

阀芯运动状态滑阀内部流场的可视化分析

赵蕾 陈青 权龙

太原理工大学

关键词: 滑阀 可视化分析 液动力 径向力 流量系数

摘要: 应用CFD软件Fluent,以全周开口滑阀为例,对滑阀阀芯运动过程内部流场变化特性进行了可视化分析。计算发现当阀处于小开口、大流量、阀芯高速运动时,瞬态液动力数值较大,设计阀时必须加以考虑。由于流道和阀体的不对称,在阀杆上还作用有不对称的径向力,会引起阀芯卡紧,很难用结构设计的方法加以平衡。通过对滑阀阀静态流量系数进行计算,发现随着阀口开度和阀芯运动状态的不同,流量系数变化较大。The flow field inside the spool valve during spool moving was simulated and analyzed using the CFD software Fluent. The simulation results show that the transient flow force is large when the valve is in small throttle with large flow and high moving speed, which must be considered during the valve design. By the analysis of the hydraulic force act on the valve-ole, there is an additional radial force that causes valve element lockup existed. This force is difficult to be balanced with only structure method. By computing the discharge coefficient valve in static and dynamic condition, the results show that the discharge coefficient changes greatly with the spool stroke and moving state.

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