张玉良,肖俊建,崔宝玲,朱祖超.离心泵快速变工况瞬态过程特性模拟[J].农业工程学报,2014,30(11):68-75

离心泵快速变工况瞬态过程特性模拟

Simulation of transient behavior in prototype centrifugal pump during rapid regulating flow rate

投稿时间： 2014-03-21 最后修改时间： 2014-05-08

中文关键词: 离心泵 数值方法 阀门 仿真 调阀 准稳态 瞬态效应

英文关键词:centrifugal pumps numerical methods valves simulation regulating valve quasisteady transient effect

基金项目:浙江省自然科学基金资助项目（LY14E090011, LY12E06002）；国家自然科学基金资助项目（51276172）

作者	单位
张玉良	1. 衢州学院机械工程学院，衢州 324000
肖俊建	1. 衢州学院机械工程学院，衢州 324000
崔宝玲	2. 浙江理工大学浙江省流体传输技术研究重点实验室，杭州 310018
朱祖超	2. 浙江理工大学浙江省流体传输技术研究重点实验室，杭州 310018

中文摘要:

为研究离心泵在不同工况点快速切换过程中的瞬态特性，该文以一台低比转速离心泵为研究对象，对其工况流量突然减小的瞬态过程，分别采用理论分析和数值计算的方式进行了外特性预测和内流场仿真研究。首先基于叶轮机械广义欧拉方程式，对离心泵模型在流量突然减小瞬态过程中的附加理论扬程进行了定量计算与分析。结果表明，同等条件下，变工况过程结束后的稳定流量越小，附加理论扬程越大，瞬态效应愈发明显；同时该瞬态过程后期的瞬态效应比前期更为明显。动静干涉效应对泵出口流动参数产生显著影响，而对泵进口流动参数的影响并不明显；动静干涉效应对小流量工况时各个流动参数的影响将尤为显著。叶片与隔舌相对位置最近时，计算扬程最小；当隔舌位于叶轮流道中间位置稍后时，计算扬程最大。同一个转动周期（T）内，选取叶片转过隔舌后的0.225 T和0.825 T位置进行单次定常计算可取得较高精度的数值预测结果。动静过流部件和粘性效应使得叶轮和蜗壳内的轴向速度分布规律完全相反。瞬态过程中流体加速效应使得瞬态流场演化整体上滞后于准稳态流场。

英文摘要:

Abstract: Transient performance of pumps during transient operating periods, such as startup and the regulating valve, has drawn more and more attention recently due to growing engineering needs. It is impossible for a pump to work at a working point forever, namely that the switching process among different working points must be existent. In order to reveal the transient characteristics of a prototype centrifugal pump in the transient process of the decreasing flow rate by the regulating discharge valve, a low specific-speed centrifugal pump was chosen as the research object to investigate by using the theoretical analysis and numerical simulation, respectively. Through the research, the external performance and internal flow field of the pump model are obtained during the transient operating period. Based on the deduced generalized Euler equation of turbomachinery, the additional transient theory head of the pump model are quantitatively calculated and analyzed in the process of decreasing the flow rate. Results show that under the same conditions, the flow rate after the regulating discharge valve is smaller, the greater the additional theory head, which manifests the transient effect that is more obvious. Meanwhile, the transient effect at the later stage is more remarkable than that of former stage. Subsequently, the RNG k-? turbulence model, sliding mesh, and user defined functions (UDF) are employed to simulate the three-dimensional unsteady viscous incompressible flow in the centrifugal pump during the rapid regulating flow rate. The results show that the rotor-stator interaction plays a dominant role in the fluctuating characteristics of flow parameters at the pump outlet, while the influence on flow parameters at the pump inlet is not very obvious. Moreover, compared with the influence on the condition of the large flow rate, the rotor-stator interaction has a more remarkable effect on the condition of the small flow rate. The predicted pump head is smallest when the relative position between blade and tongue is nearest. Similarly, the predicted head is largest when the tongue is at just after the middle of impeller channel. In a rotational cycle, choosing two relative positions, namely 0.225 T and 0.825 T as the single steady initial phase, would obtain the best numerical prediction accuracy. The different flow components and viscous effect together make the characteristics of the axial velocity distribution in the impeller and the volute opposite. The flow acceleration effect is the most important reason that the flow field evolution in transient calculation lags behind that of the quasisteady calculation as a whole.

查看全文 下载PDF阅读器

关闭