2 are the inner and outer radii of the mean streamline of the induced flow, which divides the circulating mass flow into two equal halves in the coupling circuit. Accordingly, R_1 and R_2 yield

$$R_1 = \sqrt{\frac{R_i^2 + R_m^2}{2}} = \sqrt{\frac{3R_i^2 + R_o^2}{4}} \quad (3)$$

$$R_2 = \sqrt{\frac{R_m^2 + R_o^2}{2}} = \sqrt{\frac{R_i^2 + 3R_o^2}{4}} \quad (4)$$

When the flow in each half of the coupling is assumed to reach the “solid body” rotation (no slip) and to follow the straight vane surface at exits of both pump impeller and turbine runner, as might be expected when there is a large number of vanes, see Figure 2-b for illustration with subscripts p and t denoting pump and turbine, respectively, the shaft torque M will be given by

$$M = \dot{m} (\omega_p R_2^2 - \omega_t R_1^2) = \frac{\pi}{8} \rho V_{ind} (R_o^2 - R_i^2) \{ \omega_p (R_i^2 + 3R_o^2) - \omega_t (3R_i^2 + R_o^2) \} \quad (5)$$

The induced velocity V_{ind} may be estimated in the following manner. Assuming a loss coefficient K for each wheel, the mean pressure rise in the pump impeller ΔP_p and the mean pressure drop in the turbine runner ΔP_t are approximately equal to,

$$\Delta P_p = P_2 - P_1 \approx \frac{1}{2} \rho \omega_p^2 (R_2^2 - R_1^2) - K \frac{1}{2} \rho V_{ind}^2 \quad (6)$$

$$\Delta P_t = P_2 - P_1 \approx \frac{1}{2} \rho \omega_t^2 (R_2^2 - R_1^2) + K \frac{1}{2} \rho V_{ind}^2 \quad (7)$$

where P_1 and P_2 are the pressure at the mean streamline radii R_1 and R_2 , respectively.

As a result, the induced velocity can be obtained by equating the mean pressure rise in the pump impeller ΔP_p and the mean pressure drop in the turbine runner ΔP_t .

$$V_{ind} = \sqrt{\frac{1}{2K} (\omega_p^2 - \omega_t^2) (R_2^2 - R_1^2)} = \sqrt{\frac{1}{4K} (\omega_p^2 - \omega_t^2) (R_o^2 - R_i^2)} \quad (8)$$

From the coupling geometry, the inner and outer radii of the coupling R_i and R_o can be easily related to the mean pitch diameter D of the coupling and the minor diameter d of the cross section of the coupling circuit.

$$R_o = \frac{D + d}{2} \quad (9)$$

$$R_i = \frac{D - d}{2} \quad (10)$$

Therefore, the shaft torque can be rewritten in the following form

$$M = \frac{\pi}{16\sqrt{K}} \rho \omega_p^2 D^5 \sqrt{1 - \left(\frac{\omega_t}{\omega_p}\right)^2} \left(\frac{d}{D}\right)^{1.5} \left\{ \left(1 + \left(\frac{d}{D}\right)^2\right) \left(1 - \frac{\omega_t}{\omega_p}\right) + \frac{d}{D} \left(1 + \frac{\omega_t}{\omega_p}\right) \right\} \quad (11)$$

Normally, the dimensionless torque coefficient C_m is used, which is given by

$$C_m = \frac{M}{\rho \omega_p^2 D^5} = \frac{\pi}{16\sqrt{K}} \sqrt{1 - SR^2} \left(\frac{d}{D}\right)^{1.5} \left\{ \left(1 + \left(\frac{d}{D}\right)^2\right)(1 - SR) + \frac{d}{D}(1 + SR) \right\} \quad (12)$$

Where $SR = \omega_t/\omega_p$, the speed ratio between the pump impeller and turbine runner.

By assuming a loss coefficient K for a known coupling geometry and operating conditions, the coupling torque output can be obtained as a function of the speed ratio SR . A comparison of the 1D torque predictions with CFD results and experimental data will be discussed in the next section.

2.3 CFD Meshing

To reduce the computational requirements, all the calculations were performed on a one pitch domain with assumed pitchwise periodicity and 20 vanes for both wheels.

A numerical investigation on mesh dependence was then conducted to determine the appropriate size of mesh for the present CFD study using the small fluid coupling model. A typical mesh for the small fluid coupling model and the time mean shaft torque thus obtained with four meshes are shown in Figures 3-a and 3-b, respectively. The axis of rotation is the x-coordinate, as depicted in Figure 3-a. The cell numbers of the four meshes used in the investigation are given in Table-1. The meshes for the pump and turbine wheels are identical except their axial orientations. The sliding plane is located in the middle of the narrow axial gap between the two wheels to facilitate the unsteady simulations. The stall condition $SR=0$ with a rotational Reynolds number $Re = 0.682 \times 10^6$ was chosen for the investigation. It can be seen from Figure 3-b that a mesh independent solution is achieved for the later two meshes. Therefore, mesh-3 with 115,904 cells was considered adequate and thus employed for the small fluid coupling CFD model in the present study.

For the large fluid coupling, the number of mesh cells was increased in the present investigation to 648,968 in proportion to the physical size of the fluid coupling. By doing so, the mesh resolution of the large coupling model is comparable to the small one.

All the meshes were generated in GAMBIT by first defining a 2D mesh on a cross sectional plane of the turbine runner, and then extruding this circumferentially. The mesh for the pump part was obtained from a mirror copy of its turbine counterpart. The meshes generated are of hexahedral type with a uniform angular spacing in the circumferential direction. The observed mean non-dimensional near-wall distance (y^+) was typically between 34 and 84 in the calculations for both the CFD models, although a few local y^+ values were out of the range for a proper implementation of the wall function. Meshes for the angled vane calculations were obtained by deformation of the radial vane meshes, as will be described later.

Previous work on turbulent flow simulation using the standard k- ϵ model for a fluid coupling by Bai et al. provides further evidence to support the conclusion obtained in the present mesh dependence investigation. In Bai et al.'s work, the rotational Reynolds number of their test case is $Re=6.7 \times 10^6$, which is higher than the present study. They investigated three meshes with cell numbers of 24,288, 54,880 and 76,440, respectively, and concluded a mesh independent CFD solution was achieved with the later two meshes. Furthermore, these authors claimed that the CFD results obtained using the second mesh with 54,880 cells, were in good agreement with experimental data. Compared with the CFD model used in Bai et al.'s work, the mesh resolution employed in the present study is finer, and the rotational Reynolds

number is smaller. This supports the adequacy of the mesh resolution used in the present investigation.

2.4 CFD Modelling

As stated above, fully turbulent flow was assumed. The turbulence was modelled using the k- ϵ model with the standard wall functions. The use of any turbulence model does, of course, introduce some uncertainty. The choice of the k- ϵ model for the present study was based on its widespread use in many rotating and non-rotating engineering flows, and the lack of any other clearly established model for this application, where intensive flow separations take place. This approach follows that of other workers. As noted above, Bai et al. reported success with the k- ϵ model for fluid couplings. Furthermore, good numerical results were obtained with the k- ϵ model recently by Huitenga and Mitra for a fluid coupling and by Flack and Brun [21] for a torque converter pump.

The commercial CFD software FLUENT [22] was employed in the present investigation. The time-accurate interaction between the two wheels was resolved using the sliding plane technique. Consequently, the flow simulation is defined as three-dimensional, turbulent and unsteady. Circumferential periodic boundary conditions were applied for the one pitch domain. The time step was set so that the pump wheel moved one circumferential mesh space forward relative to the turbine wheel per time step in the unsteady calculations. Thus the time step is equal to the time for one blade passage of the pump to pass a turbine blade divided by the number of mesh spacings in the pitchwise direction. There were 48 and 80 mesh spacings per passage for the small and large fluid couplings, respectively. Hence, for a difference

of rotational speed between pump and turbine wheels being equal to 1000 rpm, for example, the time step is 6.25×10^{-5} and 3.75×10^{-5} seconds for the small and large couplings, respectively. The pitchwise spacing was determined to guarantee the non-dimensional near-wall distance (y^+) on the blade surface to be within the range for a proper implementation of wall function. As stated in the previous paragraph, the observed near-wall distance (y^+) was typically between 34 and 84. All walls were assumed to be no-slip except the axial gaps between the pump and turbine, where slip conditions were applied. No inflow and outflow were imposed for the coupling under investigation unless otherwise stated. For heat transfer calculations, all wall temperatures were assumed constant and uniform.

All the CFD solutions were obtained using an implicit pressure correction algorithm in the absolute frame. The spatial and temporal discretizations were 2nd order upwind scheme (UWS) and 2nd order implicit, respectively, for all the simulations except for a few test cases at empty filling conditions where the Mach number of the air flow is close to transonic or higher and calculations suffer from instability with the second order solver. In those cases, a first order solver was used instead.

Working fluid properties were set to model either oil (fully-filled) or air (empty state). The flow was regarded as incompressible at the fully-filled condition (oil-filled) and at low Mach number conditions when air was the working fluid. Otherwise, the ideal gas law was used for the air flow cases. Heat transfer calculations were conducted for the air flows.

3. RESULTS

The first three subsections below present CFD results for shaft torque characteristics, flow dynamics inside the coupling and heat transfer. The results are based on the CFD models described above for the enclosed fluid couplings with radial plate vanes. The effects of cooling throughflow, number of vanes and angled vanes are discussed in three further subsections. These include description of modifications made to the baseline CFD models.

3.1 Shaft Torque Characteristics

A typical time history of turbine shaft torque is shown in Figure 4. This CFD result was obtained for the oil-filled small coupling at the stall condition $SR=0$, with a Reynolds number $Re=0.682 \times 10^6$. Periodic variation of the shaft torque at the blade passing frequency is evident. The shaft torque reaches its minimum and maximum as the pump blade sweeps across the tip of the turbine blade. The magnitude of fluctuation over the mean is about $\pm 19\%$ for this test case. This large fluctuation of shaft torque is mainly due to a strong interaction of the flows between pump and turbine wheels at the stall condition, $SR=0$. The assumed equal number of vanes for both pump and turbine impellers also contributes to the large fluctuation of shaft torque. Further examination shows that all the flow properties, velocity and pressure as well as the circulation mass flow in the passage exhibit similar fluctuations as well.

The periodic fluctuations are enhanced by flow separations occurring in the coupling passage. A snapshot of relative velocity vectors on an inner and outer radial surfaces for this test case at $SR=0$ is shown in Figure 5. At this instant the pump blade is in the middle of the turbine passage. The flow separations on the turbine and pump blade suction surfaces can be easily identified. The CFD simulations also indicate that the strength of flow separations varies as the pump blade sweeps across the turbine

passage, as do the shaft torque and other flow properties. As the speed ratio SR increases, the periodic fluctuation weakens. As this unsteady effect is not a topic in the present study, no further analysis was pursued. However, the fundamental flow features are discussed in the flow dynamics section.

Mean shaft torques were obtained from time averaging over the periodic cycles. The shaft torque coefficients obtained for both the small and large couplings are shown in Figure 6. The circles and triangles denote the CFD data for the small coupling run at oil-filled and air-filled conditions, respectively; whereas the squares give results for the large coupling. The solid curve is the fitted torque characteristic based on this CFD data. It can be seen that the two couplings gave a similar normalised torque characteristic. Therefore the effect of coupling size can be considered as small and a similarity law approximately holds, as far as the geometry scale under investigation is concerned. The effect of Reynolds number on the normalised torque characteristics is also small.

The CFD results obtained were also compared with experimental data from various publications, including the rig test data by Kurokawa et al. and Middlemann. For clarity, only the data from Kurokawa et al are presented in Figure 6. It can be seen that the CFD data obtained are comparable to the experimental data, although the geometrical shapes of the couplings and test conditions of these experiments are different from the present ones. The cross section geometry of the experimental fluid coupling employed by Kurokawa et al. is of “pear” shape, whereas the one by Middlemann is rectangular. It is also interesting to see that the torque characteristic of Kurokawa et al.’s coupling exhibits a steep drop at a medium speed ratio of about $SR=0.55$. It was claimed by these authors that this is related to occurrence of air

bubbles in the coupling, as well as a transition into a large scale flow separation, as observed in the experiment.

Results from the elementary, one-dimensional vortex flow model described in Section 2.2 are also included in Figure 6, assuming the loss coefficient $K=0.5$, 1.0 or 1.5. It appears that at high speed ratio the appropriate loss coefficient K for the 1D model is approximately equal to 0.5, whereas at low speed ratio close to the stall condition, the loss coefficient K approaches roughly to 1.0. This could be associated with increased flow separation and hence losses at lower speed ratios.

3.2 Flow Dynamics

An instantaneous velocity vector plot in an angular plane is shown in Figure 7. The CFD result is obtained for the oil-filled large coupling at a high speed ratio $SR=0.8$ with a Reynolds number $Re=0.635 \times 10^6$. The induced vortex can be clearly identified. The vortex structure is further illustrated in Figure 8, which shows the time mean axial velocity and tangential velocity profiles in the mid-axial plane (this corresponds to the sliding plane of the CFD mesh). The almost linear variation of the axial velocity demonstrates that the mean induced circulation is close to a forced vortex. The mean tangential velocity at each wheel's exit is close to the rotation speed of each wheel. When the speed ratio SR decreases, the CFD simulations show that the strength of the forced vortex increases, and greater deviation of mean tangential velocity component occurs as the flow leaves the turbine and pump wheels. It can also be seen from Figure 8, that the induced circulation velocity is comparable in magnitude to the wheel rotation speed at this speed ratio. Part of the mean tangential velocity ratio profile at pump exit in the outer radius of the coupling circuit is slightly

bigger than the wheel “solid-body” rotation. This is attributed to secondary tangential flow in the pump rotation direction in the mid-axial plane.

Due to the strong induced circulation, the positive radial pressure gradient due to the wheel rotation is partly suppressed and consequently the mean pressure profile at the mid axial plane exhibits a “V” shape, with a minimum at the induced vortex centre. This is shown by the time mean pressure profile in Figure 9. Hence for the oil-filled state, there is a risk of air capture in the core region of the coupling, as air bubbles appear due to cavitation or come into the coupling circuit with the cooling throughflow. The CFD simulations show low instantaneous pressures and these could trigger cavitation. A circumferential variation of pressure at the axial gap including wheel’ inlets and outlets is indicated in Figure 9. These pressures are obtained for the angular position of the rotating coupling shown in Figure 7. Cavitation could occur in the coupling if the operating pressure is not high enough for the lowest absolute instantaneous pressures to be greater than the vapour saturation pressure. Higher fluid temperatures would decrease the saturation pressure and consequently make the situation worse. Air bubbles might also enter with the cooling inflow or be sucked from the outlet of the cooling throughflow due to a reverse flow, as discussed later. When the speed ratio SR decreases, the “V” shape of the pressure profile becomes stronger and minimum instantaneous pressures decrease further. Thus air capture and cavitation are more likely at lower speed ratio and running at higher speed.

As discussed in section 3.1, the CFD solutions also reveal extensive flow separations in the coupling. These are mostly on the vane suction sides and in the core region of the coupling, where local instantaneous pressures are low. The flow separations on the vane suction sides at wheels’ inlets are considered as a result of large flow incidence. The flow incidence β is defined as

$$\beta_{p/t} = \text{atan}((r\omega_{p/t} - V_\theta)/V_m) \quad (13)$$

Where $V_m = \sqrt{V_a^2 + V_r^2}$ denotes the meridional velocity. V_a , V_r and V_θ designate the axial, radial and tangential velocity components, respectively, and the subscripts p and t denote pump and turbine wheels, respectively. Figure 10 gives CFD predictions for time mean flow incidence for oil-filled coupling at SR=0.8. A circumferential variation of flow incidence for the turbine obtained at the angular position in Figure 7 is also shown. It can be seen that the mean flow incidences are, β_p around 17 deg at the inner radius (pump inlet), and β_t around -35 deg at the outer radius (turbine inlet). At outlets of the pump and turbine, the mean flow incidences are small, around 2 deg for the turbine outlet at inner radius and -5 deg for the pump outlet at outer radius for this test case at high speed ratio SR=0.8. It appears that the time mean flow is more constrained at the inner radii due to the smaller pitch distance there. Splitter vanes might be employed to further constrain the flow at outer radii. The flow incidence obviously varies considerably in the pitchwise direction, as indicated by the results for the turbine runner in the figure.

Further insight into the flow is given by the instantaneous axial flow contours on the sliding plane in Figure 11. The flow is highly non-uniform. High speed flow regions (the jets) are located on the vane pressure sides near the vane fillet corners in this angular position for the test case. Low flow velocity wake and reverse flows at exits are located on the vane suction sides near the vane fillet corners. Both the strength and the locations of the jets and wakes are subject to periodic oscillation as the two wheels sweep against each other. When the speed ratio SR decreases, the CFD simulations indicate that the high speed flow jets move towards wheel casings and the jet-wake structures become stronger consequently, the flow incidence and the flow separations also strengthen.

3.3 Heat Transfer

The power loss due to the work done on the fluid inevitably heats the fluid in the coupling. To get some insight into the heating effect, and accordingly the cooling requirements, a heat transfer investigation was conducted. For the proposed application under investigation, it was recognised that the heating effect could be far more significant when running empty with air at very high speed. Hence, only the empty condition running with air was considered in the investigation. The heat generated should be equal to the difference of power between pump input and turbine output, and reaches its maximum at the stall condition. Therefore, the range of low to medium rotation speed ratio SR was selected in the investigation. Table 2 gives a summary of the operating parameters for test cases studied. The working fluid air was treated as an ideal gas, with its viscosity and thermal properties specified as a function of temperature. All walls were assumed to be at a uniform temperature. The unsteady CFD solutions were run until they were periodic and the overall energy and momentum balances for the coupling were satisfied to less than 1% representative values.

Figure 12 shows the time mean bulk air temperature rise over wall temperature obtained in the pump impeller part for both small and large couplings from the stall condition SR=0 to a medium speed ratio SR=0.67. The temperature rise obtained in the turbine runner part from the CFD solutions is very close to that for the pump part. The mean bulk air temperature rise over wall temperature ΔT is a time average value of a temperature difference between the relative total bulk temperature and wall temperature for either pump or turbine part of a coupling:

$$\Delta T = T_{\text{air_relative_total}} - T_{\text{wall}} \quad (14)$$

The rotational Mach number is defined at the pump outer radius:

$$Ma = R_o \omega_p / a \quad (15)$$

where a is speed of sound at the operating condition. It can be seen from Figure 12 that ratio of the temperature rise over $\frac{1}{2}(\omega_p R_o)^2 / C_p$ varies between 4 and 7 over a range of the rotational Mach number up to 0.72 for the test case investigated. Although the CFD data is not sufficient to establish a general correlation, this initial heat investigation gives an indication of fluid temperature levels, dependence on the rotational Mach number, and the possible cooling requirements. For the higher Mach numbers the temperature obtained in the CFD solutions is typically over 100K higher than metal temperature. It is to be expected that the temperature rise will be dependent on speed ratio SR and Reynolds number as well as Mach number, and this has not been fully investigated in the present study.

A noticeable temperature rise will inevitably affect on the fluid coupling performance. The output drop in torque from its corresponding incompressible flow solution is included in Figure 12. It can be seen that the torque drop is significant at high rotational Mach number. For low Mach number flow, the torque drop is small or negligible, as expected.

The torque drop is clearly related to the temperature distribution in the coupling. A snapshot of an instantaneous static air temperature is shown in Figure 13. The solution is obtained for the large coupling at a speed ratio SR=0.56, rotational Mach number Ma=0.52 and Reynolds number Re=3.43x10⁶. It can be seen that the high temperature air impinges on the blade pressure surfaces at their inlets. These hot spots result in a lower density and a lower pressure locally at these regions, and consequently a lower torque output as the CFD results indicate.

To give a picture of heat transfer features in the coupling, the corresponding instantaneous Nusselt number contours on the vane pressure surfaces for the test case are shown in Figure 14. The Nusselt number Nu is defined as

$$Nu = - q d / \Delta T \kappa \quad (16)$$

Where q and $\Delta T = T_{\text{air_relative_total}} - T_{\text{wall}}$ are the local wall heat flux and air temperature rise, respectively, and κ denotes the thermal conductivity of air flow at wall temperature. It can be seen that the maximum heat transfer occurs at the inlets on the vane tips and pressure surfaces for both pump and turbine wheels.

The mean Nusselt number for the pump and turbine halves of the couplings can be obtained by time averaging over pitch-periodic cycles in terms of area weighted average heat flux q and bulk temperature rise ΔT . The results are shown in Figure 15. A correlation is also generated based on the CFD data in the form of $Nu = a \cdot Re_d^b$ for the whole coupling, which yields

$$Nu = 1.0 Re_d^{0.57} \quad (17)$$

Although the CFD data is not sufficient to make an accurate correlation, the obtained power index 0.57 for the correlation might be useful in some engineering calculations for convective heat transfer.

Note that the passage Reynolds number is used in the Nusselt correlation, as this is directly related to the mean induced circulating velocity that drives the heat transfer. The passage Reynolds number is defined as

$$Re_d = \rho V_{\text{ind}} d / \mu \quad (18)$$

where V_{ind} is the mean induced circulating velocity as shown in Figure 2, which can be obtained from Equation (1) in terms of the circulating mass rate as follows.

$$V_{ind} = 2 \dot{m} / \rho \pi (R_o^2 - R_i^2) = 2 \dot{m} / \rho \pi D d \quad (19)$$

Note also that in general the Nusselt number may also depend on speed ratio SR as well as the rotational Reynolds number.

3.4 Effect of Cooling Throughflow

A cooling approach considered in the proposed application is that the cooling flow is introduced at the inner radius from the gap between the two wheels and is discharged at the outer radius, as shown in Figure 16. A corresponding CFD model was thus built by extending the baseline one-pitch CFD model for the large coupling to include inlet and outlet flow paths. The mesh around the axial gap was locally refined to accommodate the flow gradients caused by the oil throughflow, and the final mesh had 762,568 hexahedral cells.

Two oil throughflow rates were investigated, $\dot{m}_q=0$ and $\dot{m}_q / \pi \rho h D^2 \omega_p = 0.006$, where h is the width of the axial gap. The simulations were conducted for oil flow at the stall condition $SR=0$, with a rotational Reynolds number $Re=1.36 \times 10^6$. In the simulations, the velocity of the oil inflow relative to the pump at the inlet was assumed to be uniform and parallel to the axial direction. At the outlet of the oil throughflow, the gauge pressure was assumed to be zero. However, initial calculations showed severe reverse flow at the outlet. A blockage with 50% of the outlet area was therefore added for the $\dot{m}_q / \pi \rho h D^2 \omega_p = 0.006$ test case. For $\dot{m}_q=0$ case, the outlet was given a non-penetrating, zero shear boundary condition. All other boundary conditions were as for the baseline CFD model.

Compared to the induced circulation the mean throughflow is very weak and no discernible effect on the torque output was found in the CFD calculations. Mean velocity profiles on the mid-axial plane are shown in Figure 17. Note that values of the non-dimensional radius less than 0 or greater than 1 are beyond the inner and outer radii of the coupling chamber, and represent the extended inner and outer flow paths for the cooling oil flow. It can be seen that the induced circulation is very strong at the stall condition, obviously stronger than the wheel rotation. The mean radial velocity is close to zero on this surface as expected. To get a picture of spatial distribution of the radial velocity component under the effect of cooling flow, the instantaneous radial velocity on this surface at an angular position is also given in the figure. It can be seen that radial velocity does exhibit some degree of scatter. However, a short distance into the extended inlet and exit flow paths, the flow becomes uniform circumferentially. These results show that the main circulating flow in the coupling was not significantly dependent on the cooling flow. As a result, the effect of the cooling flow on the torque output would also be insignificant for the conditions considered.

3.5 Effect of Number of Vanes

To show the influence of the number of vanes used, simulations were conducted for the large coupling with 10, 20 & 40 vanes. The coupling with 20 vanes is the baseline test case as described above. The domains and meshes for the couplings with 10 and 40 vanes are obtained by doubling or halving the baseline coupling domain. The flow conditions for all three test cases are identical with oil flow, a speed ratio $SR=0.8$, and a Reynolds number $Re=0.635 \times 10^6$.

A graphical representation of the torques obtained for the three test cases is shown in Figure 18. The pitch-to-diameter ratio $\pi D/dZ$ is used as the abscissa, recognising as discussed by Takenaga et al. that this is a more representative geometrical parameter than the number of vanes. It can be seen that the baseline test case with 20 vanes gives the highest torque output, followed by the 40 vane case. The mean flow velocity profile on the mid axial plane for the 10 vane case shows the weakest flow circulation as well as a severe flow distortion at the outer radius due to large scale separation. The lower torque output for the 40 vanes case may be due to a decrease in the coupling circuit volume resulted from the doubled vane volume and the enlarged surface area. The volume fraction of the vane blockage in the 40 vane case is 15%, compared with 7.5% for the 20 vane case. A check on the mean flow profiles on the mid axial plane also reveals a lower flow circulation for the 40 vane case. Therefore, the 20 vane option is likely to be an appropriate choice for the coupling under investigation.

3.6 Angled Vanes

An illustration of the angled vane and corresponding velocity triangle is shown in Figure 19. A vane angle of 20° from the axial direction was chosen to investigate the potential enhancement of torque output. For simplicity, the identical vane angle was considered for the pump impeller and, as before both wheels have 20 vanes. The CFD mesh was obtained by simply projecting its baseline radial vane counterpart to the current angled vane option. As the 20 deg tilt is not too far from the radial vane, the obtained mesh should be of similar quality. The boundary conditions employed are identical to the baseline CFD model. The calculated conditions considered are

again oil-filled at a Reynolds number $Re=0.635 \times 10^6$, but with three speed ratios $SR=0.8, 0.5$ and 0.25 .

A comparison of the CFD data obtained from the 20° angle vanes with published experimental data is shown in Figure 20. The CFD data from the baseline radial vane calculations is also plotted in the figure. It can be seen that the torque output is almost doubled by using the 20° vane at high speed ratio $SR=0.8$. This CFD result is also comparable to Wiennholt's experiment at high speed ratio, although the geometry and vane angle of Wiennholt's coupling are quite different. In Wiennholt's experiment, the vane angle was 15 degrees, and the coupling section was of rectangular type with d/D being approximately equal to 0.5. These, especially the cavitation effects could be responsible for deviation of the present CFD result from Wiennholt's measurements at low speed ratios. With a bigger vane angle in the present coupling model, flow separations could be more severe at low speed ratio region

The CFD solutions indicate that the enhancement of torque output is due to a significant increase in the difference of tangential velocity between the wheel exit and inlet, or the vane load. The strength of the induced circulation increases moderately. The mean absolute pressures at the centre of the induced vortex are much lower than its radial vane counterpart. When moving towards the low speed ratio, i.e., higher slip, the torque increases rapidly, and the minimum pressures drops significantly. Checking the CFD solutions at the assumed operating pressure level, it was found that negative absolute pressures occur in the coupling even at the high speed ratio $SR=0.8$. As a result, cavitation will occur.

A crude correction was proposed to take the cavitation effect into account in estimating the torque output from the single-phase flow solution. In this proposed

approach, the minimum absolute pressure used in calculating the moment was limited to the saturation value, i.e.

Absolute pressure $P_{\text{abs}} = P_s$ when $P_{\text{abs}} < P_s$ in estimating the shaft torque

As the saturated vapour pressure of gas turbine oil at temperature of interest is very low (eg typically less than 25Pa at 200°C) and P_s was approximated as zero. The corrected shaft torque for the 20° vane is included in Figure 20. This crude correction indicates that the impact of cavitation on the torque characteristics is highly significant and is highly likely to strongly limit the potential for increased torque generation.

4. CONCLUSIONS

It has been shown that CFD is a useful tool in understanding and predicting the behaviour of fluid couplings. Some important factors affecting the industrial coupling performance, including fluid coupling size, cooling throughflow, number of vanes and angled vanes were investigated using a k- ϵ turbulence model and unsteady solution techniques. The normalised torque characteristics obtained were comparable to the available measurements from publications. The effects of coupling size and cooling throughflow are found small, but use of angled vanes can give a significant increase in the torque output at high speed ratio. However, air capture and cavitation are likely in the coupling due to the induced vortex flow. Recognising the importance of two phase flow effects, an air-liquid two phase flow study has also been conducted and this will be reported in a separate paper.

The heat transfer calculations with a coupling running empty show that the bulk air temperature rise over wall temperature can be substantial for high Mach number

flow, which could result in a noticeable drop in the torque output. Other flow features, like flow separations due to large flow incidence and jet-wake flow structure, hot fluid impinging on vanes as well as mean flow profiles and low pressure distributions have been extracted.

Further topics could be explored in future investigations in addition to the proposed two-phase flow study. These could include sensitivity to turbulence modelling, which could be significant in these highly separated flows, and improved design to reduce flow separation and hence losses in the device. Effects of unsteadiness and secondary flows on the coupling efficiency could be further explored, together with other factors, such as geometry variations, different combinations of vane numbers between pump and turbine wheels, etc. All these would contribute to a full understanding of torque performance and flow physics of fluid coupling, and consequently a better design for engineering applications.

ACKNOWLEDGEMENTS

This study was sponsored by Rolls-Royce plc., UK. Assistance from Fluent Europe and all colleagues at TFSUTC at Surrey University is gratefully acknowledged.

Nomenclature

a	Speed of Sound
C_m	Torque coefficient = $M/\rho\omega_p^2 D^5$
C_p	Specific heat at constant pressure
d	Diameter (hydraulic diameter) of the coupling cross section = $(D_o - D_i)/2$
D	Mean pitch diameter of the fluid coupling = $(D_o + D_i)/2$
K	Loss coefficient or correlation factor
M	Shaft torque for full coupling
Ma	Rotational Mach number = $R_o\omega_p/a$
$\dot{m}$	Induced mass flow rate
$\dot{m}_q$	Cooling oil throughflow rate
Nu	Nusselt number = $-q d / \kappa\Delta T$
P	Pressure
q	Heat flux
r	Radius
R_1, R_2	Inner and outer radii of the mean streamline of the induced flow in the 1D model in Figure 2

Re	Rotational Reynolds number = $\rho D_o^2 \omega_p / \mu = \rho D^2 (1+d/D)^2 \omega_p / \mu$
Re _d	Passage Reynolds number = $\rho d V_{ind} / \mu$
R _m	Mean radius at the centre of the induced vortex in the 1D model in Figure 2
R _i , R _o	Inner and outer radii of fluid coupling
SR	Speed ratio = ω_t / ω_p
T	Temperature
t	Time
U	Periphery velocity
V	Absolute velocity
V _a	Axial velocity
V _{ind}	Induced velocity
V _r	Radial velocity
V _θ	Absolute tangential velocity
W	Relative velocity
Z	Number of vanes

Greek

α	Flow angle measured from the axial direction
β	Flow incidence
γ	Vane angle inclined from the axial direction

ΔP	Pressure rise or drop
ΔT	Bulk temperature rise or difference for each wheel = $T_{\text{rel_tot}} - T_w$
Δt	Time step
μ	Dynamic viscosity
ν	Kinetic viscosity $\nu = \mu / \rho$
ρ	Density of working fluid
κ	Thermal conductivity
ω	Angular speed of rotation

Subscripts

1,2	Inner and outer radii of mean streamline in 1D model in Figure 2
avg	Average
i	Inner radius of the fluid coupling
p	Pump impeller
ind	Induced flow
o	Outer radius of fluid coupling
p	Pump
ref	Reference value
t	Turbine

REFERENCES

1. **Sinclair, H.**, Recent developments in hydraulic couplings. Proc. Institute of Mechanical Engineers, 1935, **130**, 75-191.
2. **Sinclair, H.**, Some problems in the transmission of power by fluid couplings. Institute of Mechanical Engineers, Proc. of the Meeting on 22nd April 1938, 82-183.
3. **Qualman, J. W.** and **Egbert, E. L.** Fluid couplings passenger car automatics transmission. Passenger Car Autom. Transm., SAE Transm. Workshop Meeting, 2nd Edn, Adv. Engng, **5**, 137-150. (SAE., New York)
4. **Wallace, F. J.**, **Whitfield, A.** and **Sivalingam, R.** A theoretical model for the performance prediction of fully filled fluid couplings. International Journal of Mechanical Sciences, 1978, **20**, 335 – 347.
5. **Whitfield, A.**, **Sivalingam, R.** and **Wallace, A.** The performance prediction of fluid coupling with the introduction of a baffle plate. International Journal of Mechanical Sciences, 1978, **20**, 729-736.
6. **Ishihara, T.**, **Furuya, S.** and **Mori, K.** Characteristics of fluid coupling. Bulletin of Japanese Society of Mechanical Engineers, 1967, **33**, 496-508 (in Japanese).
7. **Uchiyama, K.**, **Takagi, T.** and **Okazaki, T.** Relationship between fluctuation of output shaft rotation speed and internal flow in fluid coupling of variable filling type. Bulletin of JSME, 1980, **46** (405), 893-901 (in Japanese).
8. **Uchiyama, K.** and **Takaya, H.** Internal flow in scoop tube of a fluid coupling. Bulletin of JSME, 1985, **51** (468), 2515-2522 (in Japanese).

9. **Tagenaga, H., Tsutsumi, M., Masuzawa, C. and Suguro, Y.** A flow visualization of internal flow of scoop tube type fluid coupling. Proc of 3rd ASME/JSME Joint Fluids Engineering Conference, July, 1999, San Francisco, Paper no FEDSM99-7201.
10. **Matsui, J., Kurokawa, J. and Imamura, H.** An experimental study on the very slow reduction of torque in a variable capacity type fluid coupling. Bulletin of JSME, Turbomachinery, 2003, **31** (12), 739-744 (in Japanese).
11. **Middelmann, V.** Analysis des systemverhaltens hydrodynamischer kupplungen bei variation der kreislaufgeometrie. PhD thesis, 1992, Ruhr-Universitat Bochum (in German).
12. **Wiennholt, H.** Verlustzustand und instabilitaten der stromung in hydrodynamischen kupplungen bei variation der schaufelgitter. PhD Dissertation, 1993, Ruhr-Universitat Bochum (in German).
13. **Kurokawa, J., Imamura, H., Sugiyama, K., Kimura, K., Matsui, J. and Kitahora, T.** Investigation of flow characteristics in a fluid coupling of variable capacity type, Bulletin of JSME, 2000, **66** (646), 1392-1398 (in Japanese).
14. **Kost, A., Mitra, N.K., and Fiebig, M.** Computation of unsteady 3D flow and torque transmission in hydrodynamic couplings. Proc. of the International Gas Turbine and Aeroengine Congress and Exposition, Hague, Neth., 13-16 June 1994, ASME paper 94-GT-70, 1-7.
15. **Bai, L, Kost, A., Fiebig, M. and Mitra, N.K.** Numerical investigation of unsteady incompressible 3D turbulent flow and torque transmission in fluid couplings. Proc. of the International Gas Turbine and Aeroengine Congress and Exposition, Hague, Neth, 13-16 June 13-16 1994, ASME Paper 94-GT-69, 1-8.

16. **Formanski, T., Huitenga, H., Mitra, N.K. and Fiebig, M.** Numerical investigation of 3D flow and torque transmission in fluid couplings under unsteady working condition. Proc. of the International Gas Turbine and Aeroengine Congress and Exposition, ASME, Houston, TX, USA, 5-8 June 1995, ASME Paper 95-GT-81, 1-8.
17. **Huitenga, H., Formanski, T., Mitra, N.K. and Fiebig, M.,** 1995. 3D flow structures and operating characteristic of an industrial fluid coupling. Proc. of the International Gas Turbine and Aeroengine Congress and Exposition, ASME, Houston, TX, USA, 5-8 June 1995, ASME Paper 95-GT-52, 1-6.
18. **Bai, L., Fiebig, M., and Mitra, N.K.** Numerical analysis of turbulent flow in fluid couplings. Journal of Fluids Engineering, ASME Trans., 1997, **119** (3), 569-576.
19. **Huitenga, H. and Mitra, N.K.,** Improving startup behaviour of fluid couplings through modification of runner geometry: part 1 – fluid flow analysis and proposed improvement and part 2 modification of runner geometry and its effect on the operation characteristics. Journal of Fluids Engineering, ASME trans, 2000, **122**, 683-687 and 689-693.
20. **Bai, L., Blomerius, H., Mitra, N.K. and Fiebig, M.** Numerical simulation of unsteady incompressible 3D turbulent flow in fluid couplings with inclined blades. Proc. of the 1994 International Mechanical Engineering Congress and Exposition, ASME, Chicago, IL, USA, 6-11 November 1994, Dynamic Systems and Control Division (Publication) DSC, 55-2, Dynamic Systems and Control, 2 (2), 609-616.

21. **Flack, R.** and **Brun, K.** Fundamental analysis of the secondary flows and jet-wake in a torque converter pump - part I: model and flow in a rotating passage.

Journal of Fluids Engineering, ASME trans, 2005, **127**, 66-74.

22. Fluent User's Guide, <http://www.fluent.com>.

Tables

	Mesh-1	Mesh-2	Mesh-3	Mesh-4
Number of mesh cells	16,744	61,104	115,904	213,720

Table 1 Meshes used in mesh dependence investigation

	Small coupling		Large coupling	
	HTC-1	HTC-2	HTC-3	HTC-4
Metal wall temperature T_w (K)	373.15	373.15	473.15	473.15
Operating Pressure P_o (Pa)	1.0×10^5	1.0×10^5	0.35×10^5	1.0×10^5
Speed Ratio $SR = \omega_t / \omega_p$	0	0.67	0.56	0.56
Rotational Reynolds $Re = \rho \omega_p D_o^2 / \mu$	7.33×10^5	4.40×10^6	1.19×10^6	3.43×10^6
Rotational Mach number $Ma = R_o \omega_p / a$	0.12	0.73	0.52	0.52

Table 2 Summary of test cases for the heat transfer investigation

Figures.

(1-a) Working mechanism

(1-b) Wheel geometry

Figure 1 Schematic illustration of an industrial fluid coupling

(2-a) Velocity distribution

(2-b.1) At R_2

(2-b.2) At R_1

(2-b) Velocity triangle

Figure 2 1D model

Figure 3-a The CFD model and computational mesh for the small coupling

**Figure 3-b Torque obtained vs number of mesh cells, Small coupling, Oil, SR=0,
 $Re=0.682 \times 10^6$**

Figure 4 Typical time history of the shaft torque obtained from the CFD simulation, Small coupling, Oil, SR=0, Re=0.682x10⁶

Figure 5 Typical flow separations obtained from the CFD simulation, Small coupling, Oil, SR=0, Re=0.682x10⁶

Figure 6 Comparison of shaft torque characteristics

Figure 7 A instantaneous velocity vector plot on an angular plane, Large coupling, Oil, SR=0.8, Re=0.635x10⁶.

Figure 8 Time Mean velocity profile on the mid-axial plane, Large coupling, Oil, SR=0.8, $Re=0.635 \times 10^6$.

Figure 9 Time Mean gauge pressure profile on the mid-axial plane and instantaneous gauge pressures in the gap, Large coupling, Oil, SR=0.8, $Re=0.635 \times 10^6$.

Figure 10 Time Mean and instantaneous flow incidence on the mid-axial plane, Large coupling, Oil, SR=0.8, Re=0.635x10⁶.

Figure 11 Instantaneous axial velocity contours on the mid-axial plane, Large coupling, Oil, $SR=0.8$, $Re=0.635 \times 10^6$.

Figure 12 Bulk temperature rise over wall temperature and mean torque drop from air flow solutions.

Figure 13 Instantaneous static air temperature contours, Large coupling, Air, $SR=0.56$, $Ma=0.52$ and $Re=3.43 \times 10^6$

Figure 14 Instantaneous Nusselt number contours on vane pressure surfaces, Large coupling, Air, $SR=0.56$, $Ma=0.52$ and $Re=3.43 \times 10^6$

Figure 15 Mean Nusselt number for air flow in couplings

Figure 16 Sectional View of the CFD model for the cooling throughflow investigation

Figure 17 Velocity profiles on the mid axial plane, large coupling, cooling throughflow $m_q=0$, Oil, $SR=0$, $Re=1.36 \times 10^6$

Figure 18 Effect of number of vanes, three test cases with 10, 20 and 40 vanes, Oil, $SR=0.8$, $Re=0.635 \times 10^6$

(19-1) At R2, as shown in Figure 2

(19-2) At R1, as shown in Figure 2

Figure 19 Illustration of the angled vane and velocity triangles

Figure 20 Comparison of shaft torque with and without angled vanes