Digital Design for Centrifugal Fans

John Abbitt, Sam Lowry
Department of Mechanical & Aerospace Engineering, University of Florida / Simerics, Inc.

Abstract

An objective in the curriculum in the Department of Mechanical & Aerospace Engineering at the University of Florida (UF) is for students to learn the fundamentals of the fluid mechanics of centrifugal pump performance in a course called EML4304C Thermo-Fluids Design and Lab. EML4304C is a senior level advanced thermodynamics and fluid mechanics laboratory class with a focus on design for turbomachinery, pipe flow, compressible flow, and refrigeration cycles. As part of the process, students learn to design an impeller for a centrifugal fan to achieve a specified performance curve, i.e., head vs. flow rate. For years, students at UF have used traditional techniques that involve Euler's Turbomachine Equation and velocity triangles to predict fan performance. Results have shown that these predictions are less than satisfactory. As a result, UF began to explore the possibility of using a digital design process that uses CAD, computational fluid dynamics (CFD), and 3-D printing to enhance the ability to predict performance.

Keywords

Turbomachinery, CFD, 3-D printing, Centrifugal Fan

Designing the Prototype Impeller using Traditional Techniques

A performance curve, i.e., head vs. flow rate, is specified by the instructor for each design team. There are about thirteen to sixteen teams per semester. The design process for the students begins by applying the Euler Turbomachine Equation and using velocity triangles to determine the inlet and exit blade angles for the impeller, and then adding the blades to a template of the impeller hub provided by the instructor in the 3-D computer aided design (CAD) program Solidworks® [1] as shown in Figure 1a. Ultimately, the final product will be installed into the existing housing shown in Figure 1b, but there is much analysis left to be performed before the final product is built. At this point in the design process, only classic design guidelines have been applied.

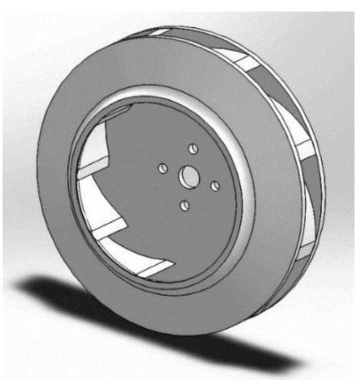

Figure 1a. Solidworks drawing of an impeller

Figure 1b. Blower Housing in test rig

Designing the Prototype Impeller via CFD

Once an initial design is created using traditional techniques, the digital design begins by numerically analysing the performance of the impeller. The purpose of the numerical CFD simulation is to iterate through different blade angles, shapes and lengths in a virtual test simulation to generate a curve for head rise for a range of flow rates which is closest to the target head vs. flowrate performance curve.

First, a virtual model of the fluid system to be analysed is created using Solidworks. The inlet and outlet to the system as well as the volute were modelled in Solidworks using measurements of the actual system. This virtual model is provided to the students who then add their impeller design to the assembly. In Solidworks, the students obtain the fluid volume of the system, and save it in stereo lithographical (STL) format. The STL format includes the triangulation of the surfaces of the inlet, volute, and impeller down to a level that captures the surface details. The STL file is imported into PumpLinx where the surfaces are split into their relevant features, and the volume meshes are created from these surfaces. This splitting and meshing is automated, typically requiring less than 10 minutes and can capture details down to the order of microns.

Having created the mesh, the students then input the properties, boundary conditions, and operating parameters corresponding to the physical test. For this experiment, air is assumed as an ideal gas, and the pressure far away from the inlet nozzle is atmospheric, and the pressure at the exit of the apparatus is also atmospheric in accordance with subsonic nozzle flow theory. Turbulence is modelled using the standard K-epsilon model[2]. The flow is regulated with ten different smoothly contoured nozzles with cross-sectional areas corresponding to 5%, 10%, 15%, 20%, 25%, 30%, 40%, 50%, and 70% of the fully open flow at the outlet. The system is tested with the nine nozzles plus a shut-off plate (for no flow) to generate a head rise versus flowrate graph of the performance curve. The geometries of the nozzles, inlet, volute, and outlet are all meshed and included as part of the numerical model. Specifying the geometry, inlet and exit pressures, and the rotational speed of the impeller allows PumpLinx to determine both the flow rate and the head rise. However, for comparison with the experimental data, flow rate is calculated using the predicted pressures at the nozzle inlet and atmospheric pressure and inserting these into Bernoulli's equation with an appropriate nozzle coefficient.

Selection of Numerical Parameters

In addition to the operating conditions, there are several numerical parameters, such as higher order schemes and the choice of transient vs. steady-state analysis that need to be considered. While not intended as a course on CFD, the advantage of higher order numerical schemes in resolving the flow field is discussed and demonstrated during the project by running with and without. Perhaps more importantly, the advantage of running full three dimensional transient simulations which can account for the pulsations created by blade passage is stressed and demonstrated by comparison with the experimental data and the more common but less accurate steady-state analysis. Evidence of this event can be seen through a cross-sectional view of the impeller while a transient solution is running.

Once created, the numerical model can be run in the same manner as a physical hardware test, generating data for different operating conditions. Specifically, the students can vary the rpm and flow geometry to predict the corresponding flow field, flow rates, pressures, torques, powers, and loads throughout the blower. The students use the results to test different iterations of their designs, all with the same number of blades and thicknesses, but with varied blade angles, shapes and blade lengths. They also investigate the generated performance curve as well as the flow field in the impeller to determine how well each design iteration performs and to gain insight into what the next design should improve upon. Once a final design has been chosen, created, and tested the CFD results can be compared directly to experimental data

The predicted pressure field for the entire fluid volume for a selected design is shown in Figure 2a, and a cross-sectional view of the corresponding flow field in the housing is show in Figure 2b. This run took 35 minutes to reach a steady-state solution on a Dell Precision T5810 computer with an Intel® Xeon® CPU E5-1620 v3@3.5 GHz. Pressure is indicated by color, and velocity vectors are shown which are proportional in size to the magnitude of the velocity. Cross-sectional views like this are used to guide the design of the impeller blade shapes. The shape of the blade is adjusted to minimize flow separation.

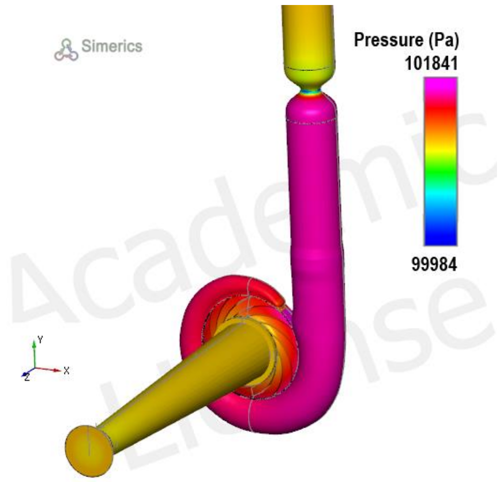

Figure 2a. 3-D pressure distribution of the flow field predicted by the simulation

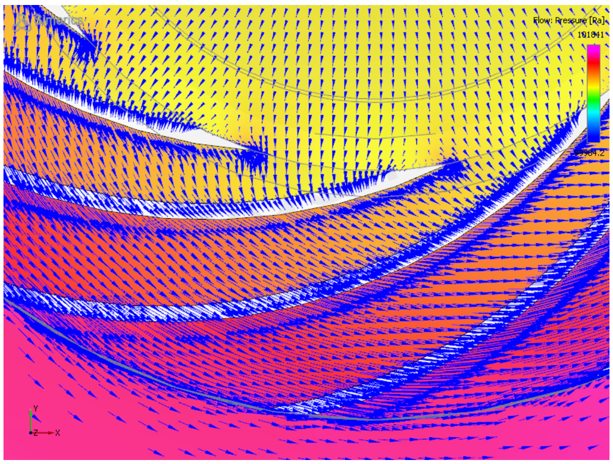

Figure 2b. Close-up of the velocity vectors on a plane passing through the fan blades. The goal here was to reduce separation in the flow.

3D Printing

Once a prototype design has been decided based on the CFD results, the Solidworks model is sent to either a Stratasys Dimension 1200es or a Fortus 360mc 3-D printer which is used to create a working model of the impeller made of either ABS plastic or polycarbonate. An example of an ABS plastic model can be seen in Figure 3. The surface finish is somewhat rough. It is unknown at this time what the effect of the roughness is, but initial testing indicates that this effect is not significant. This printing process takes between 30 and 35 hours per impeller, depending upon the number and shape of blades.

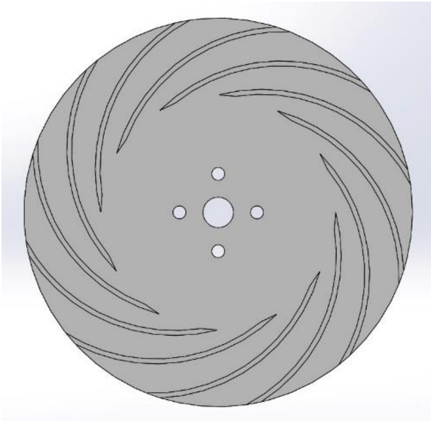

Figure 3a. Cross section of impeller used in experiment.

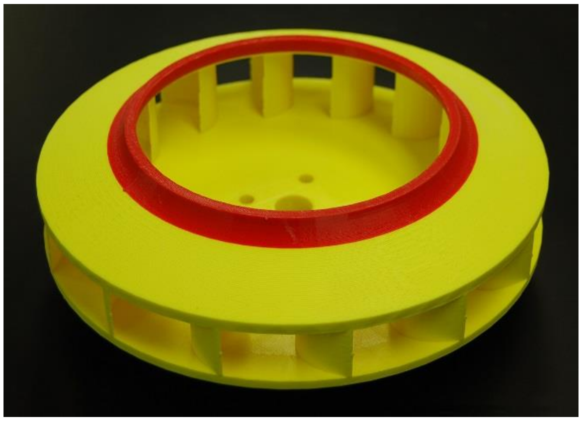

Figure 3b. Impeller prototype made of ABS plastic, designed and printed by one of the student teams.

Experimental Testing

Once a working model of the impeller has been printed, i.e., fabricated, it is subsequently mounted to the fan housing and tested in the UF Undergraduate Fan Performance test bed. The apparatus is powered by a 1-hp electric motor connected to a belt which turns the impeller. A motor controller which can maintain rotational speed to within 0.1% is used to set the rpm. The flow rate is controlled by means of a set of smoothly contoured control nozzles of different cross-sectional areas. An example of a control nozzle is shown in Figure 4. The volumetric flowrate through the apparatus is computed by measuring the difference in static pressure in the calibrated inlet nozzle and atmospheric pressure, and then computing the velocity based on the Bernoulli equation with an appropriate nozzle coefficient. The power of the motor is measured using a dynamometer that is calibrated prior to testing. Static pressures are measured at several locations within the blower using a bank of manometers. Manometer fluid with a nominal specific gravity of 0.827 is used in the manometer. The actual specific gravity of the fluid is measured by the students. Ambient conditions including temperature, atmospheric pressure, and relative humidity are measured and included in the experimental data reduction.

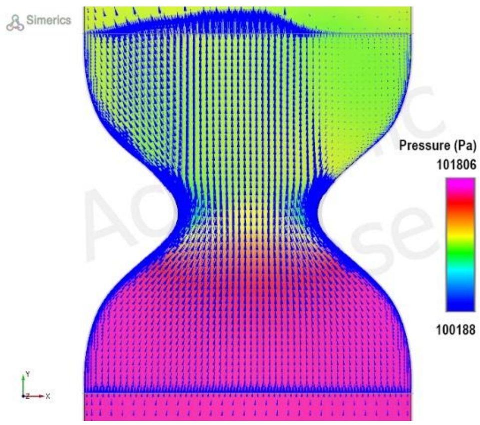

Figure 4. PumpLinx simulation of the control nozzle with 10% opening.

Experimental Results

Experiments were conducted at two different rpms, 1500 and 2500 rpm, and head rise was obtained for flowrates beginning at no flow, i.e., shutoff conditions, and then in increasing areas for the outlet flow until 100 % flow area was achieved. The results are shown in Figure 5. This impeller was designed to have a negative slope throughout its operating range. Using just the Euler Turbomachine equation, a linear slope of -546 m/(m^3/s) is predicted at 2500 rpm, but the actual curve deviates from that prediction. The Euler Equation predicts a maximum head of 111 m at 2500 rpm at shutoff conditions while 43 m was actually obtained. At 1500 rpm, the Euler Equation predicts a slope of -327 m/(m^3/s), and a maximum head of 40 m at shutoff conditions while 16 m at shutoff was actually obtained in the experiment.

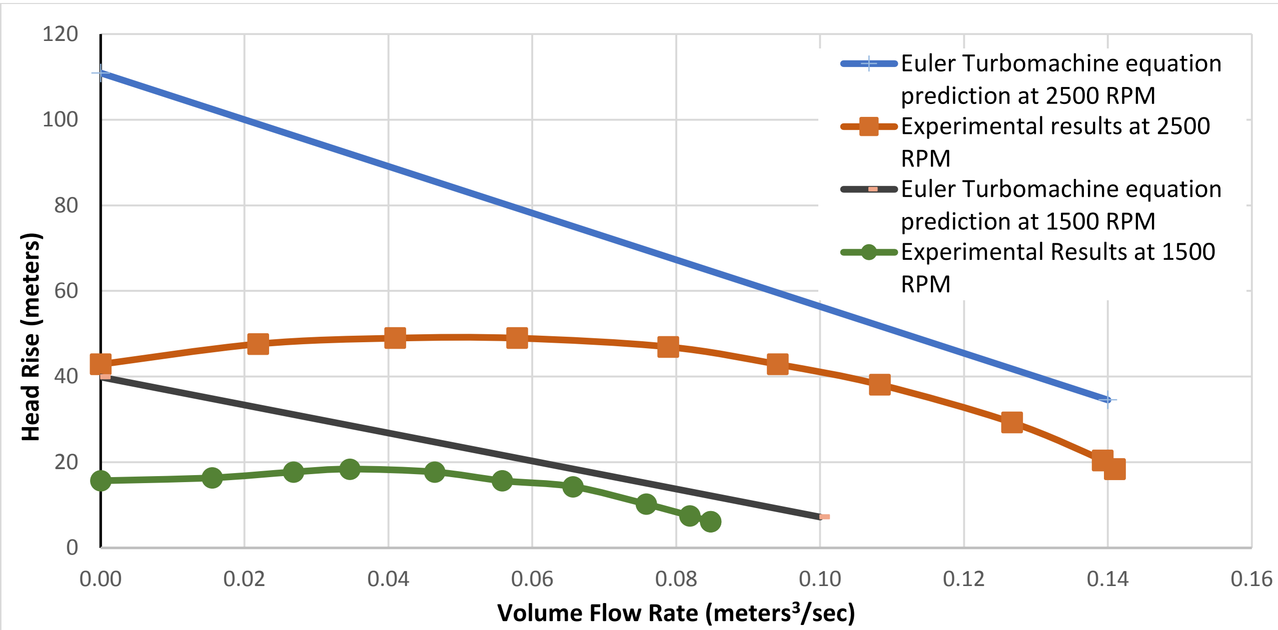

Figure 5. Experimental results at 1500 and 2500 rpm. The curve predicted by the Euler Turbomachine Equation is also shown. The experimental curves at each rpm deviate significantly from those predicted by the Euler Equation. The Euler equation assumes an infinite number of blades with zero thickness to guide the flow perfectly. Since the impeller blades have a finite thickness, and there are gaps between the blades where the flow is not controlled, there are losses called circulatory losses (due to finite number of blades) and passage losses (due to friction between the fluid and the blade surface). There are also losses due to what is called "slip" which is a result of a small circulation of flow from the exit of the impeller to the entrance in the gap between the wall of the volute and the outer surface of the impeller.

Simulations vs. Experiments

The results of the PumpLinx predictions for head vs. flowrate are compared with the experimental counterpart in Figure 6. PumpLinx offers several options which include obtaining a "steady state" solution which can be computed relatively quickly, or a "transient" solution with a 2nd order differencing which is more computationally intensive, but more accurate. The results shown here are for transient solutions. As can be seen in the figure, the CFD results and experimental results are quite close over a large range of flowrates. There is a deviation at the shutoff head of 8.03 m at 2500 rpm which corresponds to a pressure difference of 0.013 psi. This discrepancy is under investigation at this time, but it is suspected to be due to a temperature effect which would not exist under any other flow condition. At the highest flowrate for which calculations were performed, the deviation is less than a meter, and is essentially identical. Over the range of flowrates from 0.02 m^3/s to 0.11 m^3/s which includes 65% of the obtainable range of flowrates in the experimental apparatus, the maximum difference between the experimental results and the PumpLinx results is also essentially identical, as the difference is within the experimental error. The difference between the experimental results and PumpLinx predictions is essentially negligible for all the tests made at 1500 rpm.

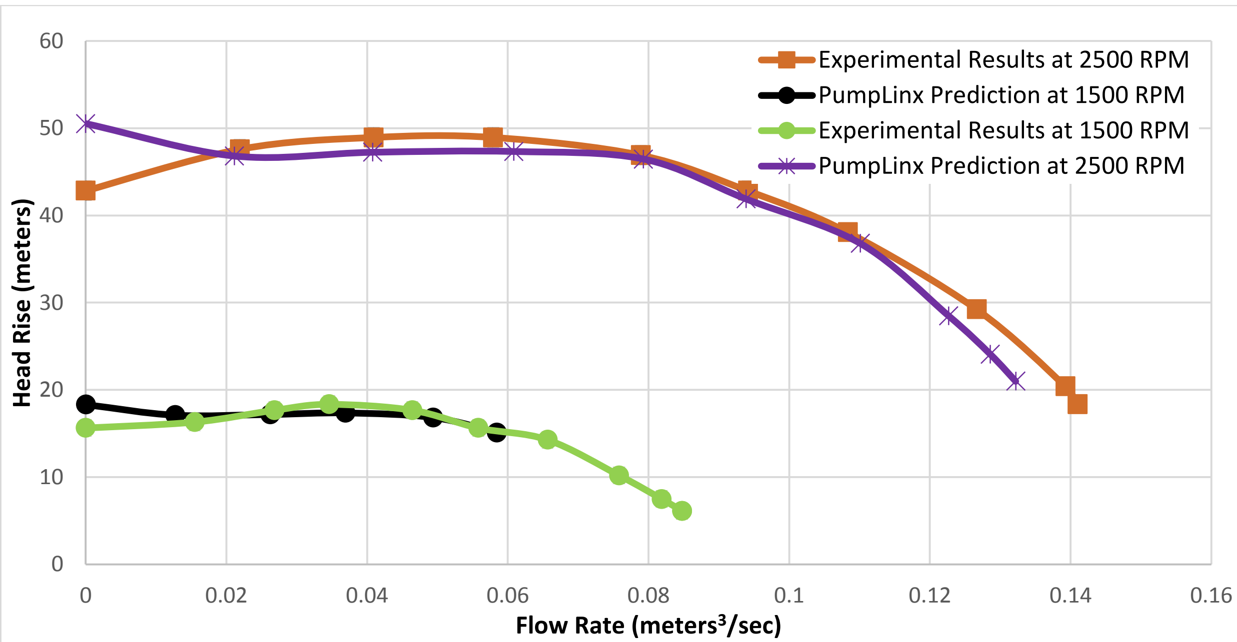

Figure 6. A comparison of the head rise predicted by PumpLinx with the experimental results are shown here. For comparison, one meter of head corresponds to 0.0017 psi. Over the range of flowrates from 0.02m^3/s to 0.11 m^3/s at 2500 RPM, the predictions from PumpLinx are essentially identical to the experimental results. Over all the range of flowrates at 1500 RPM, the predictions from PumpLinx are essentially identical to the experimental results.

What We have Learned

The original premise of the course Thermo-Fluids Design and Lab was a simple introduction to predicting fan performance using the Euler Turbomachine equation and velocity diagrams. This evolved to designing impeller models for a specified performance curve using traditional methods and then building physical models with a 3-D printer and testing these models in the lab facility. Finally, computational fluid dynamics was implemented using the PumpLinx software from Simerics, Inc. Prior to the application of CFD, the comparison between predicted performance and actual performance varied by as much as 100%. In contrast, the prediction of the actual performance curve provided by PumpLinx has turned out to be very accurate over the majority of the available range of flowrates. However, the objective of the design project was not to just predict performance, but to design a fan to achieve a specified performance curve, i.e., head vs. flow rate. At this point, it has been demonstrated that it is possible to use CFD to accurately predict the performance of a design without having to resort to hardware testing. The project has progressed to a point now where the next step will be to actually achieve the specified performance curve. Getting to this point has required extensive work in refining the fluid model which includes obtaining accurate dimensions of the apparatus and designing a flow control system as known quantities for input into the CFD models. The next step will require students to create multiple impellers and systematically adjust the design using CFD until they can arrive at a satisfactory configuration. Although the material is new to the students at the beginning of each semester, it appears to be practical to introduce traditional theory, numerical simulation, fabrication, and testing in one semester and obtain satisfactory results. The partnership between students in the Department of Mechanical & Aerospace Engineering at the University of Florida and Simerics, Inc. has been quite positive and has resulted in a comprehensive study of the fluid mechanics of turbomachinery, and introduces approximately 400 students per year to the most modern analytical tools in the industry.

Figure 7. Dr. Abbitt examining the impellers.

References

[1] www.solidworks.com

[2] Ding, H., Visser, F.C., Jiang, Y. and Furmanczyk, M. 2011 “Demonstration and validation of a 3D CFD simulation tool predicting pump performance and cavitation for industrial applications", J. Fluids Eng. – Trans ASME, 133(1), 011101

[3] Munson, Bruce R., Okiishi, Theodore H., Huebsch, Wase W., Rothmayer, Alric P., Fundamentals of Fluid Mechanics, 7th Edition, 2013, John Wiley & Sons, Inc.

[4] Tuzson, John, Centrifugal Pump Design, 2000, John Wiley & Sons, Inc.

[5] Cengal, Yunus A., Cimbala, John M., Fluid Mechanics, 1st Edition, 2006, McGraw-Hill.

John Abbitt

Dr. Abbitt received his Ph.D. from the University of Virginia in 1991. Prior to that he served as a P-3 and T-28 pilot in the U.S. Navy. He is now a Senior Lecturer at the University of Florida where he teaches three courses, EML4304C (Thermo-Heats Design and Lab) and EML4147C (Thermo-Fluids Design and Lab), and EAS4939 (Experimental Aerodynamics).

Sam Lowry

Sam received his Ph.D. from the University of Tennessee Space Institute in 1990. Subsequently, he was Director of Advanced Technology at the CFD Research Corportation, and an Engineer at NASA. He also served in the Marine Corps. Sam is now President of Simerics, Inc.