

液力耦合器三维瞬态流场大涡模拟与特性预测

褚亚旭 刘春宝 马文星

北华大学

关键词: 液力耦合器 大涡模拟 特性预测

摘要: 采用大涡模拟、流动控制方程耦合求解法及多可动区域计算的滑动网格法,对液力耦合器内部瞬态三维流动控制方程组进行了耦合求解。对三维流场模拟结果进行深入分析,以进一步了解耦合器内部流动规律,优化设计其结构。同时,根据三维流场数值解计算了各个工况下液力耦合器叶轮转矩,进而预测其性能,将性能预测结果与实验结果进行比较,二者误差在7%以内,验证了大涡模拟方法及特性预测的准确,说明采用的液力耦合器流场的模拟方法具有良好的工程应用价值。 The transient control equations of 3-D flow in hydrodynamic coupling were coupled calculated by using the methods of large eddy simulation (LES), flow control equation coupled solution, and sliding mesh method for multi-flow regions in CFD software. In order to learn the flow laws of flow field deeply and optimize the structure, the numerical solution of the 3-D flow field has been analyzed. The torque in each impeller of hydrodynamic coupling was figured out based on the numerical solution of the 3-D flow field. Then the characteristic of hydrodynamic coupling was predicted. The prediction characteristic has been compared with the experimental result; it is found that the error between them is less than 7%, which could prove the effectiveness of the numerical simulation method and characteristic prediction. Then the conclusion has been drawn that the numerical simulation method for the flow field of hydrodynamic coupling has applicable value in engineering.

[查看全文 \(请使用Adobe Acrobat 6.0版本浏览\)](#) [返回首页](#)

[引用本文](#)