Topology optimization for design of a water monitor

Alamgir Choudhury and Jorge Rodriguez

June 11, 2018

Topology optimization for design of a water monitor

Alamgir Choudhury Ph.D[1], Jorge Rodriguez Ph.D[2]
[1]EDMMS Department, Western Michigan University, USA, alamgir.choudhury@wmich.edu
[2] Western Michigan University, USA, Jorge.rodriguez@wmich.edu

Abstract* – For a given firefighting pump, the delivery of the maximum amount of water often determines effectiveness of an overall system. In typical commercial water monitors, about 10% of water pressure is lost due to friction in the flow path of the water monitor. With proper analysis of the flow system and shape geometry, this pressure loss can be minimized significantly. This would allow fire fighters to pump more water in a shorter period, and in turn, save lives and property. A typical commercial water monitor has a diameter of four inches and delivers 2,500 gallons per minute (GPM) of water at 100 psi pressure. In this paper, the design analysis of pressure loss and shape optimization of a larger water monitor is presented. The monitor has a diameter of six inches and would deliver 3.000 GPM water at 100 psi pressure. Two path radii define the shape of the water monitor and its size is constrained by a rectangular prism of given dimensions. Based on a pressure loss cost function, a commercially available Computational Fluid Dynamics (CFD) tool is used to optimize the geometry of the flow path. The shape of its cross section is optimized by utilizing the derivative of the cost function with respect to the cross sectional area of the water monitor. The resulting system improved pressure loss by more than 44% compared to a circular cross sectional flow path in the standard design of a commercial system.

Keywords-- Water Monitor, CFD, FEA, Cost function and Topology optimization.

I. INTRODUCTION

A water monitor, also known as a water cannon, is an important part of a firefighting system. The purpose of these monitors is to allow the delivery of the maximum amount of water or firefighting chemicals in the right trajectory to reach the desired location. Depending on the application, these monitors may be fixed at a site or portable with a firefighting system. A water monitor can be as simple as a unidirectional nozzle that guides water to the fire source. To enhance the versatility of guiding water in right direction, commercial water monitors are designed as multi degrees of freedom system. Since the loss of water pressure results in the reduction of flow, the design of a water monitor requires an investigation of pressure loss and its relationship to dimensional and shape geometry.

A wide variety of commercial water monitors are available on the market. Their discharge capacity varies from a few hundred gallons to thousands of gallons per minute (GPM). Elkhart Brass designs [1] have fully enclosed and sealed gear cases and have built in override devices. The Cobra EXM model is designed to be used on pumper and aerial trucks in the field. The inlet of a monitor varies from 3.5 to 4 inches in diameter with a maximum flow rate of 1500 GPM at an operating pressure of 500 psi. Many of these monitors are made of hard anodized aluminum alloy, and their surfaces are Teflon impregnated. The waterway is vaned and goes from a circular waterway to an ellipse when making a hard turn. The goal of these features is to improve the flow and reduce friction loss. At 1500 GPM, a typical water monitor has about 30 psi pressure drop from the inlet of the monitor to outlet. Other monitors may be made of caste brass with an inlet diameter of 4 inches and an outlet diameter of 3.5 inches with a maximum flow rate of 2,000 GPM and an operating pressure of 250 psi. The waterway may be open or fully vaned to reduce turbulence. They may be controlled manually or come with an electronic control to guide the water jet. The Scorpion EXM monitor is a similar product with a 4-inch inlet a 3.5-inch outlet. The maximum flow rate of the Scorpion is 2500 GPM and can operate at a maximum pressure of 250 psi with a pressure loss of just under 60 psi. These monitors are used with remotely controlled electronic motors in pumper trucks and de-icing vehicles. Akron Brass [2] has monitors of different designs. The 3690 Storm Monitor features a 5-inch nozzle and allows for a full 360-degree rotation. The vertical travel for this monitor goes from +85 degrees to -45 degrees for a total range of 130 degrees. This brass monitor has a capacity of 2,000GPM and is controlled with hand wheels. The StreamMaster II is made of light weight Pyrolite and has a 6-inch operating envelope which is very compact in its current design. This monitor can flow up to 2,000 GPM and allows almost a full rotation (355 degrees) and an elevation range of 165 degrees; ranging from +120 degrees to -45 degrees. The StreamMaster II monitor also features a waterproof control system which allows the user to be at a safe distance when operating the monitor with the use of a hand-held remote. The friction loss for this design ranges from (almost) 0 psi at 250 GPM, up to 42 psi at 2,000 GPM which is shown in the graph to the right in Fig. 2. It has an outlet nozzle that ranges from 2.5 to 3.5 inches and works as a 12 or 24 Volt unit. Williams Fire [3] monitors have of 4, 6, or 8-inch nozzles, and can pump water or foam up to 6,000 GPM. Pressure loss in 4-inch nozzles vary from 0 psi at 435 GPM to 13 psi at 1500 GPM. For the larger 6 and 8-inch monitors, pressure drops are 16.7 psi at 3,000 GPM and 21.2 psi at 6,000 GPM respectively. The monitors' range can discharge water from +80 degrees to -40 degrees vertically and full 360 degree horizontally. The monitors with 4-inch nozzle and has a pressure loss from approximately 0 psi at 435 GPM to 13 psi at 1500 GPM.

Due to market demand for water monitors with a high flow rate, an industry supported water monitor design problem was investigated at Western Michigan University in 2013 [4]. The goal was to study the pressure loss characteristics in a 6-inch water monitor at a pressure of 100 psi and flow of 3,000 GPM. In the conceptual model, two tangential circular arcs rotating the flow direction by 90 degrees defined the flow path contour.

The water way was created by a 360 degree sweep of the contour. The radii of the arc primitives were defined as parametric variables in the mathematical model. Geometric features of standard monitors were utilized to develop the complete solid model of a six-inch diameter water monitor. The final solid model is composed of two flow guide elements, a flanged base and a flow nozzle. It also included an internal vane along the flow path and standard water monitor fittings. The development of the solid model was done using a professional version of SolidWorks [5]. Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD) methods were used to analyze the characteristics of the flow system. CFD analysis of the water flow problem showed that pressure loss generally decreases with the increase in path radii. Within the constraints of the monitor volume envelope, optimal pressure loss was at specific combinations of the path radii. The analysis resulted in a design with a 3.75 and 3 inch radius of the curvature with a pressure loss of 3.229 psi which is over a 34% improvement compared to existing monitors of similar capacity.

To be competitive in the market, water monitor designers consider many factors, such as function, material, size, weight, operation and control, product life cycle, manufacturability, maintenance, environmental conditions, power source, and - above all - cost. Like other mechanical systems, designing a water monitor can be a complex process. Though the previous design process improved the performance of a system with the given constraints, the process sought a solution to the design problem through incremental changes within the existing design parameters. Therefore, the design was based on a localized solution to a problem. For global solutions to the problem, besides the use of parametric solid model and CFD analysis, the optimization [5] of an objective function is also necessary. This paper presents an analysis for the performance and efficiency of the flow system. The behavior of the water flow system as related to the flow path geometry and cross section is investigated and dimensional parameters of the shape and cross section are optimized. This paper presents the use of a parametric solid model and CFD analysis in topology optimization [7] for a design analysis of the water flow system.

II. PROBLEM DEFINITION

The design problem is based on flow path parameters and the cross section of the earlier system [4]. The water monitor is constrained in a $26X15X6$ parallelepiped volume with a vertical entrance and horizontal exit of fluid. There are four primitives joined tangentially to generate the flow path trajectory in the envelope. There are two 2.9 inch horizontal and vertical lines at the entrance and exit, and arcs with 3.5 and 3-inch radii in the middle. The path primitives are swept to generate the 4-inch diameter water monitor. An ANSYS parametric model would be used to define the geometry and its mesh morphing method will be used to vary the geometric shape. An ANSYS Fluent Adjoint Solver [8] tool will generate corresponding fluid pressure loss and flow data. To generate the optimal shape of the system, a cost function based on the design objectives will be utilized in the optimization process.

III. THEORETICAL BASIS

The fluid flow characterization in most hydraulic systems can be derived from the Navier Stokes equation

$$\frac{\partial \rho}{\partial t} + \nabla(\rho V) = 0 \quad \ldots. (1)$$

Where, ρ = density, V = flow velocity   vector divergence operator of a general flow field.

Considering energy loss in a flow system, equation (1) for one dimensional fluid flow simplifies as Bernoulli's equation

$$\frac{P_1}{\gamma} + z_1 + \frac{V_1^2}{2g} - h_L = \frac{P_2}{\gamma} + z_2 + \frac{V_2^2}{2g} \quad \ldots. (2)$$

where h_L is head loss between the inlet and exit of a flow conduit.

Based on flow criteria in the system, equation (2) can be used to calculate pressure loss ΔP as

$$\Delta P = \gamma\, h_L \quad \ldots. (3)$$

Since, the flow rate and pressure are function of geometry, temperature, and other fluid properties, in general, pressure loss can be expressed as a nonlinear function of n different parameters of the flow process given by

$$\Delta P = F\,(q_1, q_2, q_3, q_4, \ldots q_n) \quad \ldots. (4)$$

where q represents different flow system parameters.

The study of pressure loss in a standard 90-degree bend [9] shows that the loss can be minimized by increasing the bend radius in general. But dimensional constraints of the water monitor volume envelope, beyond a certain radius, where this pressure loss starts increasing [4]. To determine the optimal parameters of the flow system, a cost function needs to be minimized. Parametric modeling, shape morphing, and the topology optimization process is shown in Figure 1 and 2.

Parametric CFD [10] is used in both the conceptual and improvement stages of a design process to validate or rectify a particular design. Topology optimization [11] of a nonlinear fluidic system often suffers from the slow

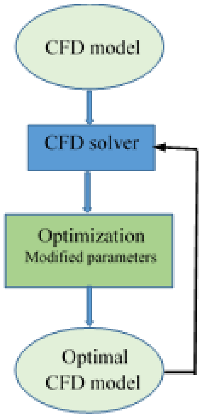


Figure 1: Parametric Optimization process

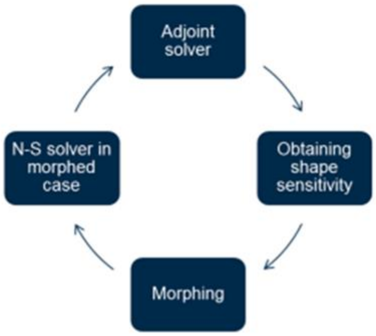


Figure 2: Adjoint optimization loop

convergence of the optimization process, as well as the robustness at an increased Reynolds number. Non-smooth material distributions may trigger the premature onset of stationary flow, which cannot be treated as a steady-state flow model. In general, parametric level-set methods allow for the control of the smoothness of boundaries and yield a non-local influence of design variables. They also decouple the material characteristics from the flow field discretization.

IV. SOLUTION OF SHAPE OPTIMIZATION PROBLEM

Conventional optimization techniques like CAD parameter-based schemes or morphing methods are often limited due to their usual extensive computational requirement as well as their intrinsic limited solution space [7]. In parametric optimization schemes, the starting point for an optimization procedure is a parameterized initial design guess of a base line model. But a topology optimization problem is based on only the available design space and the geometric limitation for the intended design solution. Topology optimization can be directly used for the determination of a first basic layout of a new design. It involves the determination of features such as the number, location, and the shape of "holes", i.e. introduced non-fluid regions and the connectivity of the domain. A new design is determined based upon the available design space and the boundary conditions of the flow problem. The topology optimization for CFD applications has been attempted only recently [9]. Most of the approaches were used in structural mechanics that cannot be easily transferred to fluid dynamics problems. The governing equations and the solver techniques are different and hence new optimization methodologies have to be found. Due to the strong nonlinear nature of the Navier Stokes equation, general optimization schemes are not feasible to solve the topology optimization problem on a cell-per-cell basis.

The goal of this study is to investigate the effect of shape sensitivity on the performance of an existing flow monitor as shown in Figure 3. The ANSYS Fluent Adjoint Solver is used to predict how the shape can be optimized in order to minimize pressure loss and withstand the effects of higher water pressure. Though this design process improves the performance of a system with the given constraints, the process seeks solution to the design problem within the vicinity of existing design parameters. As a result, the design is based on a local solution to the design problem. For global solutions, beyond the use of parametric solid model and CFD analysis, the optimization [6] of an objective function is essential. Here, we see the use of a parametric solid model and CFD analysis in topology optimization [6] for the design of the water flow system.

IV. TOPOLOGY OPTIMIZATION

Many optimization methods exist. Because of their computational requirements and efficiency in the convergence of the optimization process, they may not be suitable for this type of problem. The gradient based method is the most well-known method that can cope with many design variables. The gradient of the cost function is the key factor for further shape optimization.

In this study, the ANSYS Adjoint Solver tool is used for the internal flows and objective function based on pressure loss. A variety of parameters, which could influence the Adjoint solution and the morphing feature, have been investigated. The mesh type, skewness, and Adjoint discretization scheme are a few of the parameters deemed important for a robust methodology. According to the geometry of a water flow monitor (Figure 3), a compatible mesh generator with the Adjoint solver in terms of mesh quality was investigated. For clarity of the Adjoint and morphing tool, the entire geometry was morphed without limitations. The previous solution for the water monitor problem (Figure 4) was the initial guess of the shape optimization problem.

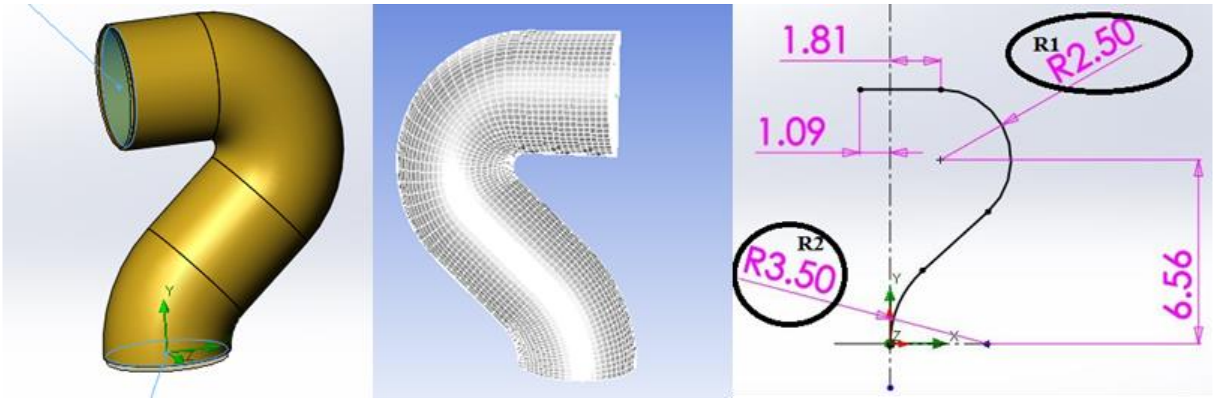


Figure 3: Initial flow monitor model and its primitives

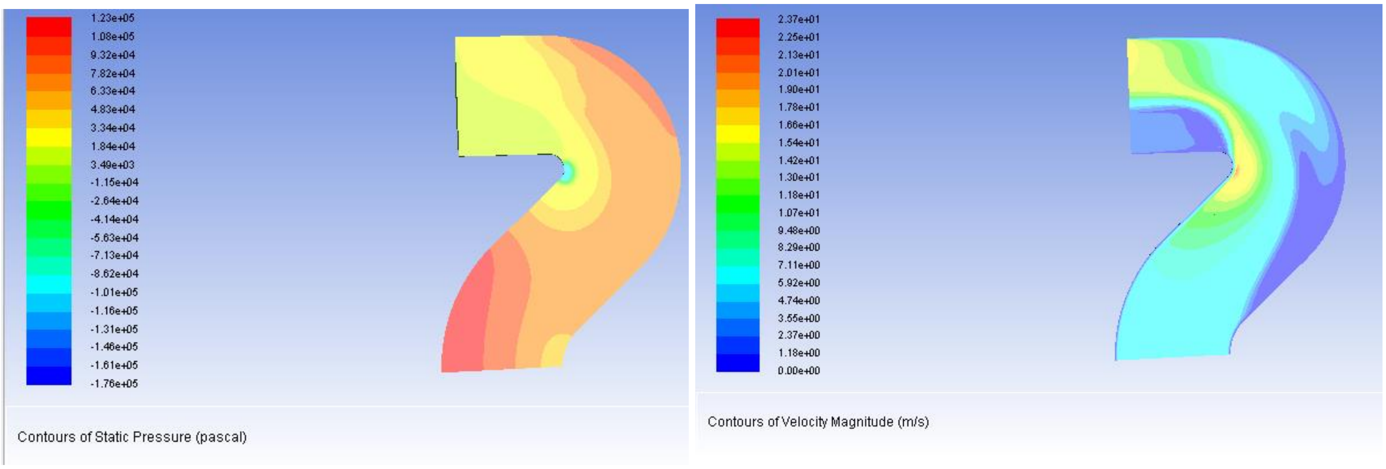
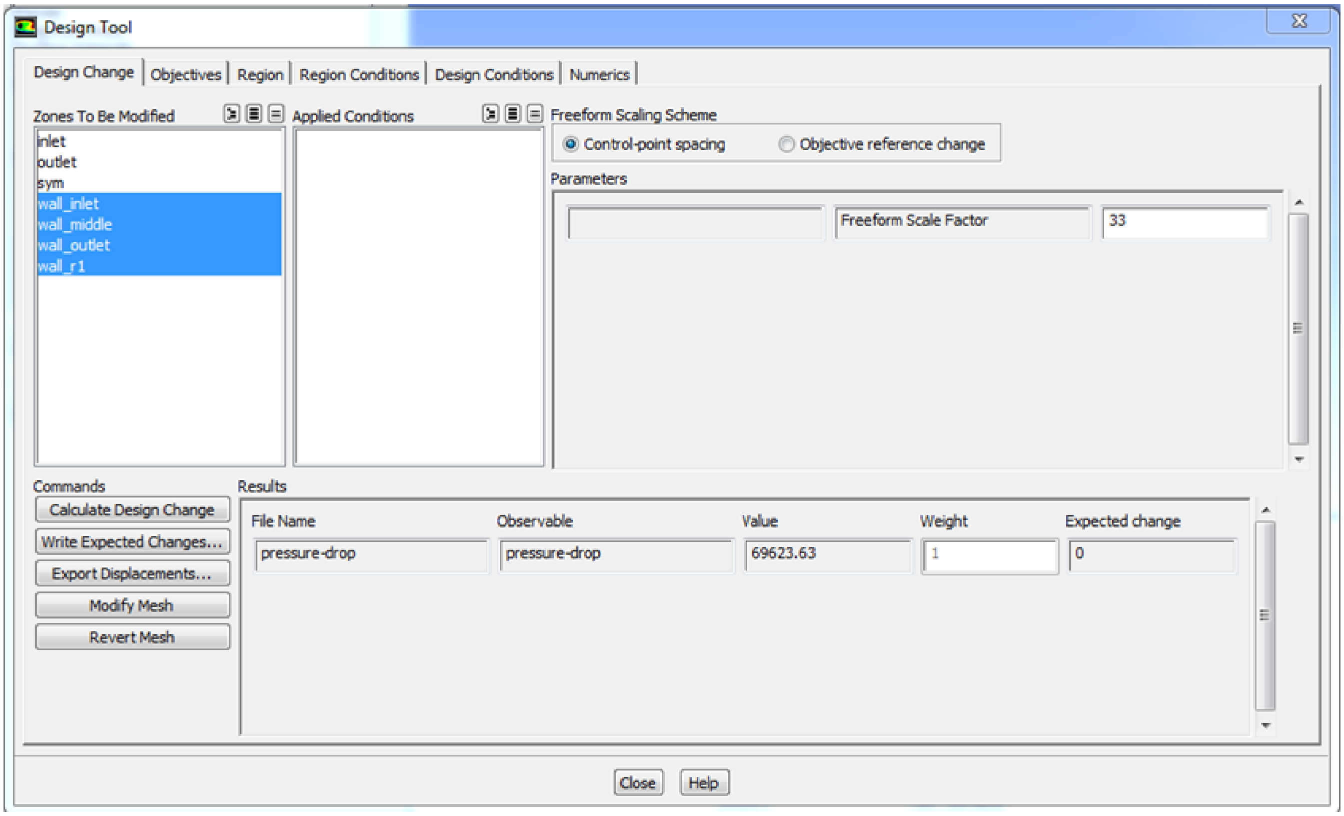


Figure 4: Pressure and velocity map of initial design

Numerical optimization methods in combination with simulations are used in this design process. These methods required the gradient of the objective function with respect to modifications in the desirable variable (in this case cross section geometry) in order to deal with the magnitude of domain change. The finite difference method is the simplest way to calculate those gradients. However, it requires an iterative procedure for the gradient components. These are obtained by independently changing each variable in finite steps, calculating the desired objective function, and estimating the ratio of differences. In each iteration, the local minimum or maximum of the objective function is calculated and the process is repeated until the gradient is closest to zero. So, in case of perturbing *n* points on the given surface, *n* flow simulations are required to obtain the data set which is used in the optimization process. Consequently, the finite difference method is computation intensive as the design variables *n* is large. In contrast, the Adjoint method is able to obtain design sensitivities of a function with respect to design variables in a Single Adjoint computation. This implies that the computational cost for the gradient calculation is equal to that of solving the equations. Because of its ability to provide robust sensitivity information in complex geometric configurations, ANSYS FLUENT solver applies the discrete approach. A converged Adjoint solution provides the domain morphing sensitivities, identifying the areas where the domain mesh should be modified. The ANSYS FLUENT mesh adaption method is based on a gradient algorithm, adjusting



Zones To Be Modified

- inlet
- outlet
- sym
- wall_inlet
- wall_middle
- wall_outlet
- wall_r1

Applied Conditions

Freeform Scaling Scheme

● Control-point spacing ○ Objective reference change

Parameters

Freeform Scale Factor 33

Results

File Name	Observable	Value	Weight	Expected change
pressure-drop	pressure-drop	69623.63	1	0

Figure 5: Adjoint Solver parameter setup

the system in a way that maximizes the effect of the change. For instance, in areas where sensitivity is relatively high, small modifications in shape will lead to a high impact on the objective function. Additionally, a smooth distributed mesh is applied on the surface with respect to the cost function. The initial setup of the problem in the ANSYS FLUENT solving system is shown in figure 4. The method successfully converged the shape problem to a variable section solution (Figure 5) with a significant increase in flow velocity. The initial reaction to such an atypical solution for a problem may lack confidence in the effectiveness of the shape optimization method. But such a solution can be seen in many biological systems in nature, such as the neck of a flamingo or swan. The pressure and velocity profile of the system demonstrates significant improvement compared to the system's initial design (Figure 7).

Similar studies have been done to evaluate different design scenarios in the ducts of car ventilation systems. After optimizing the design objectives by using the ANSYS Adjoint solver, a considerable reduction in pressure drop was achieved.

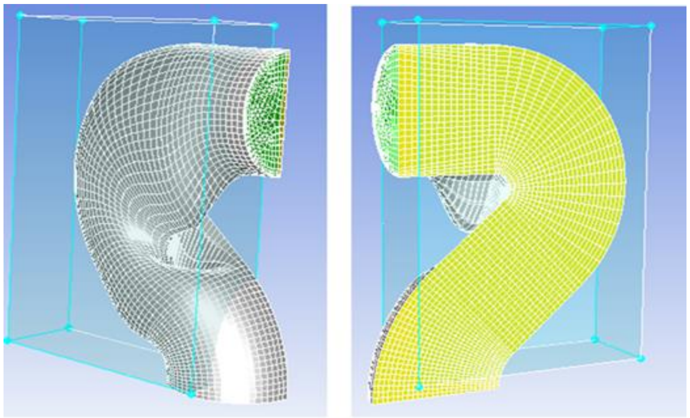


Figure 6: Shape of flow system through ANSYS Adjoint solver

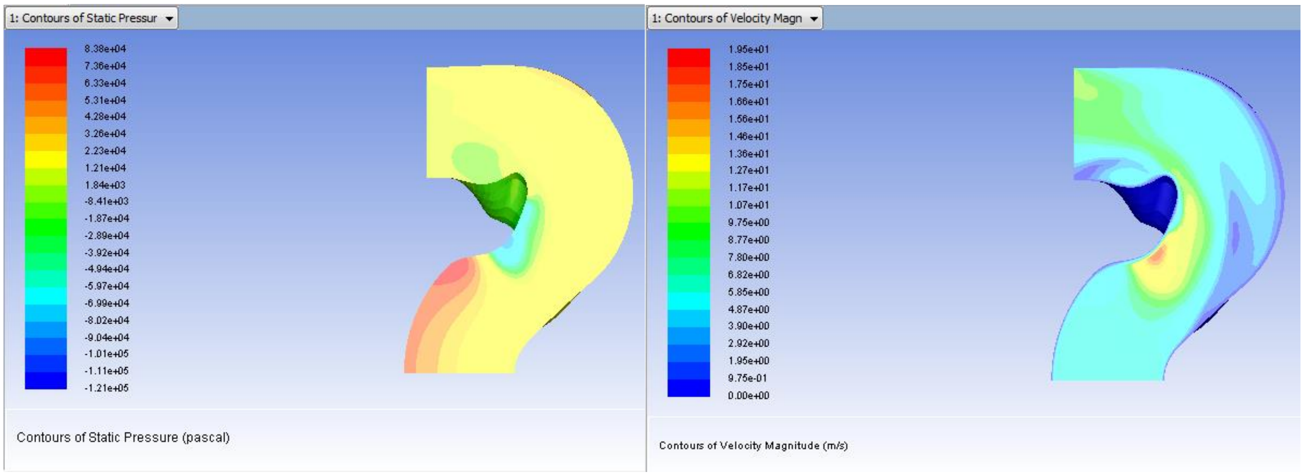


Figure 7: Pressure and velocity map for optimized model

V. CONCLUSION

The Adjoint solver is a powerful tool which can significantly reduce the development time of a water flow monitor. The cost function in this design process is the minimization of pressure loss which was successfully achieved. Based on given initial design parameters, the Adjoint solver method converged to a specific solution of the water monitor problem in just 100,000 iterations. Pressure loss in this new design reduced from 69.47 kPa (10.11 psi) to 38.53 kPa (4.56 psi), which is a 44% improvement compared to the initial solution to the problem through only a CFD analysis. The optimized geometry of the water monitor was achieved through the topology optimization method of the Adjoint solver in few hours compared to the multitude of days necessary for similar problems based on other methods. Since the design did not consider manufacturability, cost, or other constraints, the current solution may not be cost-practical, but it is the best for the given objective of only pressure loss minimization. If other objectives are considered, the method will produce different solutions consistent with those design objectives.

REFERENCES

[1] Elkhart Brass Manufacturing, Monitors Elkhart Brass - Fire Fighting Equipment. Retrieved October 14, 2013, from http://www.elkhartbrass.com/products/monitors.

[2] Akron Brass Storm Monitor and Background Information, Akron Brass Company. http://www.akronbrass.com/monitors/stormtm-monitor-with-style-5079-nozzle.

[3] Williams Fire & Hazard Control, Model 5 Water Powered Oscillating Monitor-High Flow. http://www.williamsfire.com/product/0b2a662a-8183-4b16-8282900eaed98811/Model_5_Water_Powered_Oscillatin g_ Monitor_-_High_Flow.aspx.

[4] A. Choudhury and J. Rodriguez, "Bend Radius Analysis for Water Monitor in Capstone Design Project," *14th Latin American and Caribbean Consortium of Engineering Institutions*, San Jose, Costa Rica, July 2016.

[5] Solidworks Solutions, retrieved from http://pdf.directindustry.com/pdf/solidworks/solidworks-solutions/15020-657129.html, March, 2018.

[6] L. Jiang and S. Chen, Parametric structural shape & topology optimization with a variational distance-regularized level set method, *Computer Methods Applied Mechanics and Engineering*, Vol: 321, 2017.

[7] M. Stephan, P. Haubler, and M. Bohm, *CFD Topology Optimization of Automotive Components, 4th European Automotive Simulation Conference*, Munich, Germany, 2009.

[8] ANSYS Fluent Adjoint Solver, retrieved from: https://support.ansys.com/staticassets/ANSYS/Conference/Confidence/San%20Jose/Downloads/optimization-adjoint-solver-9.pdf, March, 2018.

[9] R. Chowdhury, M. Alam and S. Islam, Numerical modelling of turbulent flow through bend pipes, *Mechanical Engineering Research Journal*, Vol. 10, 2016.

[10] L. Nitsopoulos, M. Stephan, and M. Böhm, An Efficient Approach for CFD Topology Optimization of interior flows, *3rd ANSA & META International Conference*, 2009.

[11] G. Pingen, et al, A Parametric level-set approach for topology optimization of flow domains, *Structural and Multidisciplinary Optimization*, Volume 41, 2010.