流固耦合作用对离心泵内部流场影响的数值计算 Numerical Calculation for Effect of Fluid-structure Interaction on Flow Field in Centrifugal Pump

裴吉 袁寿其 袁建平

江苏大学

关键词： 流固耦合 离心泵 流场 数值计算

摘 要： 采用双向同步求解的方法对离心泵内流场和叶轮结构响应进行联合求解，研究了叶轮流固耦合作用对离心泵内部流场的影响。流场模拟基于Reynolds时均化N-S方程和标准k-ε两方程湍流模型，采用多重坐标系法；结构响应基于弹性体结构动力学方程。并将计算所得的流道网格变形、流场静压和速度的分布以及径向力等结果与非流固耦合计算的流场进行对比分析。分析结果表明，流固耦合作用使得流体和固体区域计算网格发生微小变形，这不仅会改变流体对固体载荷的分布，而且会影响结构对流体的做功作用，从而影响流场的分布；叶片相对隔舌不同位置时，叶轮出口处和蜗壳流道内流场的静压分布变化趋势不同；流场速度变化主要出现在叶片和叶轮出口附近；各时间点上径向力的大小和方向变化较明显。A combined calculation for turbulent flow and structure response of impeller was first established using two-way coupling method to study the effect of FSI of impeller on flow field in centrifugal pump. For the calculation, the flow field is based on Reynolds-averaging N-S equations and standard two equation k-ε turbulent model with the multiple reference frame, and the structure response is based on elastic structural dynamic equation. Mesh deformation, the velocity and static pressure distributions and the radial trust were analyzed, comparing with the calculated results without FSI. The analysis results indicate that, a small mesh deformation occurs in fluid and structure domain which leads to not only different distribution of load that is imposed from flow field to structure but also different impact that is made on fluid by structure, resultingly changing the distribution of flow field; the variations of static pressure distribution are different around impeller outlet and volute flow channel, according to the position of the blade relative to the cut-water; the variations of velocity distribution mostly happen around blades and impeller outlet; the magnitude and direction of radial trust have significant changes.

查看全文（请使用Adobe Acrobat 6.0版本浏览） 返回首页 引用本文