Computers and Fluids 000 (2016) 1–9

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

Flow and performance analyses of a partially-charged water retarder

Nahmkeon Hur[a,b,*], Mohammad Moshfeghi[b], Wonju Lee[a]

[a] Department of Mechanical Engineering, Sogang University, Seoul, Korea
[b] Multi-Phenomena CFD Engineering Research Center, Seoul, Korea

ARTICLE INFO

Article history:
Received 7 April 2016
Revised 30 September 2016
Accepted 15 October 2016
Available online xxx

Keywords:
Water retarder
Partially charged
CFD
Multi-phase flow
Volume of fluid (VOF)

ABSTRACT

Two-phase flow inside a water retarder is numerically studied to understand the water retarder performance and momentum dissipation mechanism for partially and fully-charged conditions at different rotational speeds, and the effects of external cooling flow into the water retarder. In order to accurately capture the flow interaction between the blades of different rotating speeds, the transient simulations are performed using unsteady turbulent flow assumption using a sliding mesh technique.

The Realizable k-ε turbulence model and the Volume of Fluid (VOF) method are adopted to simulate two-phase motion and interface of air and water mixture inside the retarder. A momentum index analysis is proposed for investigation of flow momentum and its dissipation. Results of partially-charged cases demonstrate that as a general trend, air is accumulated at the center of the hydraulic system forming a toroidal shape. At a low charged water volume flow, circulation inside the retarder and, hence, the retarding torque are trivial. Furthermore, it is observed that the retarding torque becomes very small for the charged water values lower than a special value. Finally, the simulations of the fully-charged open system with external cooling flow (through an inlet and an outlet) reveal that the retarding torque and also flow circulation between wheels decrease with the increase in flowrate.

1. Introduction

Hydraulic retarders are being industrially employed as primary or supplementary brake systems for rotating shafts deceleration. A hydraulic retarder is basically composed of one rotating wheel (rotor) and one stationary wheel (stator), as shown in Fig. 1(a,b). As it is illustrated, the wheels include a number of straight or curved blades, which provide passages (channels) for a fluid circulation inside the system. The assembly may be fully or partially charged with oil or water. As the rotor rotates, it centrifugally accelerates the working fluid and shoots it out from the outer part of its channels. The flow then enters the stator and streams inside the channels along casing curvature towards inner radii. Meanwhile, it applies external torque to the stator walls. Since the stator is fixed, it also applies an opposite torque to the fluid, which acts as a resisting torque and, in turn, decelerate the rotor. The circulating fluid then leaves the secondary wheel at inner radii and re-enters to the primary wheel, and the cycle continues. During this process, the mechanical energy of the circulating fluid is dissipated into heat. Almost similar working principle is valid for hydraulic couplings. However, the hydraulic couplings are used for transmitting (converting) torque between separated shafts of power generators, light and heavy vehicles, marine, etc [1]. Therefore, in hydraulic couplings the second wheel is also connected to a shaft and rotates as a result of the applied torque. Hence, the rotor and stator are called primary and secondary wheels.

Because of the fundamental similarities between the hydraulic couplings and retarders, both have been usually studied numerically and/or experimentally as one category. Ainley et al. [2] measured the flow velocity inside a torque converter filled with oil at a constant pump speed at different speed ratios. The outcomes revealed a noticeable area of separation inside the hydraulic coupling. Also, mass flowrates were found to decrease as the speed ratio increased. In addition, they compared the core-to-shell and blade-to-blade torque distributions. In another set of experiments done by Flack et al. [3], the effects of pump speed at a fixed speed ratio and also different oil viscosities have been measured. The results showed that torque converter mass flowrate increased approximately linearly with increasing pump rotational speed at a fixed speed ratio. However, they reported that the flowrate was not directly proportional to pump rotational speed. In addition, their measurements showed that mass flowrate decreased as the oil viscosity decreases. Huitenga and Mitra [4] numerically studied the 3-D fluid flow in order to understand the time dependent effects and to improve the startup behavior of a hydraulic coupling. They modified the geometry of the primary wheel (pump) and investigated the effects of modifications on the pump and secondary wheel

* Corresponding author.
E-mail address: nhur@sogang.ac.kr (N. Hur).

http://dx.doi.org/10.1016/j.compfluid.2016.10.033
0045-7930/

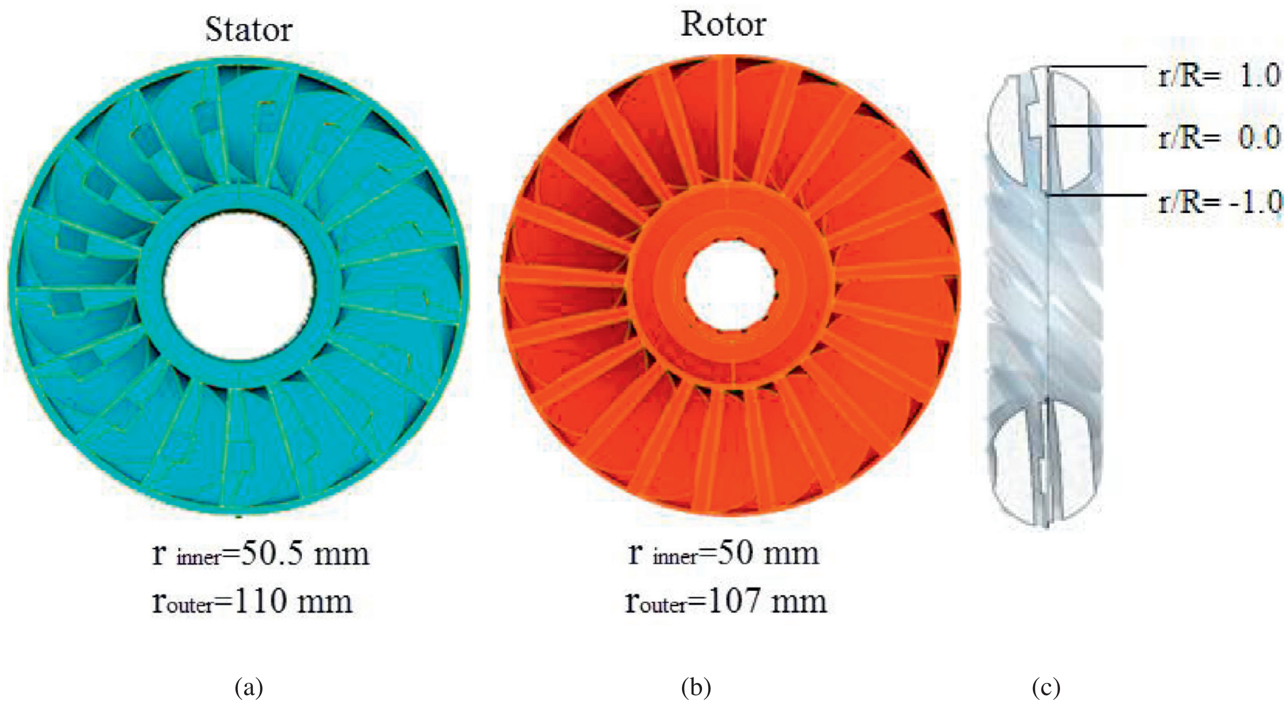

Fig. 1. Geometry and size of the baseline water retarder (a) stator and (b) rotor; (c) Convention for non-dimensional radial position.

(turbine) [5]. Habsieger and Flack [6] studied flow characteristics at the interface of the primary and secondary wheels of a scaled model of a truck torque converter at seven different speed ratios. The laser velocimetry was adopted for their measurement and flow velocity at the interfacial plane was investigated. The results showed that as the speed ratio increases, the high velocities move to the pressure-shell corner and then to both the core-suction and the pressure shell corners at the primary wheel exit. Also they observed that there exist a reverse flow at the turbine inlet at low speed ratios. Schweitzer and Gandham [7] validated CFD results of a torque converter using k-ε turbulence model with log-law assumption. In another research, Hampel et al. [8] investigated the average fluid distribution in the transversal planes of a hydrodynamic coupling through gamma tomography measurements at different speed ratios and charged water volumes. The experimental investigations reconstructed phase distributions together with useful information about inter-relation of the coupling design and the local transfer of mass and energy between the wheels. Da silva et al. [9] employed a planar array sensor method and measured flow distribution and two-phase flow patterns inside a water coupling at two different speed ratios for a partially-charged coupling. Their tests showed that at the zero slip condition a steady flow was established and the liquid level was parallel to the rotation axis and the phase transition between air and water was very sharp. Luo et al. [10] investigated the flow inside a partially-charged hydraulic coupling using RNG k-ε turbulence model and Volume of Fluid (VOF) model. The working fluid was oil (VOF = 80%) and the rotating speed was fixed at 3000 rpm. Their conclusion showed formation of an oil wrap around a toroidal-shape air accumulation at the center. In another research the steady state flow of a fully-charged water driver at different speed ratios using SST k-ω turbulence model has been investigated. The result showed similar flow field distribution, despite different rotational speeds. Also, it was concluded that the pressure distribution on the torus section became more radially distributed at higher speed ratios [11]. Jain and Tiwari [12] compared performance of hydraulic coupling for different working fluids in order to investigate the effects of viscosity of oil and water on torque transmission using two different charged conditions. Their conclusion showed that the transmitted torque decreases as viscosity of fluid increases. Hur et al. [13] numerically investigated flow physics and streamline patterns inside a water drive at three speed ratios and three charged fluid conditions. They reported the negative effects of unwet surface effects at cases at high speed ratios. Liu et al. [14] investigated internal torque converters using RNG k-ε, DES and LES turbulence models and compared flow field with experimental data. Their conclusions revealed that although the results of DES and LES can precisely predict the flow details inside highly complex geometries; however, the RNG k-ε model are also reasonably accurate for the predictions of velocity magnitudes and flow structures.

As it can be seen in the literature survey, a majority of research projects are limited to special operational conditions of hydraulic drives. Hence, there is a serious need for a comprehensive study of hydraulic retarders in a wide range of operational conditions. Therefore, the present research aims to investigate three-dimensional two-phase flow numerical analysis of a water retarder with different values of charged water at various rotating speeds in order to provide a performance map as a fundamental standpoint for interpretation and analysis of flow details inside a water retarder.

2. Physics and governing equations

The flow field is solved using the incompressible fluid assumptions for both water and air. The simulations are performed using VOF method. Hence, the same governing equations of a single phase flow are applied to each cell in the domain, and the equations are solved for an equivalent fluid whose physical properties are calculated as linear functions of the physical properties of its constituent phases and their volume fractions. In addition, all phases share the same velocity and pressure fields. For each computational cell, the equations of density (ρ) and dynamic viscosity