

高比转数轴流泵水力模型设计

杨敬江 关醒凡

江苏大学

关键词: 轴流泵 水力模型 设计 CFD

摘要: 与传统升力法和圆弧法设计不同,以流线法设计高比转数轴流泵水力模型,提出了叶轮出口线性修正的环量分布规律,冲角以轮毂到轮缘增加的新方式进行选择,取值范围为 $0^{\circ} \sim 3^{\circ}$,比转数越大,冲角取值越小。应用CFD技术,数值模拟了泵段水力模型全流道流动情况,优化了水力设计参数选择规律,大大减少了水力模型制作和模型试验的次数,研制出了系列轴流泵水力模型,在天津进行的水利部南水北调工程水泵模型同台测试中,该系列模型具有较高的效率和较宽的高效区特性。Different from traditional lifting design method and circular arc design method, streamline design method was employed to design high specific speed model axial-flow pump. The design method included linear amendment of exit circular and increased incidence angle from hub to tip of impeller. Ranges of incidence angle are from 0° to 3° , and the incidence angle becomes lower with higher specific speed. CFD technique was applied to simulate whole flow fields of model pump. Numerical results optimized the selection rules of hydraulic design parameters, and decreased more work on models made and models experiments. Then a series of model axial-flow pumps developed. These models have higher efficiency and broader high efficiency zones from the experimental results of countrywide model pump experiments in the same testing stand organized by Ministry of Water Resources in Tianjin city.

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

[引用本文](#)