湍流模型在双吸离心泵数值模拟中的适用性分析 Applicability of Turbulence Models in Numerical Simulation of Double Suction Centrifugal Pump

刘梅清 李秋玮 白耀华 刘志勇 林琦

武汉大学

关键词： 离心泵 湍流模型 CFD 数值模拟

摘 要： 以双吸离心泵为研究对象，在Fluent软件中采用标准k-ε模型、RNG k-ε 模型、Realizable k-ε模型和雷诺应力模型4种湍流模型在相同网格条件下进行三维、稳态数值模拟，计算其在不同工况下的扬程及水力效率；然后以RNG k-ε模型计算结果作为初值，用大涡模拟方法进行三维、非稳态数值模拟，计算叶轮旋转一周的平均扬程及平均水力效率，与真机数据进行对比分析。计算结果表明：Fluent软件在稳态计算的情况下，所采用4种模型均能够对双吸离心泵进行稳态数值模拟，且模拟的精度有差别，但总体与模型试验结果误差较大。使用稳态计算结果作为初值，用大涡模拟方法进行非稳态数值模拟可以得到更为精确的结果，但大大增加了计算工作量。 In order to analyze the effects of turbulence models on the numerical solution of double suction centrifugal pump through the use of Fluent software, a pump applied in domestic projects was chosen as a study object. 3-D steady numerical simulations were conducted by four different turbulence models, namely, the standard k-ε model, RNG k-ε model, realizable k-ε model and Reynolds stress equation. The results of steady numerical simulations were then used as initial values, and 3-D unsteady numerical simulations were conducted by the large eddy simulation (LES) method. Comparison of the numerical results with the experimental ones showed that in the situation of steady numerical simulations, four turbulence models could all be used in numerical simulation of double suction centrifugal pump, but the precisions of results were different. From this perspective, LES method will obtain much more accurate results.

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