THREE DIMENSIONAL UNSTEADY SIMULATION OF A MODEL FRANCIS HYDRAULIC TURBINE

Qi Shao[1], Yulin Wu[2], Shuhong Liu[3], Weizhang Wu[4], Xingming Tao[5]
1. Thermal Engineering Department, Tsinghua University, China, 100084, shaoqi97@mails.tsinghua.edu.cn
2. Thermal Engineering Department, Tsinghua University, China, 100084, yulinwu@163bj.com
3. Thermal Engineering Department, Tsinghua University, China, 100084, liushuhong@mail.tsinghua.edu.cn
4. Harbin Institute of Large Electric Machinery, Harbin, China, 150040
5. Harbin Institute of Large Electric Machinery, Harbin, China, 150040

Nomenclatures

Term	Symbol	Definition	Term	Symbol	Definition
Rotating speed of runner	n	r/min	Guide vane opening	α	mm
Pressure head of turbine	P	Pa	Peak to peak value of pressure	ΔP	Pa
Relative amplitude	$\Delta P/P$	%	Diameter of runner	D	m
Hydraulic head of turbine	H	m	Dominating frequency of vortex rope	f	Hz
Unit speed	n_{11}	$n_{11}=nD/\sqrt{H}$			

1. Introduction

For Francis hydraulic turbines, one of the major difficulties is that severe pressure oscillations can occur under part load operations. They are generated by an unsteady vortex behavior in draft tube, the so-called vortex rope. In this paper, the unsteady simulation of a model Francis hydraulic turbine is shown and the computational results are compared with the experimental data.

2. Numerical simulation methods

The unstructured computational meshes about 902,300 grid points and 1,807,500 grid cells were used through the whole passage of the turbine (D=0.3751 m, H=20 m, P=195846.8 Pa).

The simulation was based on Reynolds averaged Navier-Stokes equations with RNG k–ε turbulence model. The spatial discretization was obtained by using FVM with unstructured grid elements. Second order fully implicit scheme was applied for the time. Second order central scheme was used for the diffusion terms and source terms, and second order upwind scheme was used for the convection terms. Sliding mesh model was used to get a time-accurate solution for the strong rotor/stator interactions. During each time step, SIMPLEC algorithm was used for the solution of discretized equations. After the calculation was converged, time step moved forward, the grid of runner rotated to a new position, and then the calculation of new time step was carried on.

3. Numerical results and discussions

Three different part load operations were calculated: Case I: n_{11}=100 r/min, α=10 mm; Case II: n_{11}=100 r/min, α=12 mm; Case III: n_{11}=95 r/min, α=14 mm. In model turbine experiment, pressure transducers were located at position 1&2 on measurement horizontal section in draft tube (Fig. 1).

The calculated flow field of Case III is shown as an example. In this paper, the pressure represents the gauge pressure relative to the average static pressure at the outlet of the draft tube. As shown in Fig. 2 and Fig. 3, a steep pressure gradient like that of concentric circles is produced at horizontal sections, and the low-pressure zone that represents the vortex rope core bends towards the wall.

A Fast Fourier Transformation (FFT) was carried out for the calculated pressure signals, and the relative amplitude $\Delta P/P$ and dominating frequency f were recorded. Table 1 shows the calculated and

Paper No. : 3-1A-4

experimental results of pressure pulsations at Position 1&2. We can see that the calculation predicted the relative amplitude well, and predicted the dominating frequency approximately.

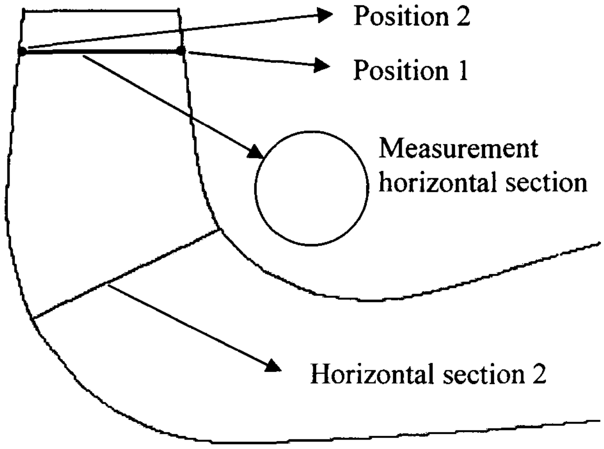

Fig. 1. Measurement Positions in model Francis turbine experiment

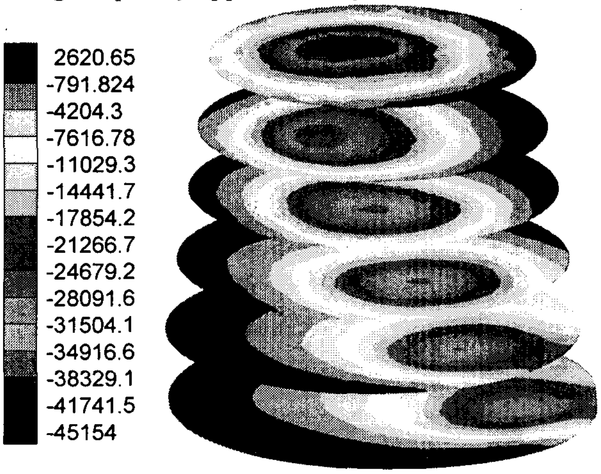

Fig. 2. Pressure distributions at 6 draft tube horizontal sections in Case III (t=0 s)

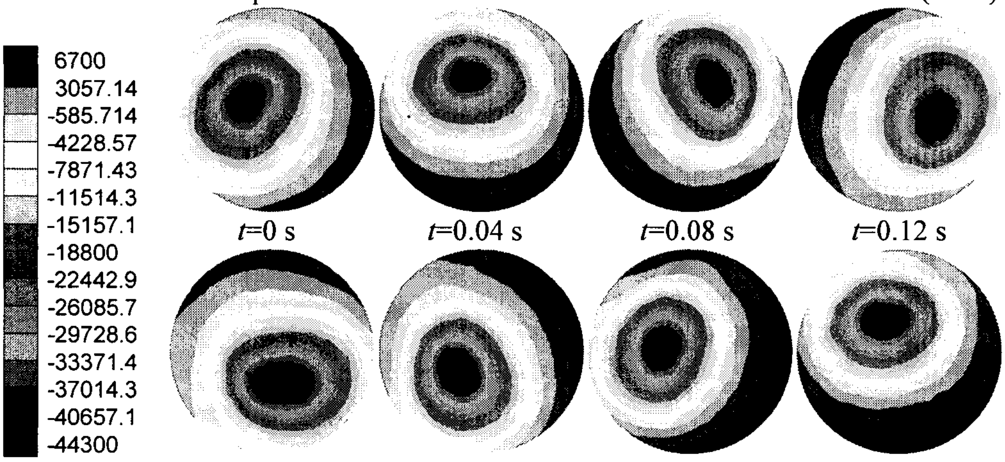

Fig. 3. Pressure distributions at measurment horizontal section in Case III

Table. 1. Calculated and experimental results of pressure pulsations at Position 1&2

Case		Position 1		Position 2	
		Relative amplitude $\Delta P/P$ (%)	Dominating frequency f(Hz)	Relative amplitude $\Delta P/P$ (%)	Dominating frequency f(Hz)
I: n11=100 r/min	Calculation	8.59	25.70	8.16	15.42
α=10 mm	Experiment	10.17	17.38	8.62	34.63
II: n11=100 r/min	Calculation	7.79	17.36	8.32	26.04
α=12 mm	Experiment	9.41	16.50	7.46	20.38
III: n11=95 r/min	Calculation	12.10	7.32	8.53	7.32
α=14 mm	Experiment	8.04	3.75	6.85	3.75

4. Conclusions

Based on Reynolds averaged Navier-Stokes equations and RNG k–ε turbulence model, the sliding mesh model was used to simulate the three-dimensional unsteady flow.

Through numerical simulation, the vortex rope that rotates in draft tube was predicted well. And the calculated results of pressure pulsations at Position 1&2 agree well with the experimental results. So the numerical methods are effective for predicting the unsteady flow in Francis hydraulic turbines.