谭 磊,曹树良,王玉明,邴 浩,祝宝山.离心泵叶轮内部流场的数值计算[J].农业工程学报,2012,28(14):47-51

离心泵叶轮内部流场的数值计算

Numerical calculations for internal flow field in centrifugal pump impeller

投稿时间： 2011-10-08 最后修改时间： 2012-06-12

中文关键词：离心泵,叶轮,流场,数值计算

英文关键词:centrifugal pump impellers flow fields numerical calculation

基金项目:国家自然科学基金资助项目（51176088）；中国博士后科学基金面上资助项目（2011M500315）；水沙科学与水利水电工程国家重点实验室开放研究基金资助课题（sklhse-2012-E-02）；

作者	单位
谭 磊	1. 清华大学摩擦学国家重点实验室，北京 100084
曹树良	2. 清华大学水沙科学与水利水电工程国家重点实验室，北京 100084
王玉明	1. 清华大学摩擦学国家重点实验室，北京 100084
邴 浩	2. 清华大学水沙科学与水利水电工程国家重点实验室，北京 100084
祝宝山	2. 清华大学水沙科学与水利水电工程国家重点实验室，北京 100084

中文摘要:

为弥补基于传统一元理论方法中流场计算的不足，该文基于流体的连续方程和运动方程，用Fortran语言编程实现了离心泵叶轮内部流场的数值计算，S1流面上采用有限单元法、S2流面上采用流线曲率法，2类流面迭代计算直至收敛得到离心泵叶轮内部流场分布。对2类相对流面方法计算得到的叶轮内部流场进行了分析。结果表明，叶轮内部相对速度分布合理，叶片头部受冲击作用，压力面和吸力面流速相差较大。叶轮内部压力分布从进口到出口逐渐增大，梯度较小，叶轮做功平稳，进口处压力从后盖板到前盖板逐渐降低。同时考虑流体的连续方程和运动方程后，对比传统一元理论方法的计算结果，计算得到的轴面速度从后盖板到前盖板间各条流线的分布规律相差较大，在叶轮进口段差别最大，具有较强的三维特征，表明本文数值计算结果可更好地反映离心泵叶轮内部的三维真实流动规律。

英文摘要:

To solve the shortages of flow field calculation based on one dimension flow theory, a computer code was developed in Fortran language to calculate the internal flow field in centrifugal pump impellers based on continuity equation and motion equation. The iterative calculation was conducted by applying the finite-element method on the S1 stream surface and the streamline-curvature method on the S2 stream surface. The distributions of pressure and velocity in the impeller were obtained from the numerical calculation as the iteration converged. Results showed that the relative velocity distribution in the impeller was reasonable. The velocities at the blade head between pressure surface and suction surface were different due to the impact action of the fluid on blade head. The pressure in impeller increased gradually from impeller import to export and the pressure gradient was small. The pressure at the impeller inlet decreased from hub to shroud. Compared to the results of the traditional method, the meridional velocity distributions of this method based on both fluid continuity and motion equations were obviously different in each stream line from hub to shroud, with the maximum difference in the impeller inlet. The three-dimensional characteristic of the velocity distribution is distinct, which demonstrates that the numerical results of this method can reflect the three dimensional flow law more accurately.

查看全文 下载PDF阅读器

关闭