DEVELOPMENT OF AN INNOVATIVE, HIGH-EFFICIENCY RADON MITIGATION FAN

Edward BENNETT, Artem IVASHCHENKO

Mechanical Solutions, Inc., 11 Apollo Drive, Whippany, New Jersey, 07981, United States

SUMMARY

Mechanical Solutions, Inc. (MSI) was awarded a contract from the US Department of Energy (DOE) to develop a high-efficiency radon mitigation fan. Current radon mitigation fan systems range in total efficiency between 15 % and 20 %, accounting for all thermodynamic and mechanical losses in the system. In collaboration with another market-leading ventilation company, MSI developed a radon mitigation fan system exceeding 60 % in aerodynamic efficiency and exceeding 25 % in total efficiency before any possible motor upgrades are added to the aerodynamic upgrades. This development should provide significant energy and operational cost savings and hopefully increase the application of this risk-reducing technology.

INTRODUCTION

Radon is a radioactive gas that is colorless, tasteless, and odorless. It occurs naturally in small quantities due to the decay of other radioactive elements such as thorium and uranium. However, unlike other radioactive elements, it is gaseous under standard conditions and can be inhaled, thus becoming a health hazard. In fact, radon is the second leading cause of lung cancer after active smoking and the second leading cause of cancer among non-smokers. Since the primary source of radon is minerals in the ground, it can accumulate in low-lying enclosed spaces such as underground garages and home basements, especially as houses become more airtight to conserve energy. The United States Environmental Protection Agency (EPA) therefore recommends that steps should be taken to remove radon in homes with levels that are higher than four picocuries per liter.

Existing radon mitigation systems currently consist of vertical fans with primitive diffusers that are inefficient, yet the fans operate 24 hours a day. The US Department of Energy (DOE) therefore awarded Mechanical Solutions, Inc. (MSI) a Phase 1 and 2 Small Business Innovation Research (SBIR) grant to develop a new radon mitigation fan that would fit into the existing structural envelope, but provide a more efficient design that would reduce the operating costs.

The design process began with establishing a manufacturing partner, one of the leading ventilation companies on the market. The partner selected a typical radon mitigation fan design to act as the basis of optimization. A comprehensive CFD analysis of the existing design was conducted to identify possible areas of improvement, and MSI focused its attention on the impeller and stationary diffuser geometries. Multiple alternate impeller and diffuser geometries were developed using a commercial turbomachinery design tool CFturbo and a commercial CFD code Simcenter STAR-CCM+. MSI then manufactured the components using additive manufacturing techniques.

The redesigned fans incorporated the same motor as the existing fan design, even though the partner company informed MSI that the motor was designed for a larger fan model, and was only incorporated into the existing fan design to save the expense of developing a new motor. Upon completion of the prototype development, the two fans were sent to the partner company, which has an AMCA 210-07 certified test facility. The tests of the two prototypes confirmed that both MSI designs significantly outperformed the existing fan design.

TECHNICAL DISCUSSION

Fan Aerodynamic Design

The original stated goal in the DOE requirements was the development of a fan design that would reduce the energy consumption by at least 25 % compared to state-of-the-art units. In addition, it would not drastically increase in size, would have similar installation and maintenance costs, and would have the same lifetime as the existing models. MSI thus teamed up with a leading ventilation company who provided performance data on an existing radon mitigation fan design, along with two common system curves that they encounter, as shown below in Figure 1. The partner company then set the design point based on the two system curves. It was also determined that the electric motor would not be replaced in this effort, so the original motor was incorporated into the new fan design.

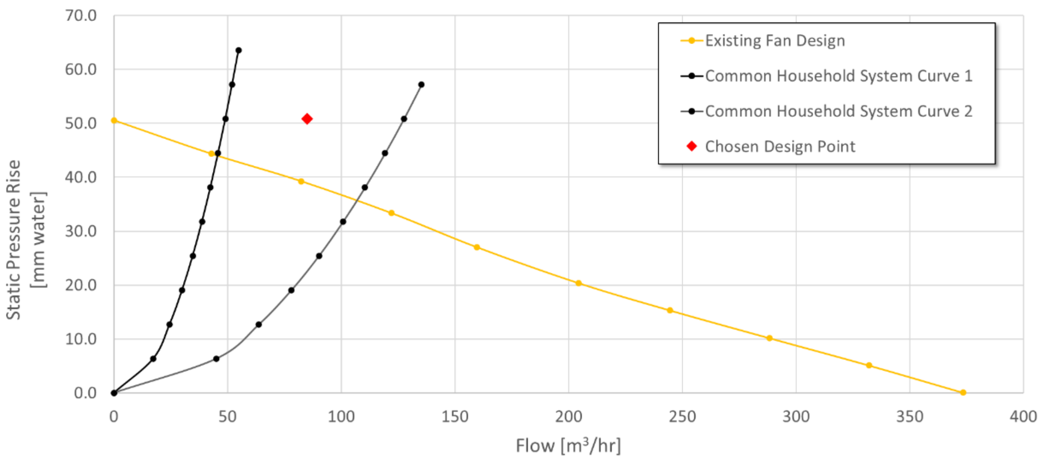

Figure 1: Performance curve for the existing radon mitigation fan design of partner ventilation company

The design point for the radon mitigation fan was the following:

- Inlet flow = 50 cfm, or 84.95 m^3/h,
- Impeller speed = 2987 rpm,
- Static head = 2 inches of H_2O, or 50.8 mm of H_2O,
- Required aerodynamic efficiency of fan > 60 %.

The original fan design is presented in Figure 2 on the next page. It shows negligible regard for flow diffusion, as the flow from the impeller is allowed to directly impinge on the casing wall, and the de-swirl vanes are plain straight struts without camber. There is also no dedicated diffuser passage, as the flow is simply allowed to expand in the space aft of the motor.

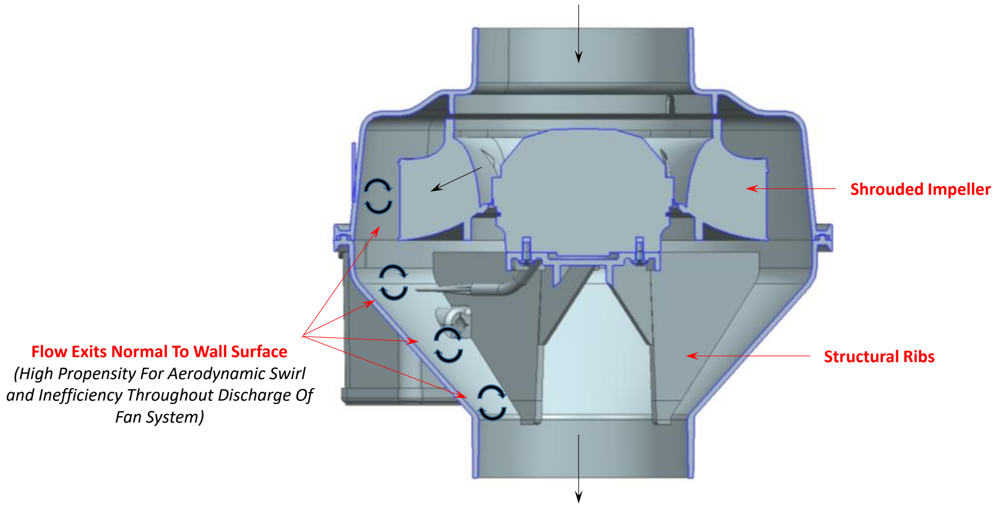

Figure 2: The original fan design selected for optimization

To create the new fan design, MSI employed CFturbo™, a commercial turbomachinery design tool. CFturbo employs a quasi-three-dimensional design methodology that constructs the blade on the classical S1 and S2 surfaces first proposed by Wu in the 1951. The meridional (or cross-sectional) view of one of MSI's designs, designated Impeller10, is shown in Figure 3, along with a two-dimensional flow solver which was used to optimize the meridional shape.

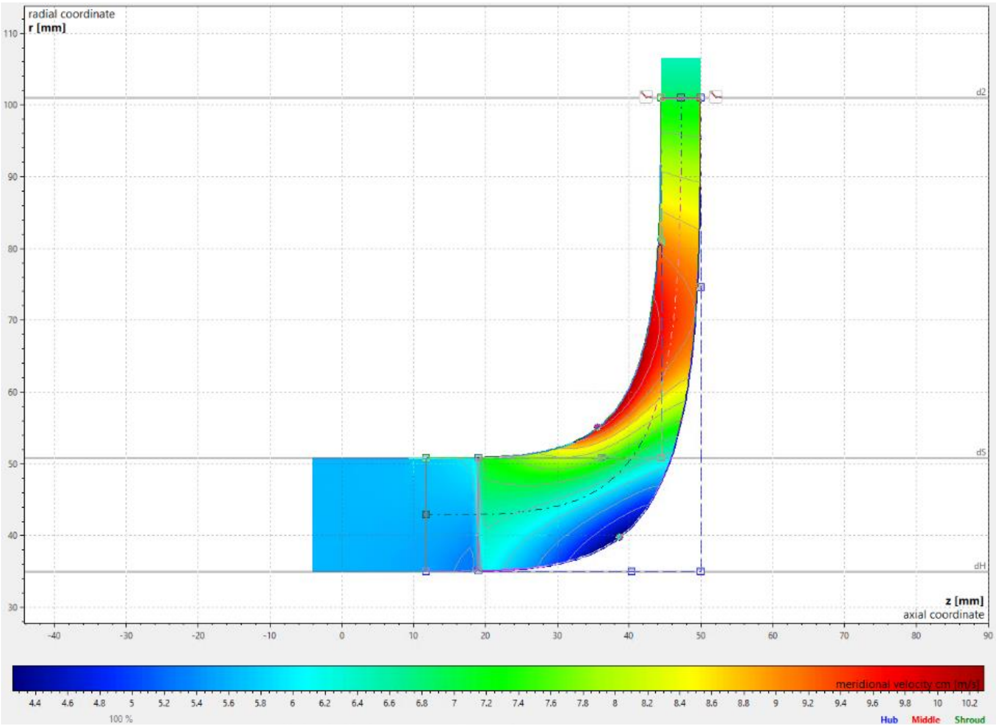

Figure 3: Meridional view of Impeller10 in CFturbo

CFturbo then employs a number of empirical correlations based upon the work of Aungier, Casey, de Haller, Eck, Ludtke, and Stantitz & Prian to estimate the fan aerodynamic performance. A two-dimensional potential solver is included in CFturbo to evaluate the blade-to-blade flow. An additional two-dimensional blade-to-blade solver based upon the classical calculations of Stanitz and Prian is included to optimize the number of impeller blades using the blade loading criterion, which should stay below a value of 0.9, as shown in Figure 4 on the next page.

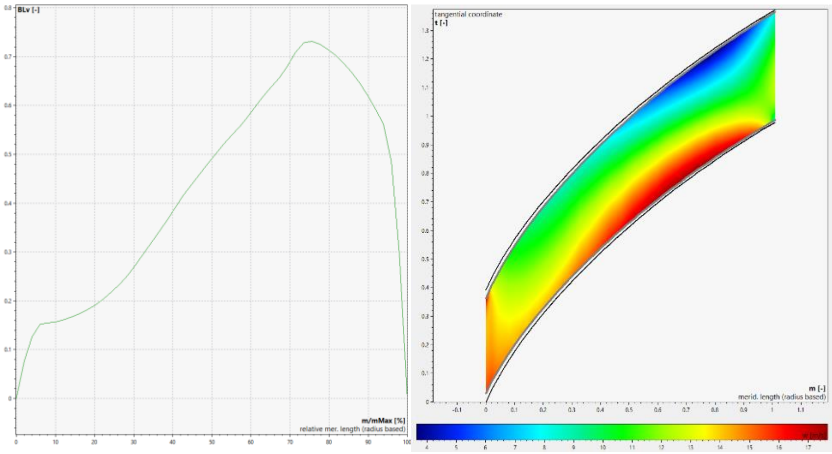

Figure 4: Blade loading of Impeller10 at the shroud in CFturbo

The diffuser was similarly designed using ANSYS BladeModeler, as illustrated in Figure 5.

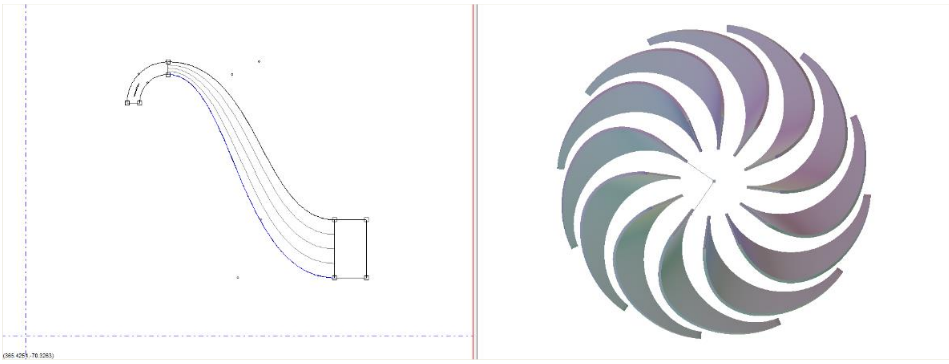

Figure 5: Diffuser design in ANSYS BladeModeler

In the initial Phase 1 effort, MSI produced designs for the impeller and diffuser components of the radon mitigation fan that were three-dimensional, meaning that the vanes were curved in all three dimensions, as opposed to being a straight axial extrusion of a base shape, as is the case in two-dimensional designs, including the base fan design. This concept is illustrated in Figure 6, where a solid model of Impeller10 is shown. MSI submitted this solid model to various manufacturers, but unfortunately, it was not carried to fruition due to manufacturing difficulties and prohibitive costs.

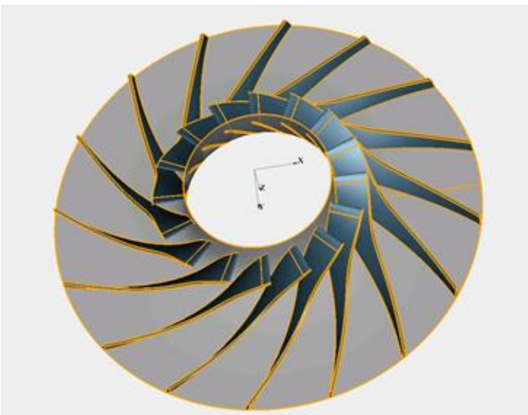

Figure 6: Solid model of Impeller10 vane design (shroud not shown)

Taking those limitations into consideration, MSI created an impeller design with straight two-dimensional axial vanes meant to reduce the manufacturing complexities, designated as Impeller18, and shown in Figure 7. It sacrificed over two points of impeller aerodynamic efficiency for the sake of lowering the manufacturing cost.

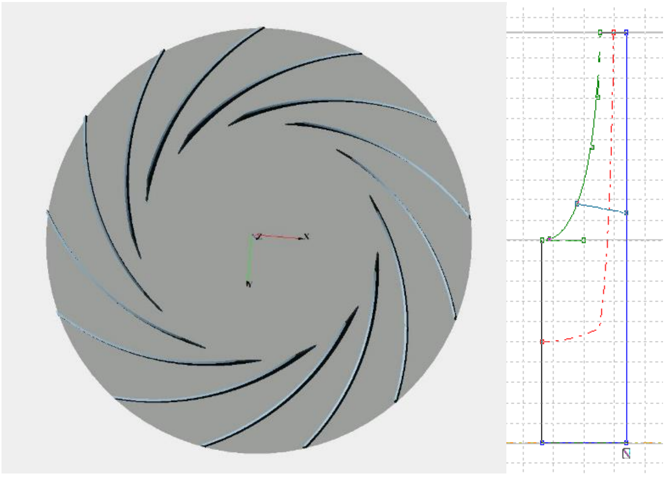

Figure 7: Two-dimensional Impeller18 design (shroud not shown)

Two diffusers were also designed and optimized, presented in Figure 8. Diffuser 15c had a three-dimensional vane with superior predicted performance, while Diffuser rc5 was designed primarily as a two-dimensional axial vane with elements of a return channel, as the vane was shifted towards the exit of the diffuser domain after a run of axial vaneless space. As a result, it did not achieve the same level of predicted performance as Diffuser 15c, but was deemed easier to manufacture due to the relative simplicity of the vane geometry.

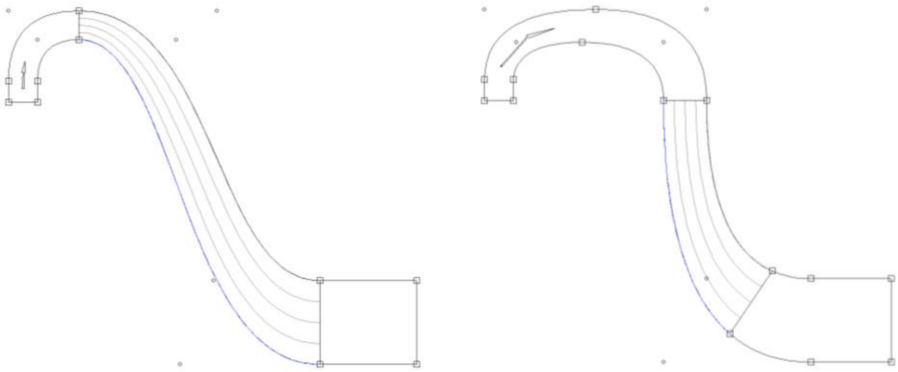

Figure 8: Diffuser 15c (left) and Diffuser rc5 (right) designs in meridional view

To summarize, the optimized fan was designed using the CFturbo™ commercial turbomachinery design tool by varying the following:

- Impeller
    a. Meridional hub and shroud contours to produce minimum diffusion,
    b. Impeller vane number to minimize wetted surface loss,
    c. Camberline profiles from hub to shroud to maintain blade loading below 0.9 at the shroud S_1 blade-to-blade surface, where blade loading is defined as $\left(\frac{w_{suction}-w_{pressure}}{\overline{w}}\right)$, w being the relative velocity and $\overline{w}$ being the averaged passage relative velocity,
    d. Impeller tip width to maintain exit tangential blade angle.
- Diffuser
    e. Maximize radius of curvature in crossover passage,
    f. Optimize de-swirl incidence for broad operation,
    g. Optimize camberline for minimum profile loss,
    h. Optimize exit vane angle to eliminate exit swirl.

Fan CFD Analysis

Both diffusers were combined with Impeller18 design, and the resulting solid models, including all secondary passages, were inserted into a CFD model of the AMCA test rig geometry to predict their performance. The geometry was constructed in ANSYS DesignModeler, and the CFD model was created in Simcenter STAR-CCM+, as shown in Figure 9 and Figure 10.

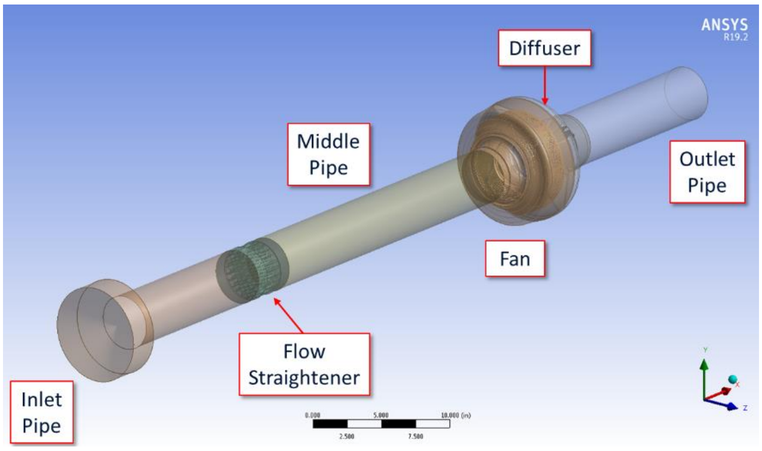

Figure 9: Test geometry in ANSYS DesignModeler

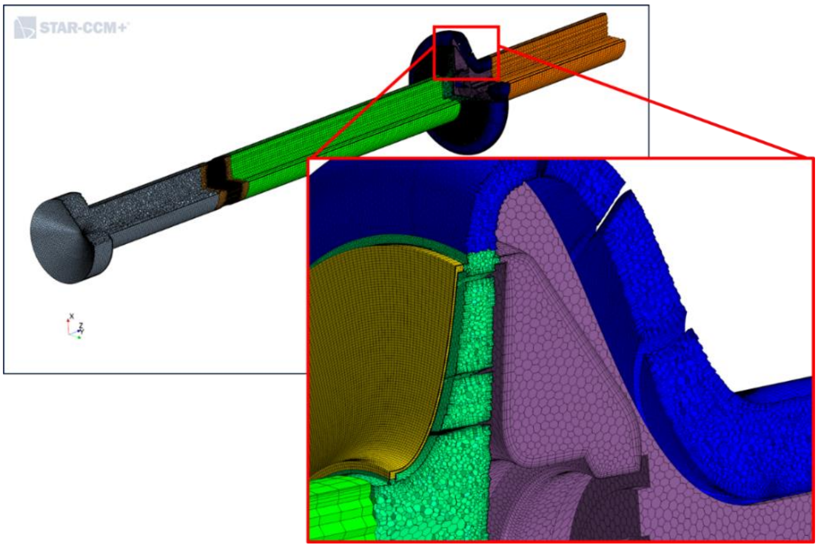

Figure 10: CFD mesh in STAR-CCM+ with Impeller18 and Diffuser15c

MSI employed polyhedral mesh in STAR-CCM+ in the main body of the flow domain and thin prismatic cell layers along the walls to better resolve the boundary layer flow, so that precise friction and torque predictions could be obtained. The non-dimensionalized Y+ values on all the walls were below a value of 5, thus enabling the proper resolution of the viscous sublayer. The CFD models were analyzed using the Shear Stress Transport (SST) k-ω turbulence model and segregated flow solver, which solves the flow equations for pressure and velocity components in an uncoupled manner, and then correlates the results using a predictor-corrector approach. The fluid was modeled as constant-density air at Standard Temperature and Pressure. The models were set up with mass flow boundary conditions on the inlet and a static pressure of 1 atm on the outlet. Pressure probes were inserted into the flow domain at the appropriate locations to monitor static pressure. The models were then analyzed in a transient simulation. The impeller mesh was rotated with respect to the mesh of the stationary components at every timestep. A new timestep solution was obtained for every variable of interest, such as pressure, torque, etc. The number of inner iterations for every timestep was set at 20, so that all residuals had time to decrease by several orders of magnitude. The variable results were then averaged over the last several revolutions, thus calculating the predicted performance at every given operating point.

The fan performance CFD results are presented in Table 1. Three points were analyzed for each design, corresponding to the design point and the two system curves shown in Figure 1. The results show that both designs are predicted to achieve the analytical performance goals established by the project, exceeding 50.8 mm of pressure rise at the flow of 82 m^3/hr. However, it is essential to note that the fan design with Diffuser 15c, which has three-dimensional vanes, exceeds the performance of the fan design with Diffuser rc5, which employs simpler vanes. Additional plots for pressure, velocity flowfield, and streamlines are presented in Figure 11. Most notably, they show more swirl at the exit of the Diffuser rc5 than Diffuser15c, as the shorter 2D vanes are not as efficient in converting the circumferential dynamic pressure of the flow into extra static pressure rise.

Table 1: Fan performance predicted by the CFD analyses

	Flow (m^3/h)	**Static Head (mm H_2O)**	**Diffuser Total Head Loss (mm H_2O)**	**Torque (N-mm)**	**Speed (rpm)**	**Power (W)**	**Aero Efficiency (%)**
Fan design with Diffuser 15c	42.5	61.2	10.6	43.3	3034	13.8	51.4
	82.1	59.8	9.0	63.8	3023	20.2	66.1
	159.7	32.1	10.3	78.9	2987	24.7	56.6
Fan design with Diffuser rc5	42.5	56.7	18.4	42.3	3034	13.4	48.8
	82.1	52.5	17.6	62.9	3023	19.9	58.8
	159.7	21.2	19.0	79.4	2987	24.8	37.1

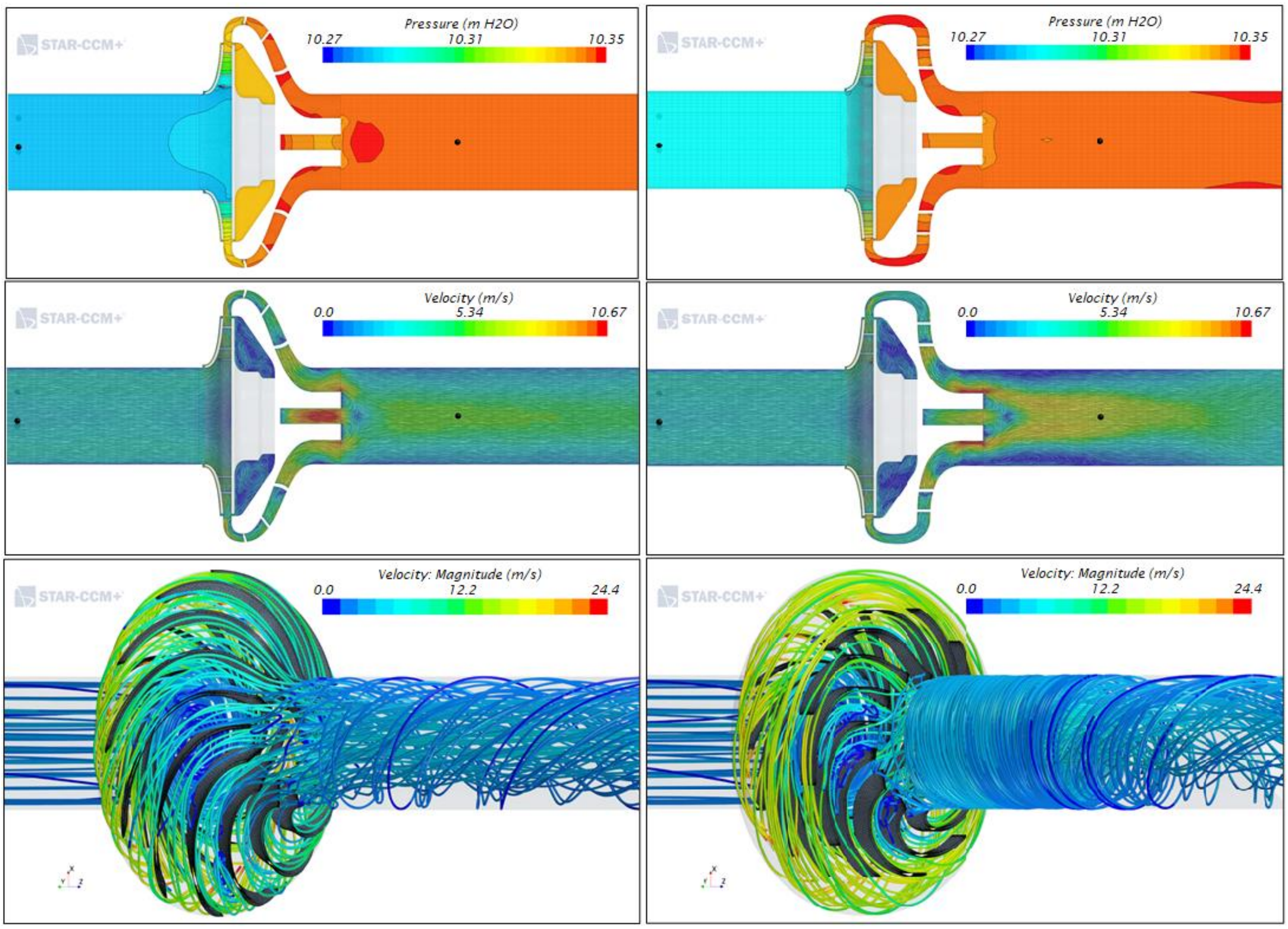

Figure 11: Static pressure with probe locations, velocity flowfield, and streamline plots for the Diffuser 15c design (left), and Diffuser rc5 design (right) at 82 m^3/h

Fan Mechanical Design, Manufacturing, and Testing

The fan assembly, shown in Figure 12, maintained the same general layout as the original design, including the motor itself. The components were then 3D-printed in Nylon 12 using the multi-jet fusion (MJF) process (Figure 13). Though the printing process provided adequate resolution and tolerance control for the hardware, the surface finish ended up being much rougher than what would be obtained from a traditional injection molding process.

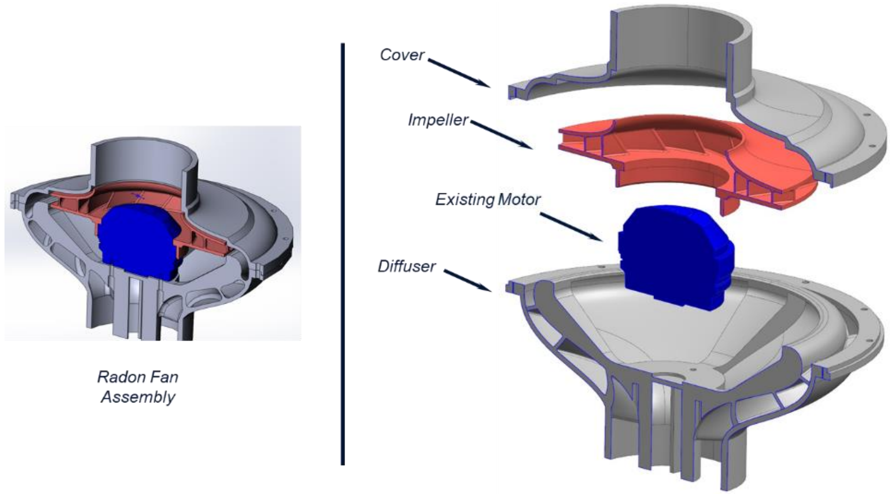

Figure 12: Final radon mitigation fan assembly (Diffuser 15c design shown)

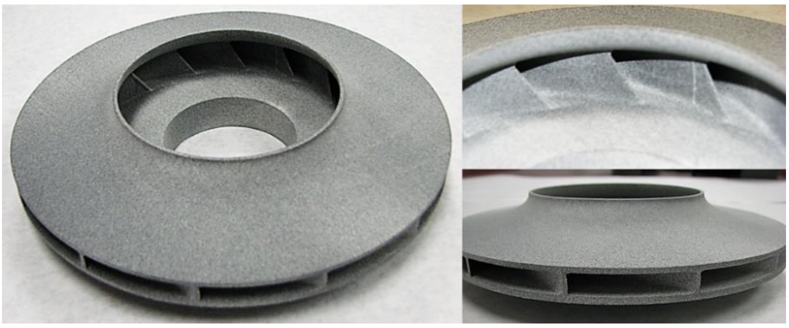

Figure 13: 3D-printed Impeller18 design used for performance testing

Each unit was operated to the intended design speed to ensure acceptable mechanical performance. Once complete, the two prototypes were shipped to the partner company, which has an AMCA 210–07 certified test facility, shown below in Figure 14, where their performance was tested.

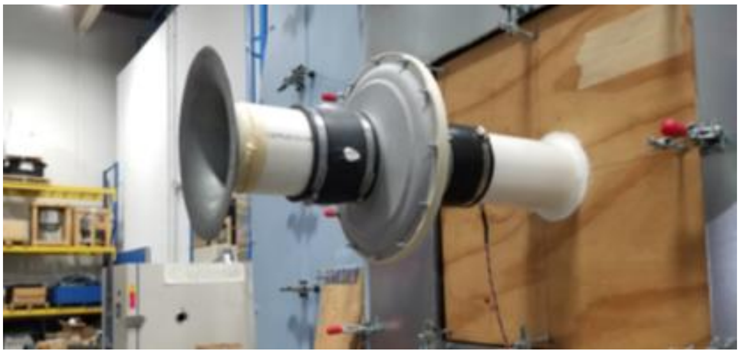

Figure 14: AMCA 210-07 certified test rig at the partner company facilities

The test results are presented below in Figure 15. The head curves closely follow the analytical predictions from CFD analyses, thus satisfying the design point criteria, and both fan designs exhibit superior performance to the original design, as the tested efficiency is about 8 points higher for the new design than for the original design.

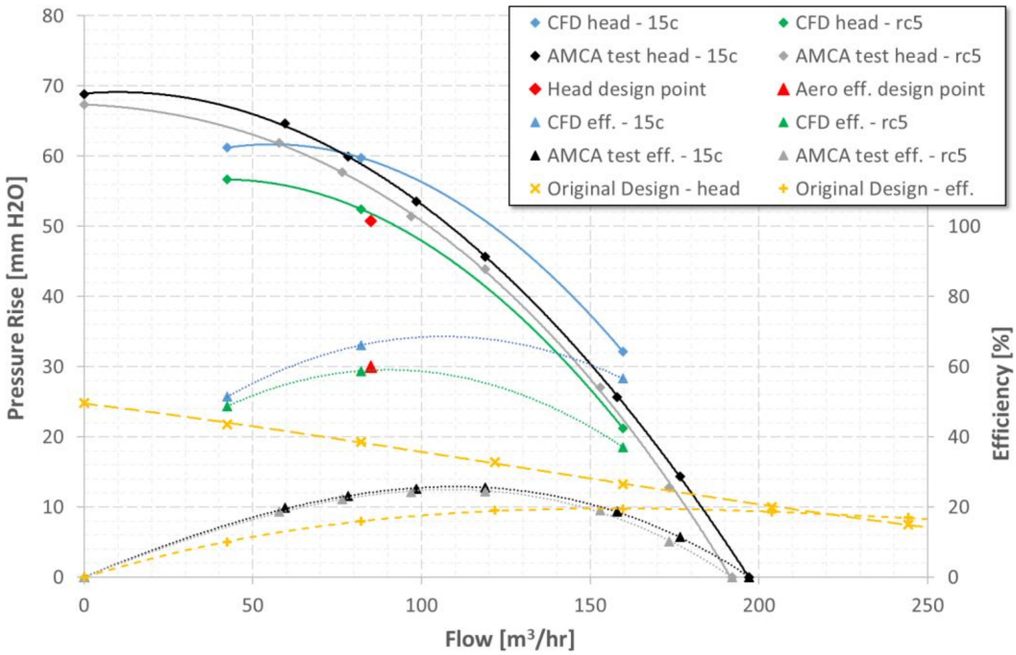

Figure 15: Fan performance curves for CFD analyses and AMCA test results

The predicted CFD efficiency of each fan is much higher than tested efficiencies, and there are several reasons for the differences. The physical test measures the combined fan aerodynamic and motor performance, while the CFD analysis only considers the aerodynamic performance. The CFD analyses did not account for the rough surface finish of the 3D-printed prototype, which was also not scanned to check for a comprehensive match with the CAD model. Hub and shroud filler were not modeled in the CFD analysis, and slight variation in shroud seal gap could also have a significant effect on volumetric efficiency. Considering the fact that the original fan was a production model with good surface finish and sealing, and its performance was inferior to the two prototypes, the difference between predicted aerodynamic efficiency and full fan efficiency is mainly attributed to the inadequate motor performance.

CONCLUSIONS

MSI successfully created a design for the radon mitigation fan that significantly improved upon the existing state-of-the-art design by achieving over 60 % aerodynamic efficiency. Overall, 18 impeller prototypes and 23 diffuser vane/return channel configurations were designed and analyzed using the MSI fan design protocol that incorporated the commercial turbomachinery design tool CFturbo, ANSYS BladeModeler and the commercial CFD software Simcenter STAR-CCM+. Ultimately, two fan designs were chosen for testing, one that incorporated a diffuser with three-dimensional vanes with a superior predicted performance, and another diffuser with elements of a return channel and straight two-dimensional axial vanes. Both design also incorporated an impeller with straight two-dimensional axial vanes, which was a compromise to reduce manufacturing cost by sacrificing some aerodynamic efficiency.

Prototypes of the two MSI fan designs were 3D-printed using the MJF process and subjected to comprehensive performance testing conducted in accordance with the AMCA 210-07 Test Standards. The tests performed at the partner company's facilities demonstrated a large increase (8 points) in overall efficiency compared to the original design. The design head was exceeded by both prototypes, which offers the possibility of additional efficiency gain through increased backsweep of the impeller vanes, thus also reducing the fan power. Further improvements in efficiency can be achieved with enhanced impeller front cover sealing, the employment of three-dimensional vanes on the impeller itself and a new motor that better matches the desired fan efficiency point.

BIBLIOGRAPHY

[1] ANSI/AMCA 210-07 – *Laboratory Methods of Testing Fans for Certified Aerodynamic Performance Rating*, An American Standard Approved by ANSI on August 17, **2007**

[2] R. Aungier – *Centrifugal Compressors*, ASME Press, **2000**

[3] M. Casey – *The Industrial Use of CFD in the Design of Turbomachinery*, AGARD, **1994**

[4] P. De Haller – *Das Verhalten von Trafflugegittern in Axialverdichtern und im Windkanal Brennstoff-Wärme-Kraft*, Band 5, Heft 10, **1953**

[5] B. Eck – *Ventilatoren*, Springer-Verlag, **1991**

[6] K. Ludtke – *Process Centrifugal Compressors*, Springer-Verlag, **2004**

[7] J. Stanitz and V. Prian – *A Rapid Approximate Method for Determining Velocity Distribution on Impeller Blades of Centrifugal Compressors*, NACA TN 2421, **1951**

[8] CFturbo – *Version 10.4 Manual*, **2019**

ACKNOWLEDGEMENT

The authors would like to thank our former colleague, Travis Jonas, who has unfortunately passed away, for his enormous input into this project, especially project management, mechanical design, manufacturing and testing. This paper would be impossible without him.