Numerical simulation of 3D unsteady flow in a rotating pump by dynamic mesh technique

S Huang, J Guo and F X Yang
School of Mechanical and Automotive Engineering,
South China University of Technology, Guangzhou, China 510640
E-mail: huangsi@scut.edu.cn

Abstract. In this paper, the numerical simulation of unsteady flow for three kinds of typical rotating pumps, roots blower, roto-jet pump and centrifugal pump, were performed using the three-dimensional Dynamic Mesh technique. In the unsteady simulation, all the computational domains, as stationary, were set in one inertial reference frame. The motions of the solid boundaries were defined by the Profile file in FLUENT commercial code, in which the rotational orientation and speed of the rotors were specified. Three methods (Spring-based Smoothing, Dynamic Layering and Local Re-meshing) were used to achieve mesh deformation and re-meshing. The unsteady solutions of flow field and pressure distribution were solved. After a start-up stage, the flow parameters exhibit time-periodic behaviour corresponding to blade passing frequency of rotor. This work shows that Dynamic Mesh technique could achieve numerical simulation of three-dimensional unsteady flow field in various kinds of rotating pumps and have a strong versatility and broad application prospects.

1. Introduction
Unsteady flow analysis in rotating pumps has important values of both scientific research and engineering applications, for understanding dynamic characteristics of pumps, improving pump performance and reliability. However, the rotational domains would interfere each other or with the stationary domains in some cases of rotating pumps, such as roots blower, gear pump, roto-jet pump, and so on. Especially, so far no numerical simulations have been reported on transient flow of roto-jet pump except some steady simulations\cite{1-2} by Frozen Rotor Approach. For those rotating vane pumps like centrifugal pump in which impeller domain does not interfere with stationary domain, so Sliding Mesh technique\cite{3-4} can be used for unsteady flow calculation. Both for Frozen Rotor Approach and Sliding Mesh technique, multiple reference frame (MRF)\cite{5} is used. The impeller domain is set in the rotating frame of reference; the stationary domains are set in the inertial frame. Data are exchanged on the interfaces among reference frames to achieve the calculation of the overall flow field.

Taking into account self-adaptive features and broad application prospects of Dynamic Mesh technique, authors carried out numerical simulation of unsteady flow in three kinds of rotating pumps, roots blower, roto-jet pump and centrifugal pump, using 3D Dynamic Mesh technique in this paper, to explore the feasibility and potential of the Dynamic Mesh technique for the 3D calculation of pumps.

2. Calculation method
Dynamic Mesh technique\cite{6-8} is generally used to model flows where the shape of the domain changes with time due to motion on the domain boundaries. The update of the volume mesh is handled at each
time step based on the new positions of the boundaries. Three mesh motion methods are often used to update the mesh in the deforming regions subject to the motion defined at the boundaries: a) spring-based smoothing method \cite{7}; b) dynamic layering; c) local remeshing method \cite{8}.

For this computation, PRO-E software was applied to create the 3D flow computational domains of the rotating pumps (roots blower, roto-jet pump and centrifugal pump), respectively. The GAMBIT software was used to generate the initial meshes of domains. In the unsteady simulation, all the computational domains, as stationary, were set in one inertial reference frame by FLUENT code. The motion of domain boundary was described using Profiles regarding to the time-dependent position, angular velocity and orientation of the centre of gravity of moving objects (vanes, impeller). The standard k-\(\varepsilon\) turbulence model was selected for this study. The following boundary conditions were set: 1) pressure values in the inlet and outlet conditions; 2) no-slip solid wall conditions and the flow near the wall determined by the standard wall function. According to pump rotational speed \(n\) and the number of rotor blades \(Z\), rotor revolution is \(T=60/n\) (s), the blade passing period is \(T'=T/Z\) (s). Therefore, the time step for calculation were set \(\Delta t=T'/10\) (s) or below. Usually, the smaller time step was required in the case of roto-jet pump calculation.

3. Calculation model and results analysis

3.1. Roots blower
A quadricuspid roots blower was selected in the study. As shown in Fig. 1, the left and right rotors turn in counterclockwise and clockwise directions respectively. Air was the medium and treated as ideal compressible gas. The air flow process was assumed to be isothermal. Figure 1 illustrates the time-dependent deformation of computational domain and meshes.

![Figure 1. Dynamic mesh of quadricuspid roots blower with time](image)

Usually, numerical solution starts to exhibit regularly time-periodic behaviour after a start-up stage (about \(T/4\) in this case according to the monitoring). Figures 2 and 3 respectively show the variation of flow field and of static pressure distribution on the central section of the blower domain after the start-up stage, where \(\theta\) denotes the angular position of the left rotor.

![Figure 2. Flow field of roots blower with time (central section of the domain)](image)
3.2. Roto-jet pump

A small type of roto-jet water pump was selected in this work. The pump design conditions parameters were: flow rate, $Q=1.8$ m$^3$/h; head, $H=100$ m. As shown in Fig. 4, the number of impeller blades is $Z=6$. The flow model of a roto-jet pump consists of four domains: inlet section, impeller, rotor chamber and pick-up tube. The impeller and the rotor chamber are fixed together to run in a rotational speed, while the pick-up tube is a stationary part embedded in the rotor chamber. Figure 4 illustrates the domain surface mesh changes over time in one blade passing period $T'$. Figure 5 shows comparison of local grids in the central section of domain adjacent to the inlet of the pick-up tube over time. As seen in Fig. 5, the computational grids of the fixed pick-up tube remains unchanged, but rotational domain constantly adjusts its boundary adjacent to the pick-up tube over time. Accordingly, the grids in rotational domain have different degrees of deformation and remeshing (pointed by the green arrow).

Figure 6 shows the calculated time-dependent curve of outlet flow rate of roto-jet pump. As seen from Fig. 6, after start-up stage ($t\approx 0.01$ s, approximately $T/2$), the value of roto-jet pump outlet flow rate begins to pulsate regularly. In one $T$, the value of roto-jet pump outlet flow rate oscillates 6 times, the pulsation frequency is corresponding to the number of impeller blades ($Z=6$).
3.3. Centrifugal pump
A single-stage single-suction type centrifugal pipeline pump with five impeller blades (Z=5) was selected for this study. The design parameters of the pump were: flow rate, Q=155 m$^3$/h; head, H=64 m. The computational domain for the flows comprised the suction pipe, impeller and the casing. Figure 8 presents a comparison of the grids of the computational domain corresponding to the central section. The grid in the vicinity of the blade tip near the exit is enlarged for the sake of clarity. From this figure, it is evident that the deformations and local re-meshing (see the green box) occur in the domain grid.

For comparison, simulation of unsteady flow in centrifugal pump were performed both using the Dynamic Mesh technique and conventional Sliding Mesh technique. The calculation model, initial grid, initial conditions, boundary conditions, and software settings were the same for both techniques. Figure 9 shows variations in the residuals obtained using the Dynamic Mesh and Sliding Mesh techniques, respectively, with increasing number of iterations. The figure illustrates variations in residues derived from the continuity equation, momentum equation along three axes (x, y and z), and the transport equations considering turbulent flows kinetic energy ($k$) and its dissipation rate ($\varepsilon$). The results are presented until simulation time $t \approx 0.2069$ s for both cases, which corresponds to near 10T. Each oscillation in the residues represents convergence for a given criteria over a time-step iteration. The results clearly show relatively rapid convergence of the solutions obtained from the Dynamic Mesh technique than those from the Sliding Mesh technique. For the final time ($t \approx 0.2069$ s) considered in the figure, the maximum residual from the Dynamic Mesh is $10^{-1}$ or less, while the
Sliding Mesh method yields peak residual in the order of 1.0. The Dynamic Mesh method involves a total of 15500 iterations for the same period, which is substantially smaller than 47000 iterations employed in the Sliding Mesh method. The iteration rate of the Dynamic Mesh technique is thus nearly three times that of the Sliding Mesh method for the case considered in this study. This is primarily attributed to the fact that the Dynamic Mesh computation is only defined in an inertial reference frame, despite the adverse influencing factors such as grid deformation and re-meshing. The topology relationship between the old and new mesh nodes are retained to ensure good computational accuracy and efficiency. The Sliding Mesh method, on the other hand, involves computation in multiple reference frames. Data transfer among the multiple reference frames affects the iteration rate and thus the computational efficiency in an adverse manner.

![Figure 8. Deformation of grids (on central section)](image)

![Figure 9. Residuals by both mesh techniques](image)

The time-histories dimensionless radial force responses, $F_r'$ and $F_	heta'$, acted on the impeller, obtained using the Dynamic and Sliding Mesh techniques, are compared in Fig. 10. From the results it is seen that the two methods rapidly approach comparable responses, although substantial differences between the results attained from the two methods are evident in the transient state. The radial force component responses from both the methods exhibit nearly steady-state behaviour at $t \geq 0.02$ s, suggesting convergence after approximately only one revolution of the impeller. The results further show nearly 5 oscillation cycles of the radial force during each impeller revolution. The oscillation frequency of each radial force component is thus 5 times the fundamental frequency, which corresponds to the number of impeller blades ($Z=5$). The two methods yield quite comparable radial force responses at $t \geq 0.10$ s or in nearly five impeller revolutions. The results thus suggest Dynamic
Mesh technique is not only feasible for simulation of 3D unsteady flows in pumps but substantially more computationally efficient compared to the Sliding Mesh technique.

4. Conclusions
(1) Dynamic Mesh technique could achieve numerical simulation of three-dimensional unsteady flow field in various kinds of rotating machines including roots blower, roto-jet pump, centrifugal pump and have a strong versatility and broad application prospects.
(2) With the same calculation model, initial grid, initial conditions, boundary conditions and software settings for centrifugal pump, the calculation results of Dynamic Mesh and Sliding Mesh approach consistent after a period of pump start-up time, and Dynamic Mesh technique has faster iterative speed.
(3) Computation by Dynamic Mesh technique for rotating pump is only defined in one inertial reference frame; the topological relationship of the old and new mesh nodes is kept to ensure good computational accuracy and efficiency.

References
[1] Liu Y, Chen J X and Li Y L 2009 *Drainage & Irrigation Machinery* 27 367-372 (in Chinese).
[2] Wang C L, Zhao C L, Zhang T F and Liu D 2012 *Materials Eng. & Automatic Control* 562-564 1369-1372.
[3] Gonzalez J, Fernandez J, Blanco E and Santolaria C 2002 *J Fluids Eng.* 124 348–355.
[4] Huang S, Mohamad A A, Nandakumar K, Ruan R Z and Sang D K 2010 *Progress CFD* 10 239-245.
[5] Luo J Y, Issa R I and Gosman A D 1994 *Int. IChemE Symposium Series* 136 549-556.
[6] Maple R C and Belk D M 1994 *Numerical Grid Generation in CFD and Related Fields* (UK: Pine Ridge Press) 305-314.
[7] Batina J T 2012 *AIAA J.* 28 1381-1388.
[8] Weatherill N P 1992 *Computers & Mathematics with Applications* 24 129-150.