

基于ANSYS的标准泵蜗壳强度分析

王洋 刘哲

江苏大学

关键词: 离心泵 蜗壳 强度 有限元分析 数值模拟

摘要: 为解决国产标准泵蜗壳壁厚较厚问题,用传统公式计算S50-160/75型标准泵壁厚,并进行修正。采用Fluent模拟小流量与设计工况下蜗壳内沿液流方向的压力分布,分别为由大到小再变大及逐渐增大的趋势。采用ANSYS计算2种工况下蜗壳所受应力与应变情况,得到易破坏部位分别为蜗壳隔舌与环形部分最大尺寸轴面处。对蜗壳强度进行校核及有限元分析,表明修正后的蜗壳壁厚满足强度要求。并通过水力性能试验,得到计算结果与试验数据相吻合。

The thickness of the S50-160/75 standard pump volute calculated by the traditional formula was amended to solve the volute wall of the domestic standard pump. The pressure distributions in the volute along the flowing were simulated by using Fluent software, and the trends were from rising to declining to rising again in low flux condition and rising gradually in rated condition. The stresses and strains have been calculated approximately in different conditions with ANSYS software, it concluded that the throat section and the largest ring of the axial region are the most easily damaged and the most dangerous parts. Strength check on the volute and finite element analysis showed that the amended thickness of the volute met the strength requirements. And the hydraulic performance tests of the pump proved that the calculated results are consistent with the actual results.

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

[引用本文](#)