谈明高,徐 欢,刘厚林,吴贤芳,崔建保.基于CFD的离心泵小流量工况下扬程预测分析[J].农业工程学报,2013,29(5):31-36

基于CFD的离心泵小流量工况下扬程预测分析

Analysis of head prediction of centrifugal pumps at low flow rate based on CFD

投稿时间： 2012-09-09 最后修改时间： 2013-02-27

中文关键词：数值模拟,计算流体力学,离心泵,扬程,性能预测

英文关键词:computer simulation computational fluid dynamics centrifugal pump head performance prediction

基金项目:国家自然科学基金（51179075、51079062）；江苏省自然科学基金（BK2010346）；江苏省教育厅项目（09KJD470002）、江苏大学高级人才科研启动基金（09JDG026）、江苏省研究生创新研究计划（CXLX11_0576）和江苏省优势学科资助项目.

作者	单位
谈明高	江苏大学流体机械及工程技术研究中心，镇江 212013
徐 欢	江苏大学流体机械及工程技术研究中心，镇江 212013
刘厚林	江苏大学流体机械及工程技术研究中心，镇江 212013
吴贤芳	江苏大学流体机械及工程技术研究中心，镇江 212013
崔建保	江苏大学流体机械及工程技术研究中心，镇江 212013

中文摘要:

为了分析和提高小流量工况下离心泵CFD (computational fluid dynamics) 扬程预测的精度，该文对一比转数为64的离心泵多个小流量工况下的内部流动进行了全流场多相位和非定常瞬态数值模拟。重点分析了定常多相位计算中的相位角和非定常计算中的时间步长对扬程预测结果的影响，并对比分析了定常多相位和非定常计算对扬预测的优劣及其原因。研究结果表明，相位角对定常多相位数值计算的扬程预测结果影响很小；时间步长对基于非定常数值计算的扬程预测结果有较大影响。非定常数计算的扬程预测精度远高于定常多相位数值计算的扬程预测精度，其相对误差都在2%以内。随着流量的减小，叶片与蜗壳的动静干涉作用也越来越明显。对小流量工况离心泵扬程进行CFD预测，必须被考虑叶片与隔舌的动静干涉作用。

英文摘要:

In order to analysis and improve the head prediction accuracy of a centrifugal pump at a low flow rate, the inner flow field in a centrifugal pump, whose specific speed is 64, is calculated by steady multi-phase simulation and unsteady numerical simulation respectively. The computational zone includes pump inlet extension, impeller, volute, shroud cham hub chamber, and pump outlet extension. The Reynolds Averaged Navier-Stokes approach is used to solve the control equations. The grid of wear ring is refined and the whole g number independency is checked. The roughness height is considered in the simulation and is set to be 25 μm. The inlet boundary condition is pressure inlet and the outlet boun condition is velocity outlet. The RNG k-ε turbulence model is applied in the simulation. The convergence accuracy is set to be 10-4. The effects of the phase angle in a steady mult phase numerical simulation and the time step in unsteady numerical simulations on the head prediction are analyzed. The comparison between the steady multi-phase simulation a unsteady numerical simulation is discussed and analyzed. The results show that the phase angle has only a little impact on the head prediction, while the time step has a great influence on the head prediction. The prediction results of unsteady numerical simulation are better than that of steady multi-phase simulation, whose errors are all below 2%. As flow rate decreases, the rotor-stator interaction between impeller and volute get more obvious. Therefore, it is necessary to consider the rotor-stator interaction when the pump he predicted by CFD.

查看全文 下载PDF阅读器

关闭