

Numerical Analysis on Three Dimensional Dynamic Flow of Vehicle Hydrodynamic Coupling

Zhifeng Dong¹, Gong Dan¹, Yuzhao Zhang¹

¹China University of Mining and Technology (Beijing), Beijing, 100083, China

Keywords: Vehicle; Hydrodynamic coupling; Flow field; Numerical analysis

Abstract. Hydrodynamic coupling is one of the important components in the vehicle transmission system, in which, the power of engine is transmitted through the circular flow of the hydro oil. The three dimensional fluid model was built, and the dynamic flow in the hydrodynamic coupling was calculated with calculation fluid dynamics software. The numerical results and graphs turn out that the pressure increases rapidly with the distance from the coupling axis, but pressure has only a small change in direction of parallel to the coupling axis.

Introduction

The hydrodynamic coupling is widely used in the field of vehicle, tractor, scraper and excavator. The hydrodynamic coupling is a component transferring energy with the kinetic energy of fluid, which provides flexible connection between driving shaft and driven shaft, and transfer the torque from the driving shaft to the driven shaft by the kinetic energy of the fluid. In the application of vehicle hydrodynamic coupling, the pump wheel is connected with the fly wheel, and the turbine wheel is connected with the input shaft of the transmission. The fluid circularly flows in the pump and turbine wheel, thus delivers the torque from the engine to the transmission.

There has been a lot of research about the internal flow of hydraulic element with CFD method, but not much in the three dimensional numerical simulation in the hydrodynamic coupling. In 1997, L. Bai et al. calculated the flow field of the hydrodynamic coupling under the rotational coordinates with the $\kappa-\varepsilon$ model and FVM, achieved the how vortex affects the performance of hydrodynamic coupling. Charles. N. McKinnon analyzed the both forward and reverse condition of the turbine wheel, achieved the energy loss and the property parameters. H. Huitenga and N. K. Mitra found that the changing in the geometry have great influence in the starting characteristic of hydrodynamic coupling.

Control equation

Here, fluid flows with viscosity in the coupling incompressible. The basic control equations are of continuity equation and the N-S equation.

Equation of continuity:

$$\text{div}u=0 \quad (1)$$

The N-S equation:

$$\rho \frac{Du}{Dt} = \rho f - \nabla p + \mu \nabla^2 u \quad (2)$$

In the equation: u represents speed vector; f represents volume force per unit mass; P represents pressure; μ represents dynamic viscosity;

∇ is the Gradient operator, $\nabla = (\frac{\partial}{\partial x_1}, \frac{\partial}{\partial x_2}, \frac{\partial}{\partial x_3})$;

∇^2 is the Laplace operator, $\nabla^2 = (\frac{\partial^2}{\partial x_1^2}, \frac{\partial^2}{\partial x_2^2}, \frac{\partial^2}{\partial x_3^2})$.

The effect of turbulence is taken into consideration, based on the equation of continuity and the N-S equation, close the equation set with $\kappa-\varepsilon$ model.

$\kappa-\varepsilon$ equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j}] + G_k - Y_k + S_k \quad (3)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_j}] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (4)$$

In the equation: G_k represents turbulent kinetic energy cause by speed gradient of laminar flow, G_b represents turbulent kinetic energy caused by buoyance, $C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}$ are constant, $\sigma_\varepsilon, \sigma_k$ are the turbulence Prandtl number of k and ε equation.

CFD modeling

The 3D model of the coupling is built as shown in Fig.1(a) to get the detailed data in the flow field of the coupling. The wheels of the pump and turbine are the same. The wheel radius is 380mm, number of blade is 12.

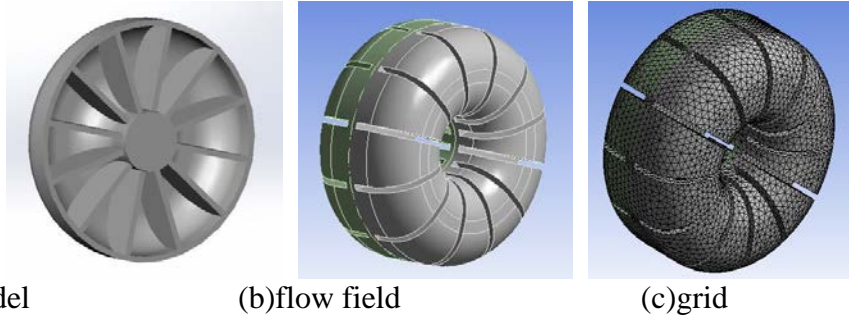


Fig.1: Model

In the starting period, both wheels are not working synchronously, so the calculating area is not symmetrical, the whole working chamber is taking into calculating. The calculating area is shown in Fig.1(b). Meshing the model with tetrahedron, the final mesh model is shown in Fig.1(c), number of element is 79884.

Before the calculating, some simplification and assumption are essential:

- 1) The density and viscosity of the working fluid are constant during the working time, considering the working fluid as incompressible and viscous. The density is 889.1 kg/m^3 , the viscosity is 0.00129 Pas .
- 2) The working chamber is full of working fluid, the pump wheel rotates at constant speed, and the whole system is under well cooling.
- 3) Generally, the fluid of cooling is less than 0.2%, the fluid loss in the working chamber is little enough to be ignored.

In the calculating, FVM is used to solve the conservative N-S equation, SIMPLE algorithm is used to solve the speed-pressure coupling, consider the turbulent model as standard $k-\varepsilon$.

The internal flow is transient, so the total chamber is divided into two zones, using the sliding grid theory to handle the interface of the two zones.

The pump zone is considered rotating at constant speed of 1800rpm, using 6DOF UDF to define the turbine zone, set the mass, center of mass, rotational inertia of the turbine wheel, just allow the DOF of rotation on Z.

Analysis of the flow field

To get better observation of the internal flow, the sections are selected as below. The two faces shown in Fig.2 are individually belong to each zone, and the face shown in Fig.3 is the section of a circle.

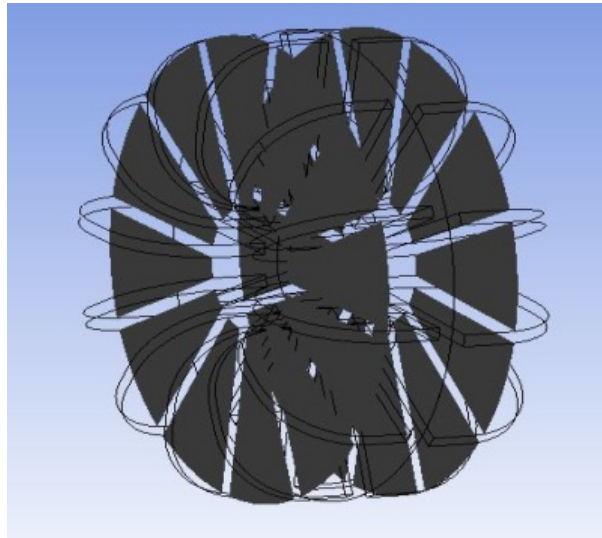


Fig.2: section1

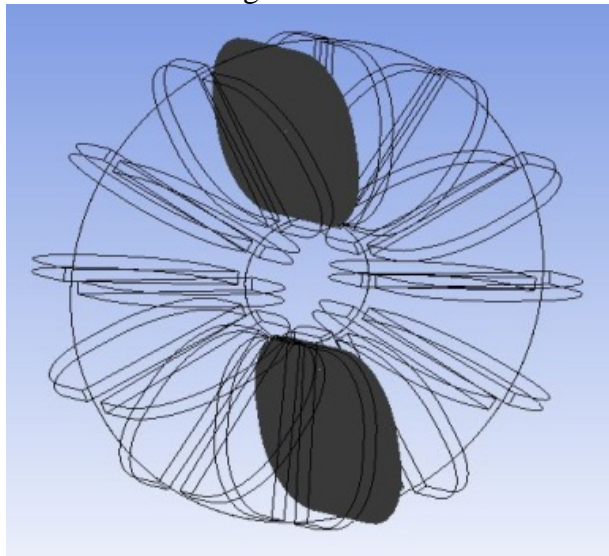


Fig.3: Section2

Extracting the pressure distribution of section1 at 0.001s and 0.1s, the result is shown in Fig.4, in which the figure face on the right side belongs to the pump zone, and face on the left side belongs to the turbine zone.

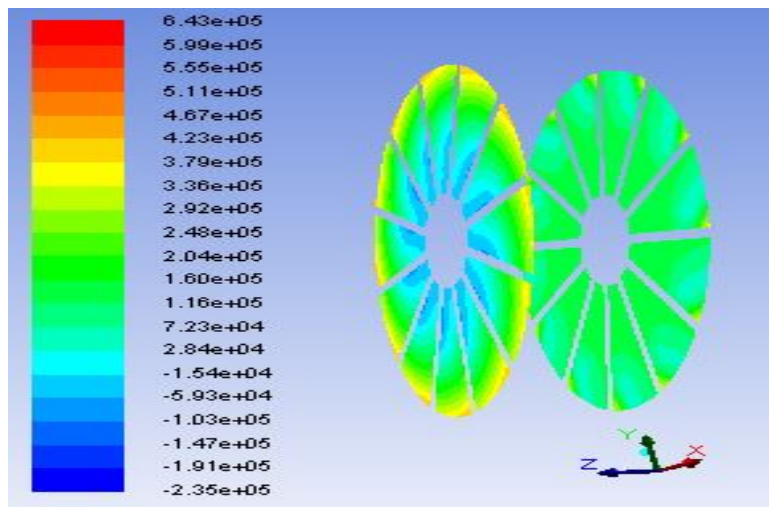


Fig.4: Pressure distribution on 0.01s

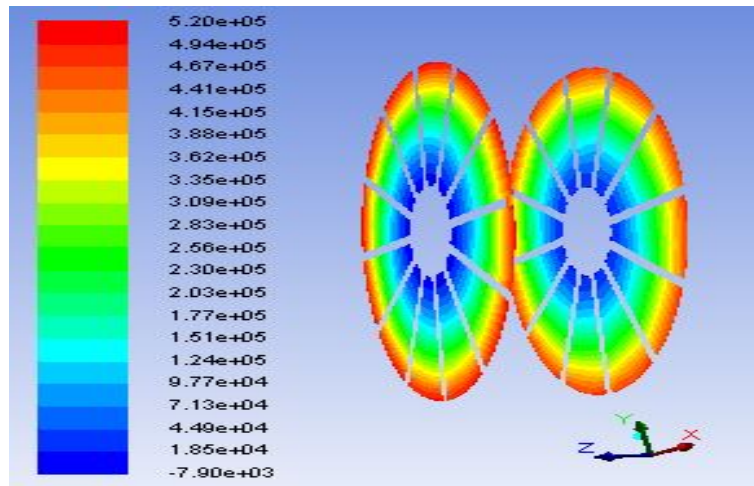


Fig.5: Pressure distribution on 0.1s

As shown in Fig.4, with the rotating of the pump wheel, the pressure in the pump zone rises then, fluid with high pressure flows to the turbine zone, impacts the turbine wheel, and drives the turbine wheel to rotate. With time growing, the pressure in the each zone tends to be stable, and the speed of turbine wheel gets closer to the pump wheel. After 0.1s, the fluid flows is mainly stable, the turbine wheel rotates with the same speed as the pump wheel.

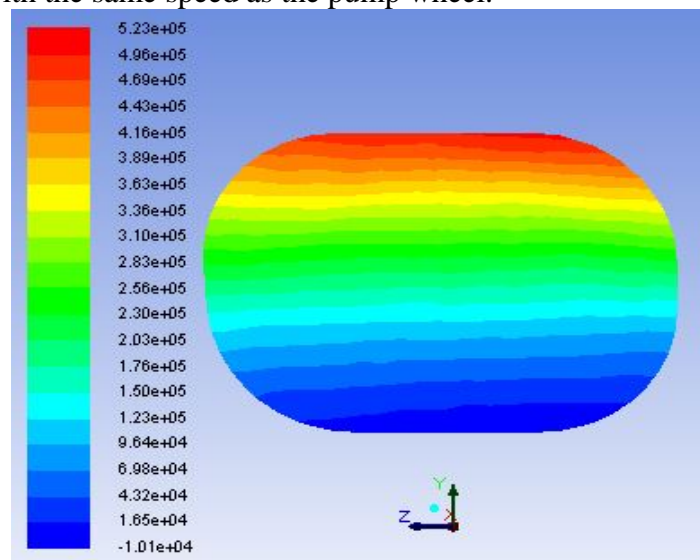


Fig.6: Result of pressure distribution in section 2

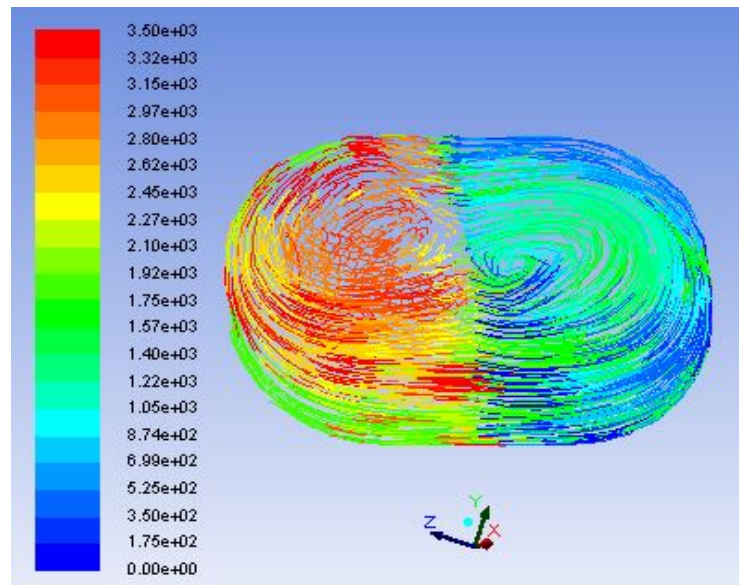


Fig.7: Path lines in section 2

Fig.6 shows the pressure distribution in a single circle. The fluid far away from the rotation axis gets higher pressure due to the centrifugal acceleration.

Fig.7 is the picture shows the track of the fluid in the single circle, it shows the fluid flow in the circle circularly.

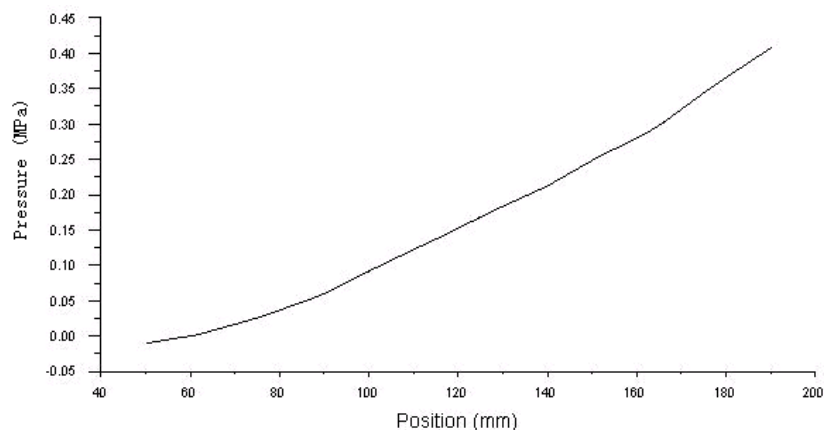


Fig.8: Pressure distribution on curves in the radial direction

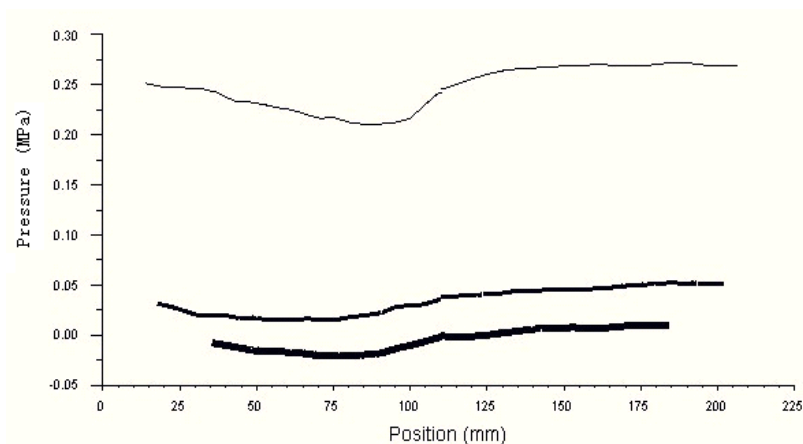


Fig.9: Pressure distribution on curves in the axial direction

Fig.8 shows the pressure change in the radial direction, in which, the pressure increases from 0.0 to 0.42 MPa with the distance from the coupling axis 50 to 200 mm. Fig.9 shows the pressure

changes in the axial direction in the different circles, in which, the curves from the top to the bottom represent separately to the places with a distance of 150mm, 80mm, and 60mm from the hydrodynamic coupling axis. The pressure on direction parallel to the axis is almost stable.

Summary

In this paper, the three dimensional dynamic flow in the hydrodynamic coupling is studied. The pressure distribution of the internal flow is investigated by the numerical calculation which reflects the basic characteristic of the internal flow in the hydrodynamic coupling. The numerical results and graphs turn out that the pressure increases rapidly with the distance from the coupling axis, but pressure has only a small change in direction of parallel to the coupling axis. The research founds the base for further analysis on the hydrodynamic coupling and understanding of the internal flow.

Acknowledgements

This work was financially supported by “The Fundamental Research Funds for the Central Universities” (No.2010YJ04). The authors would like to acknowledge and thanks for the support.

References

- [1] BAI L, FIEBLG M, MITRA N K. Numerical Analysis of Turbulent Flow in Fluid Coupling[J]. ASME Journal of Fluids Engineering, 1997, 119:569-576.
- [2] CHARLES N, MCKINNON, BRENNEN DANAMICHELE, et al. Hydraulic Analysis of a Reversible Fluid Coupling[J]. Journal of Fluids Engineering, 2001(2): 249-255.
- [3] HUITENGA H, MITRA N K. Improving Startup Behavior of Fluid Couplings Through Modification of Runner Geometry: Part I—Fluid Flow Analysis and Proposed Improvement[J]. Journal of Fluid Engineering, 2000(4):683-688.
- [4] HUITENGA H, MITRA N K. Improving Startup Behavior of Fluid Couplings Through Modification of Runner Geometry: Part II—Modification of Runner Geometry and Effects on the Operation Characteristics[J]. Journal of Fluid Engineering, 2000(4): 689-693.
- [5] CHINWOO LEE, etc. Three Dimensional Flow Field Simulation to Estimate Performance of a Torque Converter[J]. SAE2000-01-1146
- [6] T. SHIEH, etc. Torque Converter Analytical Program for Blade Design Process[J]. SAE2000-01-1145
- [7] DONG Y, LAKSHMINARAYANA B. Experimental Investigation of the Flow Field in an Automotive Torque Converter Stator[J]. ASME Journal of Fluids Engineering, 1999,121:788-797.
- [8] DONG Y, LAKSHMINARAYANA B, MADDOCK D. Steady and Unsteady Flow Field at Pump and Turbine Exit of a Torque Converter[J]. ASME Journal of Fluid Engineering, 1998, 120:538-548.
- [9] FLACK R, BRUNK. Fundamental Analysis of the Secondary Flows and Jet-Wake in a Torque Converter Pump-Part I: Model and Flow in a Rotating Passage[J]. Journal of Fluid Engineering, 2005(1): 66-74.
- [10] FLACK R, BRUN K. Fundamental Analysis of the Secondary Flows and Jet-Wake in Torque Converter Pump Part II:Flow in a Curved Stationary Passage and Combined Flows[J]. Journal of Fluids Engineering, 2005(1):75-82
- [11] FLUENT Inc. FLUENT User's Guide. Fluent Inc,2003