

# Computer Modeling Provides Accurate Calculation of Pump Performance

**By: Christopher J. Matice, Timothy J. Fry, and Eric M. Luther**  
SES Process Technology Group,  
A Division of Stress Engineering Services, Inc.  
Cincinnati, Ohio

## Introduction

Computational modeling has revolutionized the design of process equipment. By enabling the designer to test variations in a virtual environment, the computer speeds concept development, reduces prototyping, and practically eliminates cut-and-try engineering. Modeling allows the designer to quickly review a range of important features of a new design, including overall performance and identification of regions of stagnant flow, high wear and cavitation.

Until recently the power of computational simulation has not been harnessed as part of the pump design process because of its uncertain accuracy. This paper describes how computational fluid dynamics (CFD) technology has been applied to accurately simulate flow and pressure in a small regenerative turbine

pump. This technology can be applied with similar levels of accuracy to a wide variety of turbomachines and other types of process equipment.

## Product Description

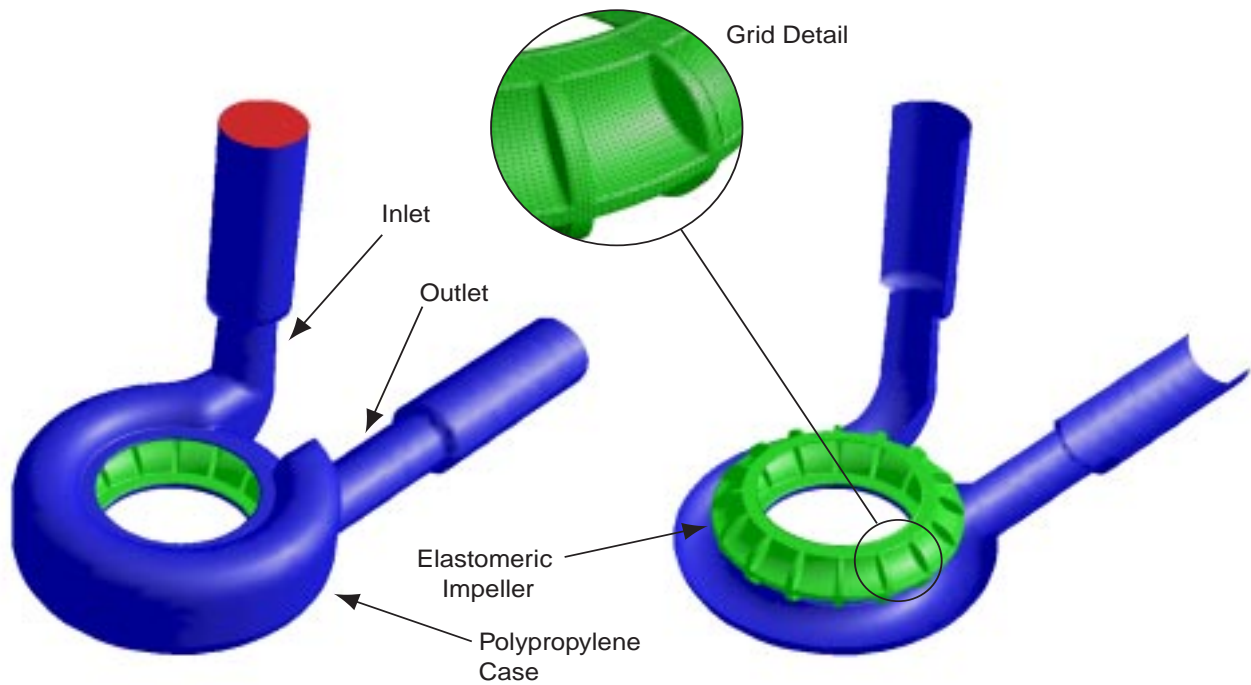
In order to demonstrate the power of computational modeling of pumps, a small regenerative turbine pump has been modeled. This pump is shown in Figures 1 and 2. The pump consists of a polypropylene housing and an elastomeric impeller. The housing is of uniform diameter without a volute. This pump is used in domestic washing machines and is designed to robustly handle a variety of solids without maintenance. This pump is referred to as the Poly-Pump.<sup>1</sup>



**Figure 1**  
**Poly-Pump Installed for Testing**



**Figure 2**  
**Section View of the Poly-Pump**



**Figure 3**  
**Flow Model of the Poly-Pump**

### Model Development

Application of a computational model to the development of process equipment requires two major steps. The first step is the development of a computational grid describing the boundaries of the equipment and the flow passages within the equipment. The surfaces forming the boundaries of the Poly-Pump model are shown in Figure 3. Grid development can be based on a geometrical description of the equipment that is input by the flow analyst or can be taken from CAD files provided by the designer. Typically this is done in a program such as AutoCAD, Pro/Engineer, or SDRC I-DEAS. The flow analyst then divides the equipment into elements as shown in Figure 3.

Grid development is the major effort in the use of a CFD model. In defining the model a trade-off is required between the needed level of accuracy and the desired speed of solution. Typical CFD models of process equipment require 50,000 to 500,000 computational elements or “cells.” The number of cells in the model is typically dictated by the smallest feature which must be simulated. For the Poly-Pump,

the small space between the O.D. of the impeller and I.D. of the case required a large number of small cells; the total cell count in this model was 400,000. Models of this size require a workstation computer for rapid solution; however, smaller models will easily run on a high-end PC.

The second step in the CFD analysis process is the calculation of the velocity and pressure in each computational cell in the model. This is accomplished using a physics-based math model of the flow (and possibly heat transfer) in the equipment. The techniques for conducting these calculations were largely developed in the 1960s and 1970s for the aerospace, defense, and nuclear industries. However, given the limits on computer speed, the technology was not commercially viable for application to most industrial problems. In the process industries, CFD technology came of age in the early 1990s with the development of inexpensive high-speed computers. There are currently a number of commercially available CFD programs on the market, including Fluent, CFX, and FLOW-3D. All of these include solvers for fluid flow, heat transfer, and the tracking of



**Figure 4**  
**Velocity Vectors on the Center-Line Plane**  
**of the Poly-Pump Impeller**

particles through the equipment. For the Poly-Pump we used Fluent to solve the fluid flow equations.

**Results**

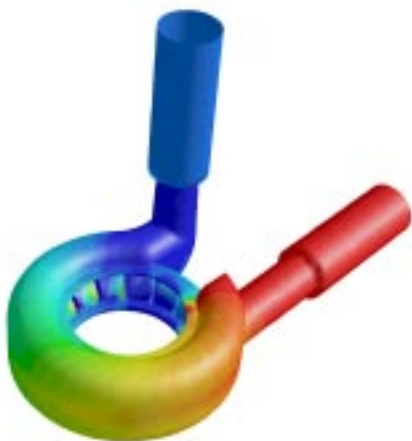
The results of the CFD simulation of flow through the Poly-Pump and are shown in Figures 4-6. Figure 4 shows a plot of velocity vectors on the center plane of the impeller. This analysis was conducted by rotating

the impeller at 400 rpm, imposing a fixed flow through the case, and computing the pressure difference from inlet to outlet. This approach is more stable numerically than the alternative method of specifying the pressure difference over the pump and computing the flow rate.

Pressure contours computed on the inside of the Poly-Pump case are shown in Figure 5 and contours on the impeller are shown in Figure 6. By isolating region of low pressure (blue contours), plots such as these can be used to predict the onset of cavitation. Detailed modeling can be used to predict the creation, convection, and collapse of the cavitation bubbles so as to map out the size and extent of the cavitating region.

In addition to computing flow and pressure, the CFD analysis also provides the opportunity to trace particles through the pump. This is shown in Figure 7. Here the particles are shown as ribbons tracing their path from inlet to the outlet. This type of display is useful in determining areas of low flow in the pump and in determining wear patterns associated with various particles sizes.

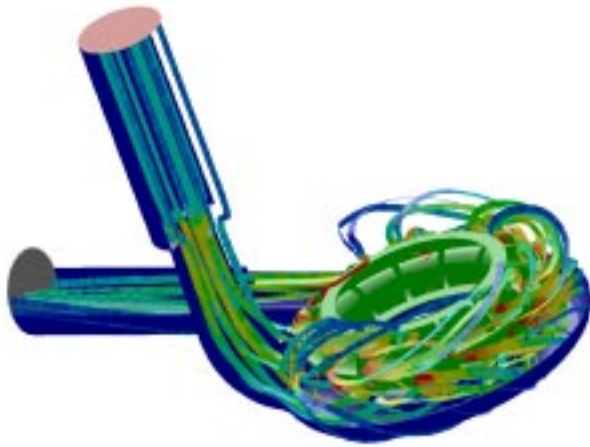
The most important result in the analysis of a Poly-Pump is the determination of the pump curve. The



**Figure 5**  
**Pressure Contours on the Inside Surface**  
**of the Poly-Pump Case**

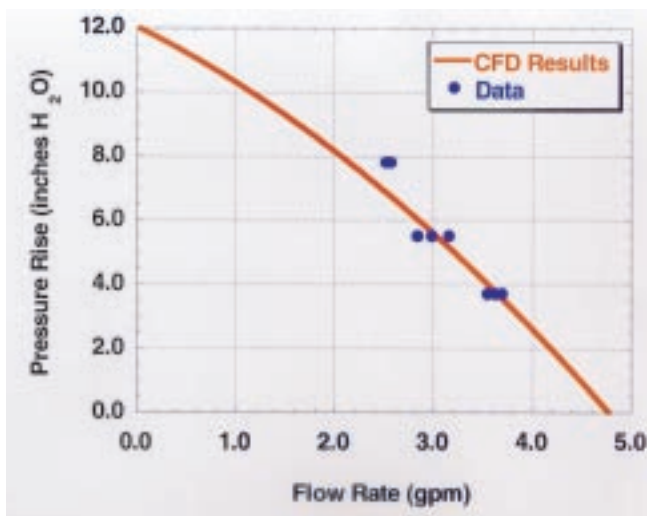


**Figure 6**  
**Pressure Contours on the Surface**  
**of the Poly-Pump Impeller**



**Figure 7**  
**Ribbons Showing Paths Taken by**  
**Particles through the Poly-Pump**

pump curve for the Poly-Pump is presented in Figure 8. This figure shows the pump curve computed using the CFD model as a solid red line and results from the experimental setup shown in Figure 1 as blue data points. This plot shows that the level of agreement between the computed and experimental results is quite good, within five to ten percent in most cases. This is typical of the accuracy provided by modern CFD analysis. In most cases a knowledgeable



**Figure 8**  
**Comparison of CFD and Experimental**  
**Results for the Poly-Pump**

analyst can obtain accuracy of 10 percent or better.

The keys to attaining high accuracy in CFD modeling of process equipment are the use of proper physical models, adequate grid refinement, and interpretation of the results by a knowledgeable analyst. Most flow problems experienced in process equipment involve turbulent flow. The modeling of turbulence is still an area of active research and no single set of equations has yet been found that will accurately simulate the effect of turbulence in all cases. Therefore, selection of a turbulence model and understanding of its limitations are key to the successful simulation of most types of process equipment. In addition, experience in non-computational design and in the testing of process equipment is invaluable as it gives the analyst a frame of reference in which to interpret the numerical results from the CFD simulation.

### Conclusion

Computational fluid dynamics provides an important new tool to the pump designer. This article has shown one case in which the pump curve generated through use of a CFD simulation closely matches that obtained experimentally. CFD can be used to dramatically reduce the number of iterations required in the typical design cycle, providing substantial savings in both the time required to fabricate and test new designs and in the cost of prototype tooling.

1. Poly-Pump is a trademark of Maytag Corporation.

*Christopher Matice, Ph.D., P.E. is a Principal with the engineering services firm of Stress Engineering Services, Inc. Dr. Matice heads the SES Process Technology Group which performs design and analysis of process equipment for clients worldwide.*

*Timothy Fry, Ph.D. is a Senior Associate with Stress Engineering Service, Inc., specializing in fluid flow testing and simulation.*

*Eric Luther, P.E. is an Associate with Stress Engineering Services, Inc. and specializes in component design applications of finite element analysis.*