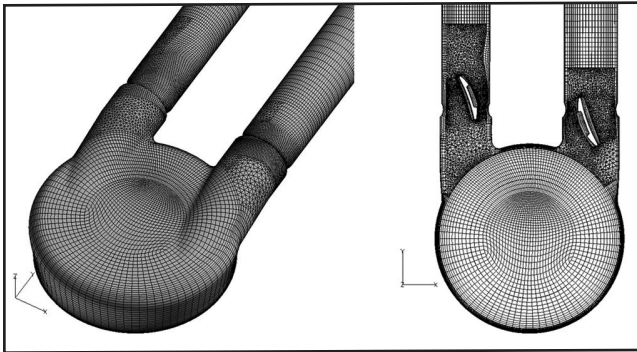


GridgenApp

A Unique Gridgen® Application



AcuSolve and Gridgen Get Hearts Pumping



Unstructured mesh used for unsteady analysis.

Researchers at Pennsylvania State University are using computational fluid dynamics (CFD) to analyze and help redesign an artificial heart. The 50cc Penn State Left Ventricular Assist Device (LVAD) is a positive-displacement type pump containing two valves and one pusher-plate. It was scaled from the larger, 70cc LionHeart, which is intended for use in patients weighing more than 70 kg.

The objectives of the current work are (1) development of a CFD methodology for modeling piston motion and valve closure, (2) model validation through comparison with previously obtained particle image velocimetry (PIV) data, and (3) advancement of understanding of flow physics.

Two separate computational analyses were undertaken during the course of the current work, with the goal of building towards a fully dynamic analysis of a pulsing cycle. The first series of calculations investigated the flow through the LVAD when a steady flow was forced through the device by means of an external pump. Here, the piston was locked in the fully expanded position and the valves remained fully opened.

The second set of computations performed an unsteady analysis of the full pumping cycle of the LVAD. The pulsatile computations required modeling of both the valve closure (to maintain properly directed bulk flow) and the pusher-plate motion (to drive the flow field).

The pulsatile analysis used the commercial flow code AcuSolve from ACUSIM Software, Inc. to solve the time accurate Navier-Stokes equations. AcuSolve, a finite element CFD solver based on a Galerkin/Least-Squares formulation, is second-order accurate in space and time and supports a variety of element types. It contains a native fluid-structure interaction capability along with a discontinuous Galerkin method for sliding mesh simulations.

Gridgen was used to create all of the CFD meshes used in this work. Meshes for both the steady and unsteady simulations used a near wall spacing of 0.001 inches. The computational meshes were simplified to a degree in the vicinity of the valves. The steady-flow simulations used a structured overset mesh to discretize the complex three-dimensional geometry.

The pulsatile-flow simulations used an unstructured mesh containing tetrahedral, brick and pyramid elements. Brick elements were used to obtain adequate near wall resolution on the wetted surfaces. To expedite the initial time accurate computations, the pulsatile-flow mesh was coarsened relative to the steady flow mesh. This mesh contained approximately 547,000 grid points.

There was very good qualitative agreement between the experiments and computations. The primary difference occurred in the minor orifice jet, potentially due to the valve support strut being neglected in the computational model.

Despite the approximations made in the current work, the overall agreement with in vitro experimental data was high. This illustrates that CFD methodology is useful for the stated objectives and provides a starting point for higher fidelity simulations in the future accounting for low-Reynolds turbulence modeling, non-Newtonian blood rheology and fluid-structure interaction.

Article by Richard B. Medvitz and Eric Paterson
The Pennsylvania State University