

# A Robust Analytical Sensitivity Analysis for Weakly Coupled Aero-Structural Systems

Koorosh Gobal\*, Ramana V. Grandhi<sup>†</sup>

*Wright State University, Dayton, OH 45324*

In this research, a continuum sensitivity formulation for the shape sensitivity analysis of weakly coupled fluid-structural systems is derived, where the flow is modeled using Navier-Stokes equations and the solid domain is represented as linear elastic domain. The conventional approach to this problem is based on body-conformal grid to represent the solid boundaries. However, these methods will fail for complex shape and/or large deformations of the solid region. This issue can be well addressed by using the immersed boundary method, where additional force terms are added to the governing equations to represent the solid boundaries. Using this approach significant computational saving is achieved by two means: 1) Highly efficient solvers can be utilized on Cartesian grids, and 2) Mesh deformation is not necessary. The sensitivity analysis consists of differentiating the continuum form of the governing equations. In this approach the sensitivity and governing equations use the same differential operators, therefore same solver is utilized for the both systems of equations. This methodology is applied to solve flow over a cylinder and a simplified aeroelastic wing model. The sensitivity of flow field around and on the cylinder, lift of the airfoil, and structural response of the wing are calculated to shape parameter. High-fidelity computational fluid dynamic (CFD) is used to solve the flow and the wing structure is represented with linear finite element model. The sensitivity results are verified using the complex step method where it shows a good agreement.

---

\*Graduate Research Assistant, Department of Mechanical and Materials Engineering, AIAA Student Member; gobal.2@wright.edu.

<sup>†</sup>Distinguished Research Professor, AIAA Fellow.

# I. Introduction

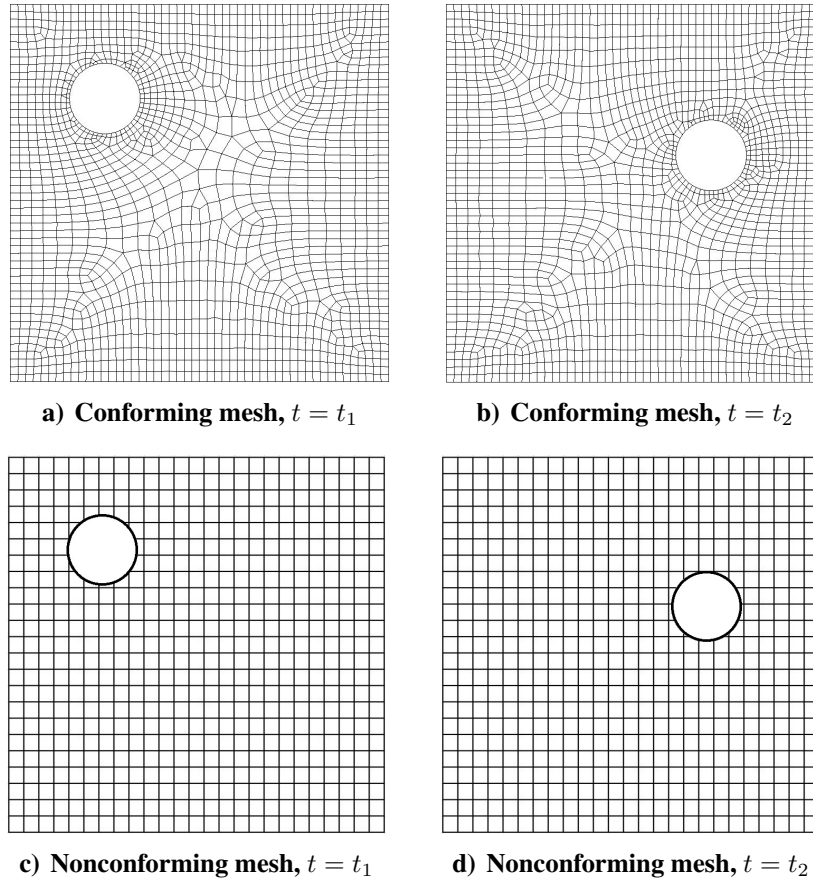
Fluid-structure interaction (FSI) problems play prominent roles in many scientific and engineering fields, yet a comprehensive study of such problems remains a challenge due to their strong nonlinearity and multidisciplinary nature. For most FSI problems, analytical solutions to model equations are impossible to obtain, whereas laboratory experiments are limited in scope. Thus, to investigate the fundamental physics involved in the complex interaction between fluids and solids, numerical simulations are employed. Nevertheless, the prohibitive amount of computations has been one of the major issues in the coupled FSI analysis. When intended to design optimization for large scale problems, it is impractical to employ finite difference sensitivity to perform shape design optimization of the coupled FSI problems. Furthermore, the inaccuracy of finite differencing could result in either convergence difficulty or premature design optimization results.

Formulation of the analytical sensitivity methods requires derivation of analytic sensitivity equations. These equations are the result of differentiating the governing equations with respect to design variables such as shape of the boundaries. Among various methods available for calculating analytical design sensitivities, the discrete analytic sensitivity method is a popular approach. This method is based on differentiating the discretized system of governing equations [1]; in order to differentiate the discretized system of equations, it is necessary to modify the source-code of the black-box CFD/FEA solver [2]. This might not be possible due to lack of availability and the complexity of modifying solver's source code. Moreover, the discrete model has high costs, especially in terms of operation costs and memory requirements [3].

The Continuum sensitivity analysis (CSA) is another approach to get the analytical sensitivities, which involves solving a set of partial differential equations named the continuum sensitivity equations (CSEs). CSA has several computational efficiencies that other sensitivity formulations lack. Aurora and Haug [4], followed by Doms and Mroz [5], were among the first to introduce CSA for structural problems. Stanley and Stewart [6] applied CSA in a fluid mechanics discipline with a goal for aerodynamic design. Pelletier and Etienne have applied CSA to numerous fluid-structure interaction (FSI) problems [7] focused mainly on sensitivities of fluid flow parameters near the structure. Liu and Canfield have employed CSA for shape optimization of nonlinear

structures subject to an aeroelastic gust response [8]. They used the finite element method to solve the potential flow around an airfoil and applied continuum sensitivity to find the sensitivity of pressure coefficient on the airfoil to the maximum camber. In these works, body conforming grids were used to model the flow around the solid bodies. The conforming mesh methods consider the interface conditions as physical boundary conditions, which treat the interface location as part of the solution and requires meshes that conform to the interface. Owing to the movement and/or deformation of the solid structure, re-meshing (or mesh-updating) is needed as the solid boundaries move. Although conforming mesh methods have been widely used in many FSI problems, they are cumbersome, if not impossible, to apply to problems with large deformations [?]. Moreover, when performing sensitivity analysis on a deforming mesh, it is required to calculate the mesh sensitivities as well [9]. This adds to the computational cost of calculating sensitivities. The shortcoming of a robust grid generation and the additional cost of calculating mesh sensitivities motivated an important research effort to develop a method that does not require fluid domain mesh modification for the optimization process.

The non-conforming mesh methods treat the boundary location as constraints imposed on governing equations. As a result, the fluid and solid equations can be conveniently solved independently from each other with their respective grids and re-meshing is not necessary. The distinction between these two types of meshes can be observed in Figure 1, where a solid body (a sphere) is moving in a fluid domain.



**Figure 1. Example of conforming and nonconforming mesh.**

Most non-conforming mesh techniques are based upon the framework of the immersed methods. The basic idea here is to employ a numerically efficient Cartesian grid for the discretization of the fluid domain and to represent the immersed fluid-solid interface by modifying the governing equations. This can be formulated as additional source terms in the momentum equation and can be performed in the continuous setting, i.e. before discretization, or by altering the linear system resulting from discretization in time and space, which is termed discrete forcing [10]. The immersed boundary method approach is very appealing, since it relieves the user from the task of grid generation which can be extremely difficult and tedious for complex configurations. In the immersed boundary method, the boundary of an immersed solid is tracked by Lagrangian markers that are convected by a fluid. Numerically, the communication between the solid and the fluid is obtained by spreading singular forces from the Lagrangian markers to nearby Cartesian grid nodes and interpolating the velocity from nearby Cartesian grid nodes to the Lagrangian markers with

the use of discrete Dirac  $\delta$  functions.

In recent years, the immersed boundary method has been applied to different FSI problems. Choi calculated the surface pressure coefficients for high Reynolds number flow around an NACA0012 airfoil using the immersed boundary technique [11]. Fadlun investigated the capability of this method to simulate high-Reynolds number turbulent flows in complex geometries. Fadlun chose an axisymmetric piston-cylinder assembly with a fixed central valve for this purpose [12]. Mittal developed a sharp interface immersed boundary method for simulating incompressible viscous flow over three-dimensional immersed bodies. He applied it to a highly complex three-dimensional moving body of bluegill sunfish [13]. His method was based on a discrete-forcing scheme that allows for a sharp representation of the immersed boundary. Vanella used the immersed boundary method for solving fluid-structure interaction problems [14]. Vanella tested the accuracy of their approach for the case of two falling plates. Most research that has been done on the immersed boundary focused on the analysis aspects of the method and the sensitivity analysis has not been extensively studied. There has been limited research on the application of immersed boundary for optimization of the internal flows where the penalization technique was used to optimize the shape of the channels for fluid flow [15, 16, 17]; however, the methodology used is only applicable to low Reynolds number flows and does not provide good accuracies near the solid boundaries.

In this research, we use the continuum sensitivity analysis to calculate the sensitivity of coupled fluid-solid interaction problems to shape parameters. The fluid-solid interaction is modeled using the continuum immersed boundary method which is capable of sharp representation of the solid boundaries. The traditional immersed boundary method uses a discrete Dirac function for transferring data between domains. This is not applicable when using continuum sensitivity analysis since the governing equations need to be differentiable. Therefore, a continuous regularized delta function is developed to satisfy this need. The methodology is applied to flow over a cylinder and the weakly aero-structural coupling for a wing model. The sensitivity of the flow over the cylinder, and airfoil is calculated to shape parameters. Using these sensitivity results, we were able to calculate the sensitivity of lift generated by the airfoil and hence the sensitivity of the structural response of the wing to the shape parameter. The results for the sensitivity analysis are verified by

the complex step method.

## II. Numerical Approach

### A. Immersed Boundary Formulation

Viscous and incompressible flow in a cartesian square domain  $\Omega$  containing an immersed boundary, as shown in Figure (2), can be modelled by the Navier-Stokes equations:

$$\rho \left[ \frac{\partial \vec{V}}{\partial t} + \vec{\nabla} \cdot (\vec{V} \otimes \vec{V}) \right] = -\vec{\nabla} P + \mu \nabla^2 \vec{V} + \vec{F} \quad (1a)$$

$$\vec{\nabla} \cdot \vec{V} = 0 \quad (1b)$$

where  $\rho$  is density;  $\mu$  is dynamic viscosity;  $\vec{V}$  is fluid's velocity; and  $P$  is the fluid's pressure.  $\vec{F}$  is given by

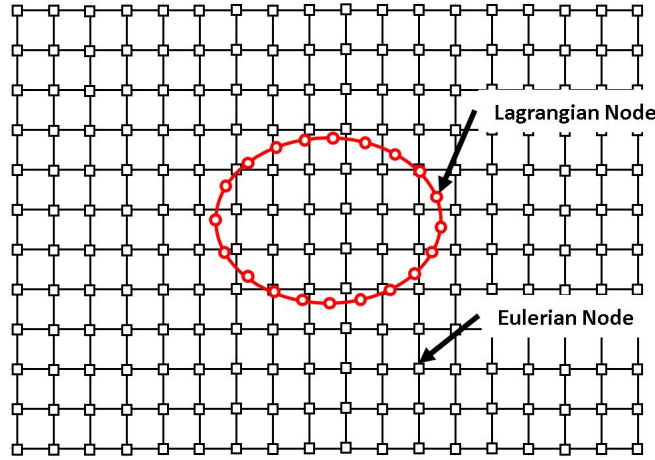
$$\vec{F}(\vec{x}, t) = \int_{\Omega} \vec{f}(\vec{x}_k, t) \delta(\vec{x} - \vec{x}_k) d\vec{x}_k \quad (2)$$

and  $\delta(\vec{x} - \vec{x}_k)$  is a Dirac delta function;  $\vec{x}_k$  are the Lagrangian points placed over the immersed boundary;  $\vec{f}(\vec{x}_k)$  is the Lagrangian force density; and  $\vec{F}(\vec{x})$  is the Eulerian force that has non-zero values over the immersed boundary. Equation (2) models the interaction between the immersed boundary and the fluid flow by injecting a force field on the fluid. The Lagrangian force terms are calculated using virtual boundary formulation that is proposed by Goldstein [18], which was used to simulate turbulent flow over a riblet-covered surface and to address other similar problems. The main idea of this method is to treat the embedded boundary in the fluid by adding a force field to the fluid, taking the same velocity as the surface. Since the force field is not known a priori, it must be calculated in a feedback way such that the velocity on the boundary is used to determine the desired force distribution. This model involves two imposed constants,  $\alpha$  and  $\beta$ , which are chosen to be both negative and large enough in magnitude to bring the fluid velocity close to the interface velocity. The feedback forcing function  $\vec{f}(\vec{x}_k, t)$  can assume the expression as

$$\vec{f}(\vec{x}_k, t) = \alpha \int_0^t \left[ \vec{u}(\vec{x}_k, t) - \vec{U}(\vec{x}_k, t) \right] dt + \beta \left[ \vec{u}(\vec{x}_k, t) - \vec{U}(\vec{x}_k, t) \right] \quad (3)$$

$\vec{u}(\vec{x}_k, t)$  is the value of the response variable from the Eulerian nodes calculated at the Lagrangian nodes.  $\vec{U}(\vec{x}_k, t)$  is the desired value of  $\vec{u}(\vec{x}_k, t)$  as the Lagrangian nodes,  $\vec{x}_k$ . For the case of a no-slip boundary condition where the velocities are zero on the immersed boundary ( $\vec{U}(\vec{x}_k, t) = 0$ ) the forcing function in Equation (3) tries to bring the velocity to zero on the surface of the boundary.  $\vec{u}(\vec{x}_k, t)$  can be calculated using the same delta function used in Equation (2).

$$\vec{u}(\vec{x}_k, t) = \int_{\Omega} \vec{u}(\vec{x}, t) \delta(\vec{x} - \vec{x}_k) d\vec{x} \quad (4)$$



**Figure 2. Immersed boundary illustration**

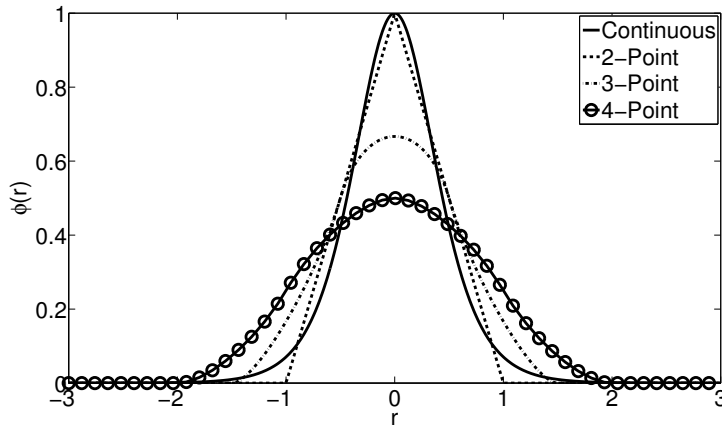
As shown in Equation (2), the delta function is used to transfer quantities between Lagrangian and Eulerian nodes. Shin investigated the stability of virtual boundary formulation for various types of regularized delta functions [19]. However, the delta functions used were not continuously differentiable. This is a requirement for continuum sensitivity analysis approach, since the governing equations need to be differentiated to get the sensitivity equations. Therefore, we propose the following regularized delta function formulation:

$$\delta_h(\vec{x}) = \frac{1}{h^3} \phi\left(\frac{x - x_k}{h}\right) \phi\left(\frac{y - y_k}{h}\right) \phi\left(\frac{z - z_k}{h}\right) \quad (5)$$

where  $h$  is the distance between the Eulerian nodes.  $\phi$  is defined as:

$$\phi(r) = \frac{4e^{-4r}}{(1 + e^{-4r})^2} \quad , \quad r = \frac{x - x_k}{h} \quad (6)$$

The comparison between different delta functions is shown in Figure 3. The delta functions represented have different bandwidth which corresponds to the number of Eulerian points where the effect of forcing function is distributed. The higher 2 of delta function will results in smearing effect near the boundaries. Moreover, the 2, 3, and 4 point delta functions shown in Figure 3 are not continuously differentiable. This results in stability issues for the solution of sensitivity equations. As shown here, the proposed function in Equation (5) is  $C^1$  continuous and has lower bandwidth to ensure the accuracy of the results near the boundary.



**Figure 3. Comparison between the different delta function formulation.**

Finally, the forcing function,  $\vec{F}(\vec{x})$ , can be written by combining equations (2), (3), and (4).

$$\vec{F}(\vec{x}, t) = \int_{\Omega} \left\{ \alpha \int_0^t \left[ \int_{\Omega} \vec{u}(\vec{x}, t) \delta(\vec{x} - \vec{x}_k) d\vec{x} - \vec{U}(\vec{x}_k, t) \right] dt + \beta \left[ \int_{\Omega} \vec{u}(\vec{x}, t) \delta(\vec{x} - \vec{x}_k) d\vec{x} - \vec{U}(\vec{x}_k, t) \right] \right\} \delta(\vec{x} - \vec{x}_k) d\vec{x}_k \quad (7)$$

As shown in the above equation, the integrals on the domain,  $\Omega$ , transfer the data between the Lagrangian and Eulerian domains. The inner integrals transfer the velocities from Eulerian to Lagrangian nodes to calculate the force at Lagrangian node,  $\vec{f}(\vec{x}, t)$ . The outer integral maps the forcing function from the Lagrangian to Eulerian nodes where the governing equations are solved.



This is shown in Figure 4.

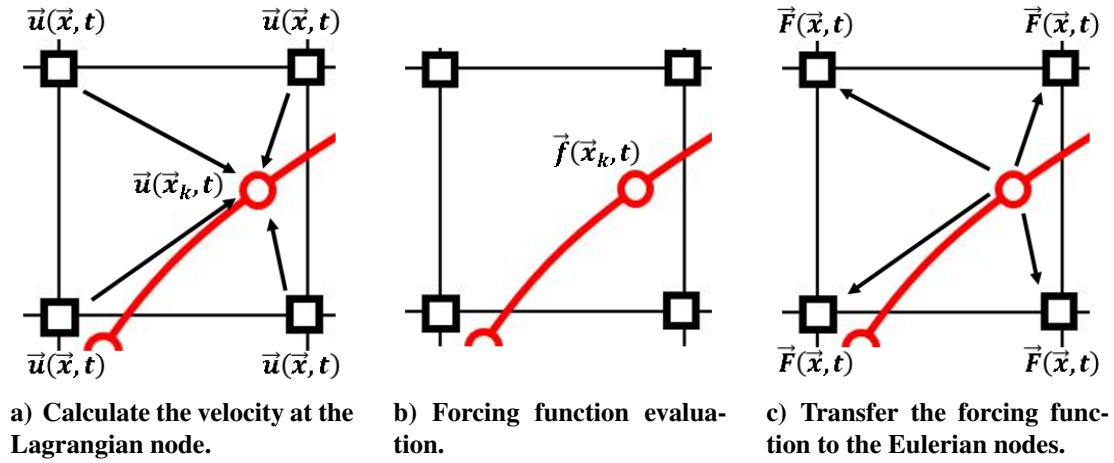


Figure 4. Data transfer between Eulerian and Lagrangian nodes.

## B. Analytic Sensitivity Formulation

Figure 5 represents the domain,  $\Omega$ , in the Cartesian space. This domain is the union of the solid,  $\Omega_s$ , and fluid,  $\Omega_f$ , domains where the Navier-Stokes equations are solved. The boundary conditions are defined on the sides of this domain,  $\Gamma$ . In this research, we assume that these boundary conditions do not depend on the design variables; this simplifies the derivation of the sensitivity equations. It should be noted that by using this approach, the boundaries of the solid domain are added to the fluid's governing equations as mentioned in the previous section.

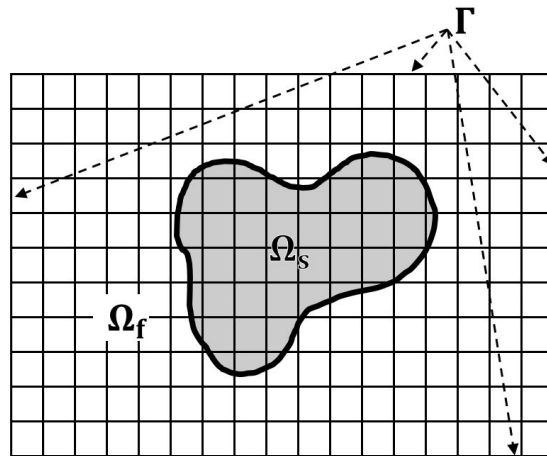


Figure 5. Computational domain,  $\Omega = \Omega_f \cup \Omega_s$ , with boundary of  $\Gamma$

The governing equation and boundary conditions can be written in the general form as

$$A(\vec{u}, \vec{b}) = f(\vec{x}, t; \vec{b}) \quad \text{on } \Omega \quad (8a)$$

$$B(\vec{u}, \vec{b}) = g(\vec{x}, t; \vec{b}) \quad \text{on } \Gamma \quad (8b)$$

where  $A$  and  $B$  are the differential operators for the governing equations and the boundary conditions respectively, and  $\vec{u}$  is the vector of response variables. These can be displacements in structural mechanics or velocities and pressures in fluid dynamics.  $\vec{b}$  and  $\vec{x}$  are vectors of design variables and spatial coordinates, respectively. The total sensitivity of response variable  $\vec{u}$  to the  $i$ th design variable  $b_i$  is calculated as

$$\frac{D\vec{u}}{Db_i} = \frac{\partial \vec{u}}{\partial b_i} + \frac{\partial \vec{u}}{\partial \vec{x}} \cdot \frac{\partial \vec{x}}{\partial b_i} \quad (9)$$

This material derivative consists of the local design derivative,  $\frac{\partial \vec{u}}{\partial b_i}$ , plus a convective term,  $\frac{\partial \vec{u}}{\partial \vec{x}} \cdot \frac{\partial \vec{x}}{\partial b_i}$ . The local design derivative is a measure of how much the response at a point changes due to change in the design parameter. The convective term tracks the movement of the material point when the spatial coordinates will change with the design variable. The geometric domain is a function of design variable in the case of conformal meshes as shown in Figure 1. However, for the case of non-body conformal meshes, the spatial coordinates will not change as the design parameter varies. Therefore, the convective term is equal to zero [20]; this means that the local and total forms of the sensitivities are equal.

The local sensitivity equations are derived by partial differentiation of Equation (1) along with its boundary conditions. In this research, we assume that the boundary conditions are design independent; therefore, their derivatives are zero at the boundaries. This greatly simplifies the derivation of analytical sensitivity equations. The differentiation to  $i$ th design variable,  $b_i$ , yields the following equation for the sensitivity analysis:

$$\rho \left[ \frac{\partial \vec{V}'}{\partial t} + \vec{\nabla} \cdot (\vec{V}' \otimes \vec{V} + \vec{V} \otimes \vec{V}') \right] = -\vec{\nabla} P' + \mu \nabla^2 \vec{V}' + \vec{F}' \quad (10a)$$

$$\vec{\nabla} \cdot \vec{V}' = 0 \quad (10b)$$

In the above equation,  $\vec{V}'$  and  $P'$  are the local sensitivities of the velocity and pressure to  $i$ th design variable,  $b_i$ , respectively.  $\vec{V}$  is calculated from the solution of the governing equation (1). Comparing equations (1) and (10), we can see that the form of the governing and sensitivity equations are the same. Therefore, we can use the same solver for solving both of these equations. This is not possible when using the discrete formulation for the sensitivity analysis as shown in the results section.

The effect of boundary movement is introduced in Equation (10) through  $\vec{F}'$ , more specifically throughout the derivative of the regularized delta function to design variable,  $b_i$ . The derivative of the forcing function is derived as

$$\begin{aligned} \vec{F}'(\vec{x}, t) = & \int_{\Omega} \left\{ \alpha \int_0^t \left[ \int_{\Omega} \vec{u}'(\vec{x}, t) \delta(\vec{x} - \vec{x}_k) d\vec{x} - \int_{\Omega} \vec{u}(\vec{x}, t) \frac{\partial \delta(\vec{x} - \vec{x}_k)}{\partial \vec{x}_k} \frac{\partial \vec{x}_k}{\partial b_i} d\vec{x} \right] dt + \right. \\ & \left. \beta \left[ \int_{\Omega} \vec{u}'(\vec{x}, t) \delta(\vec{x} - \vec{x}_k) d\vec{x} - \int_{\Omega} \vec{u}(\vec{x}, t) \frac{\partial \delta(\vec{x} - \vec{x}_k)}{\partial \vec{x}_k} \frac{\partial \vec{x}_k}{\partial b_i} d\vec{x} \right] \right\} \delta(\vec{x} - \vec{x}_k) d\vec{x}_k + \\ & \int_{\Omega} \left\{ \alpha \int_0^t \left[ \int_{\Omega} \vec{u}(\vec{x}, t) \delta(\vec{x} - \vec{x}_k) d\vec{x} - \vec{U}(\vec{x}_k, t) \right] dt + \right. \\ & \left. \beta \left[ \int_{\Omega} \vec{u}(\vec{x}, t) \delta(\vec{x} - \vec{x}_k) d\vec{x} - \vec{U}(\vec{x}_k, t) \right] \right\} \frac{\partial \delta(\vec{x} - \vec{x}_k)}{\partial \vec{x}_k} \frac{\partial \vec{x}_k}{\partial b_i} d\vec{x}_k \end{aligned} \quad (11)$$

where  $\vec{u}'(\vec{x}, t)$  is the sensitivity of velocity to design parameter,  $b_i$ . As shown in the above equation, we used the chain rule to differentiate the regularized delta function with the design variable,  $b_i$ . The shape of the design variable only affects the location of the Lagrangian points. The derivative of the Lagrangian nodes,  $\vec{x}_k$ , to  $b_i$  can be easily calculated by the problem definition since the dependency of the shape of the domain (Lagrangian nodes) and design variables are known. The derivative of the regularized delta function to the Lagrangian node,  $x_k$ , is calculated analytically by differentiating Equation (6). This results in a system of differential equations for the sensitivity response of the system. This method has a significant advantage to the conventional approaches, since there is no need to calculate the sensitivity of the boundary condition or computational mesh to the design variable. In the next section, we apply this methodology to two different problems.

### III. Demonstration Results

In this section, we will show applications of the continuum sensitivity analysis of flow response to shape design variables. In these problems, the immersed solid boundaries are modelled using the immersed boundary formulation as mentioned in the previous section. We consider two demonstration problems: 1) flow over a cylinder 2) one-way fluid-structure coupling of flow over an airfoil. The flow is modelled as an incompressible, viscous, and laminar flow. The governing equations are discretized as shown in Equation (12).

$$\frac{\vec{V}^{n+1} - \vec{V}^n}{\Delta t} + \left( \frac{3}{2}N\vec{V}^n - \frac{1}{2}N\vec{V}^{n-1} \right) = -Gp^{n+1/2} + \frac{1}{2Re} \left( L\vec{V}^{n+1} + L\vec{V}^n \right) + \vec{F}^n \quad (12a)$$

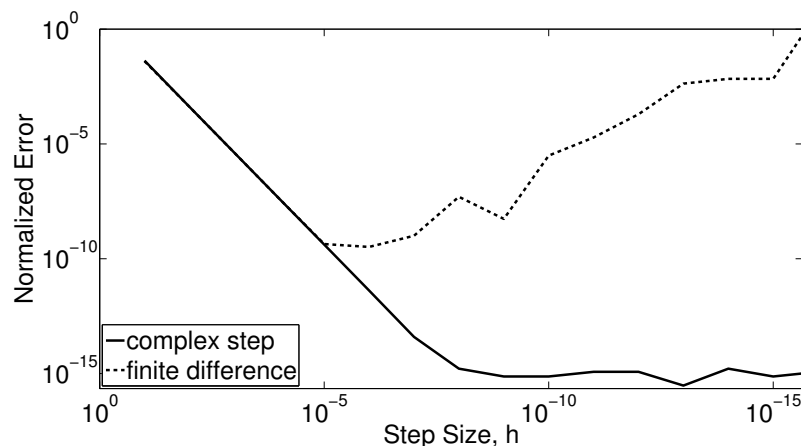
$$D\vec{V}^{n+1} = 0 \quad (12b)$$

In this equation,  $\vec{V}$  is the velocity vector;  $p$  is the pressure;  $Re$  is the Reynolds number;  $\vec{F}$  is the momentum forcing applied to enforce the no-slip boundary condition along the immersed boundary;  $N$  is the discrete convective operator;  $G$  is the discrete gradient operator;  $L$  is the discrete Laplacian operator; and  $D$  is the discrete divergence operator. Here,  $n$  denotes the  $n$ th time step and  $\Delta t$  denotes the time increment. The discrete spatial operators  $N$ ,  $G$ ,  $L$ , and  $D$  are evaluated using the second-order central finite-difference scheme. The numerical method is based on the Navier-Stokes solver, adopting the fractional-step method and a staggered Cartesian grid system. As shown above, the force terms are treated explicitly. We use the projection approach to solve this coupled system of equations [21].

In the following demonstration problems, we used the complex step method to verify the sensitivity results [22]. Complex step differentiation is a technique that employs complex arithmetic to obtain the numerical value of the first derivative of a real valued analytic function of a real variable, avoiding the loss of precision inherent in traditional finite differences. The complex step derivative is calculated as follows:

$$\mathcal{F}'(u; b) = \frac{\text{Im}[\mathcal{F}(u; b + ih)]}{h} \quad (13)$$

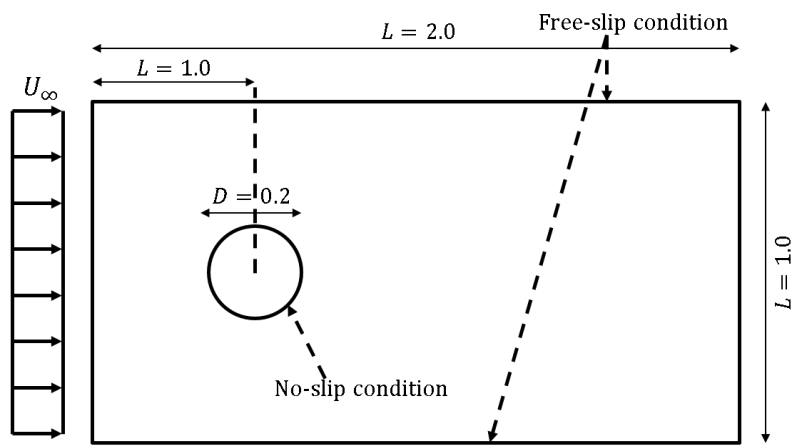
This means that we perturb the design variable by an imaginary value of  $ih$  and then look at the imaginary portion of the resulting response divided by  $h$ . Using the complex step method, we can choose a small step size for  $h$  without losing accuracy. Sensitivity of an analytical function in the form  $F(x) = \frac{e^x}{\cos^3 x + \sin^3 x}$  as  $x = \pi/4$  for complex step and finite difference methods are shown in Figure 6. As shown in Equation (13), the normalized error in the finite difference calculation increases as the step size becomes smaller.



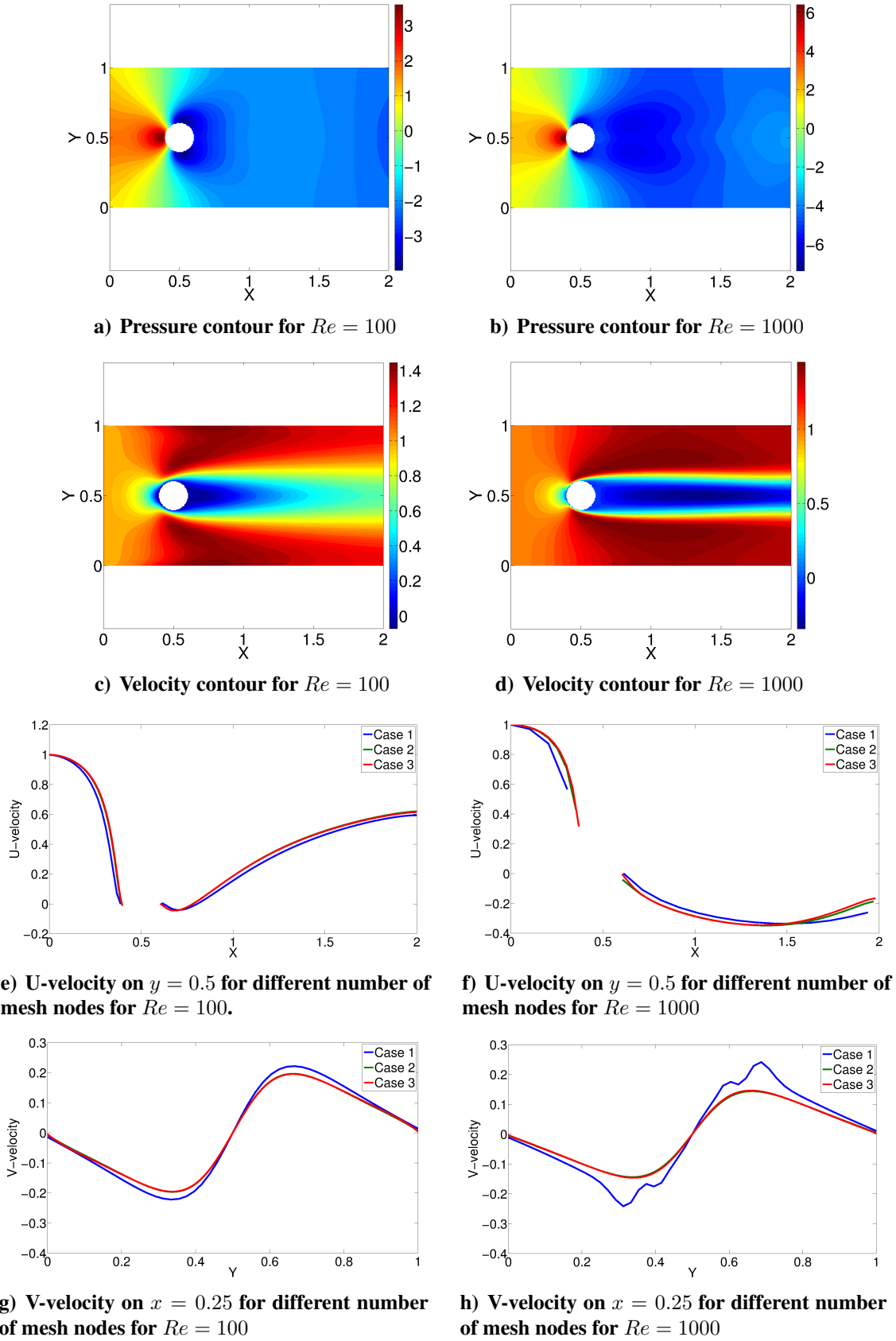
**Figure 6. Relative error in the sensitivity estimates of the finite-difference and the complex-step methods using the analytic result as the reference. Normalized error =  $|F' - F'_{ref}| / F'_{ref}$**

## A. Flow Over a Circular Cylinder

We modelled the flow around a circular cylinder and calculated the sensitivity of the flow to the cylinder radius. The domain is defined as a rectangle with length and height of 2 meters and 1 meter respectively, as shown in Figure 7. The cylinder radius is selected as 0.1 meter and located 1 meter from the inlet. The inlet boundary condition is defined as constant velocity and the outlet is modelled as an outflow boundary condition. This means that the gradient of  $u$  and  $v$  velocities are zero at the outlet. The top and bottom walls are modelled as free-slip boundary conditions. The flow is modelled by choosing the Reynolds number as 100 and 1000. The boundary of the cylinder is modelled using 100 Lagrangian nodes. The flow results and mesh convergence study for different Reynolds numbers are shown in Figure 8. For this analysis, the  $\alpha$  and  $\beta$  parameter of Equation (3) are selected as  $-10^4$  and  $-50$  respectively. As shown in Figure 8, the forcing function satisfied the zero velocity condition on the surface of the cylinder.

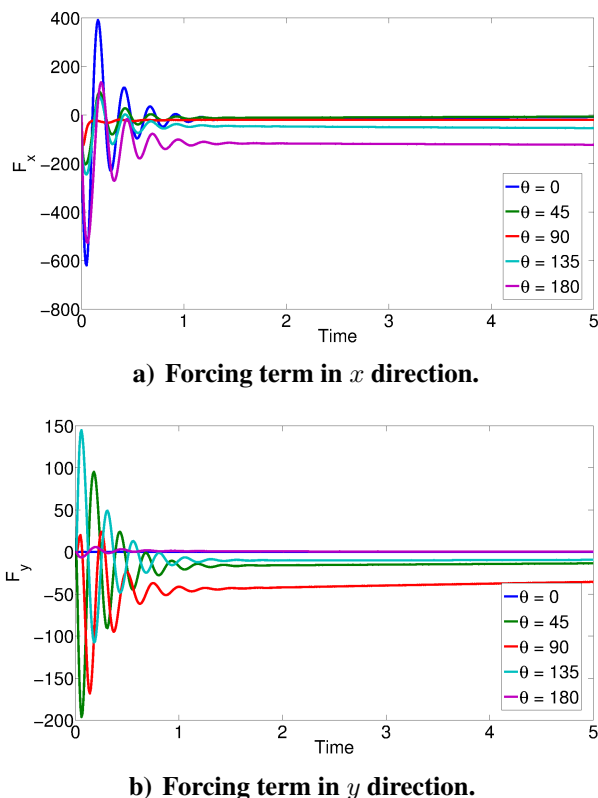


**Figure 7. Physical domain for flow over circular cylinder.**



**Figure 8. Results of convergence study for flow over cylinder. Case 1, case 2, and case 3 are domains with  $100 \times 50$ ,  $200 \times 100$ ,  $300 \times 150$  nodes.**

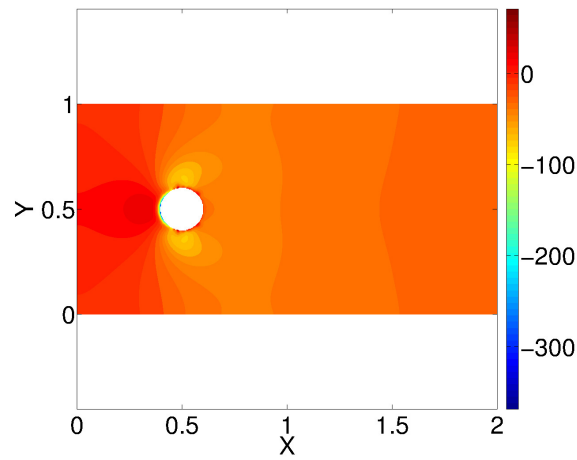
The time history of force terms of the surface of the cylinder are shown in Figure 9. We choose five probe locations on the surface of the cylinder to track the force values. It should be noted that these are the force terms added to the Navier-Stokes equations to represent the boundary, not the forces acting on the surface of the cylinder due to hydrodynamic pressure. As shown in Figure 9, the values of the forcing function starts oscillating at the beginning and reaches a steady-state value after sometime. This indicates that we reached a steady-state solution.



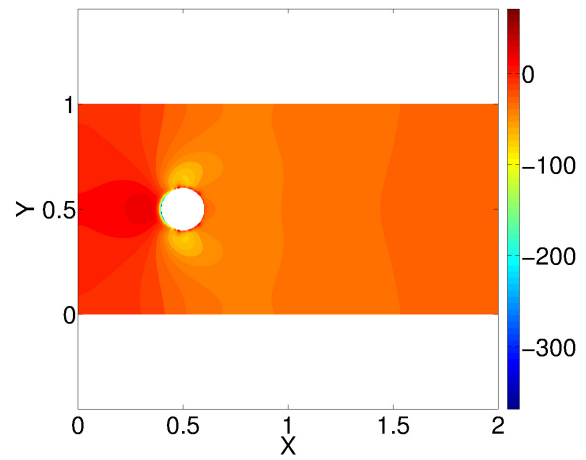
**Figure 9.** Forcing terms on the surface of the cylinder.  $\theta = 0$  is on the front and  $\theta = 180$  is on the back of the cylinder for  $Re = 100$ .

The sensitivity contour plots for the pressure and velocity for different values of Reynolds numbers are shown in Figures 10 and 11. As shown here, the contour plots for the continuum sensitivity analysis and complex step results agree well with each other.

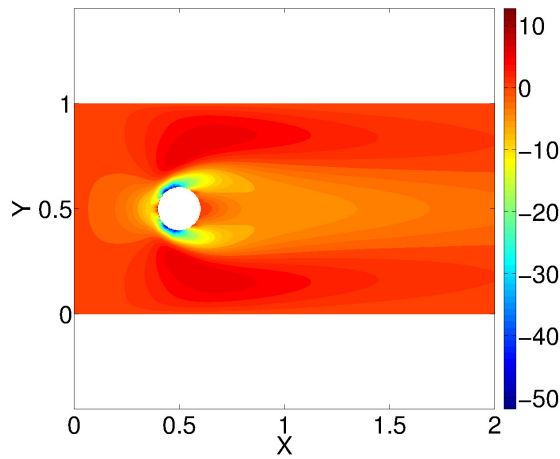




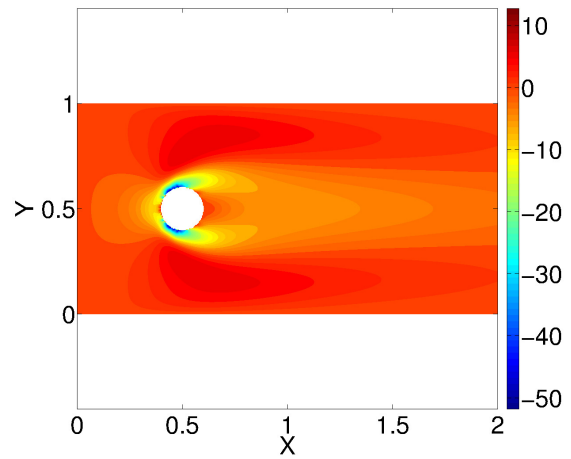
**a) Continuum sensitivity result for pressure sensitivity**



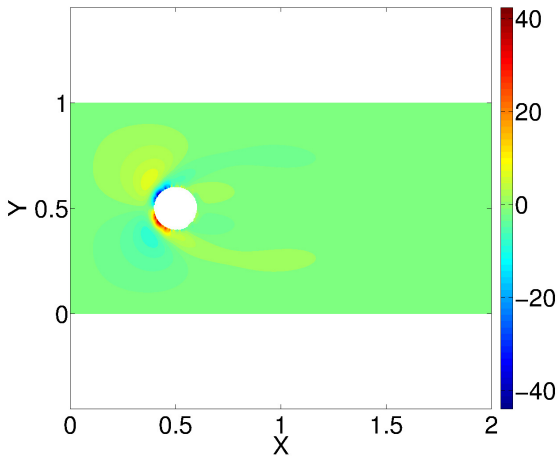
**b) Complex step result for pressure sensitivity**



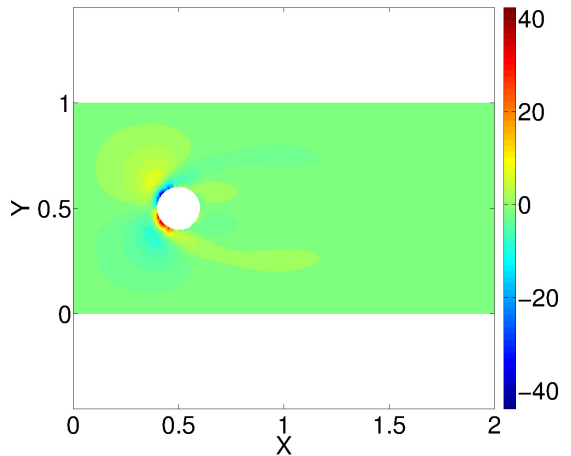
**c) Continuum sensitivity result for u-velocity sensitivity**



**d) Complex step result for u-velocity sensitivity**

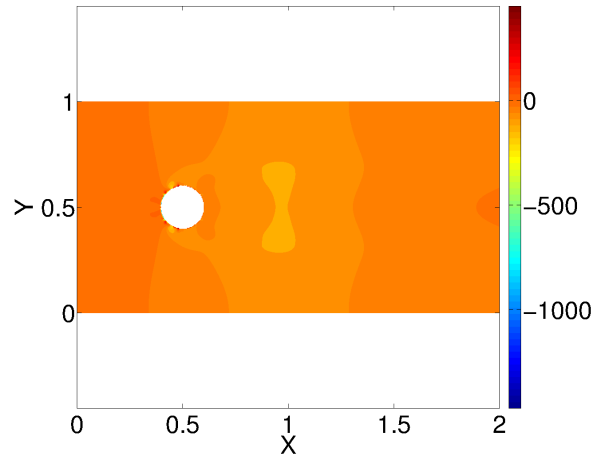


**e) Continuum sensitivity result for v-velocity sensitivity**

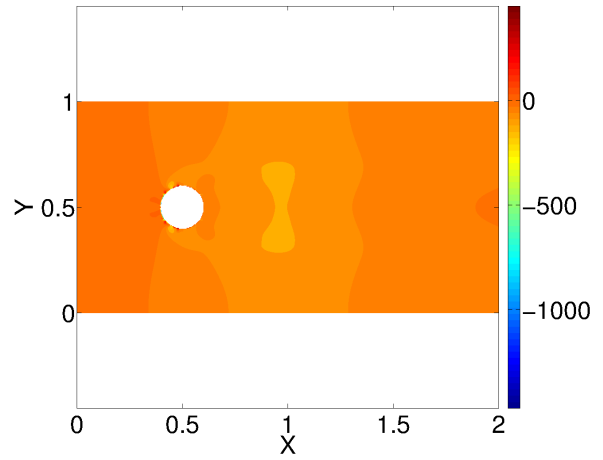


**f) Complex step result for v-velocity sensitivity**

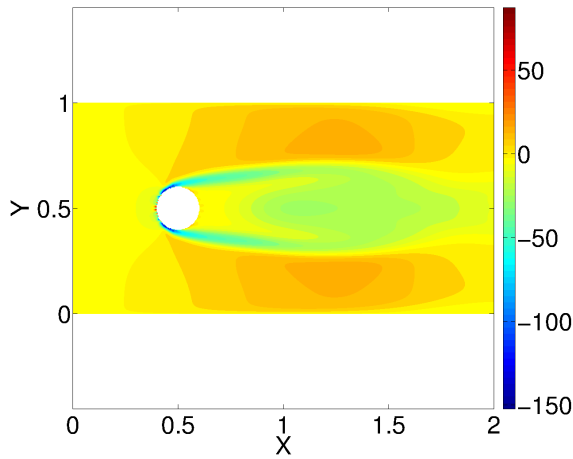
**Figure 10. Comparison of sensitivity contours for flow over cylinder,  $Re = 100$ .**



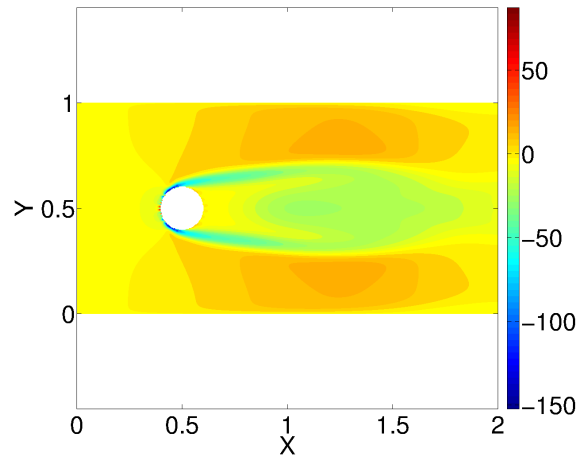
a) Continuum sensitivity result for pressure sensitivity



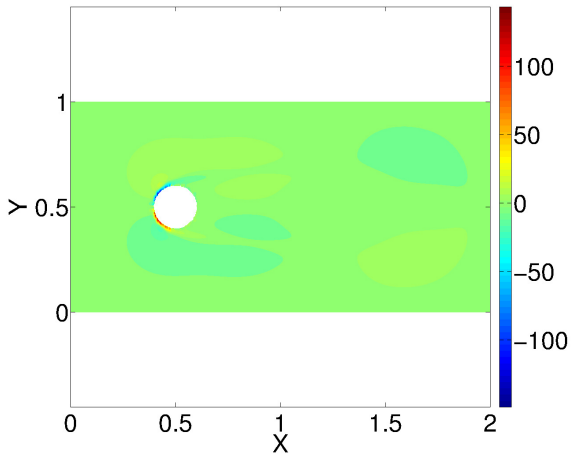
b) Complex step result for pressure sensitivity



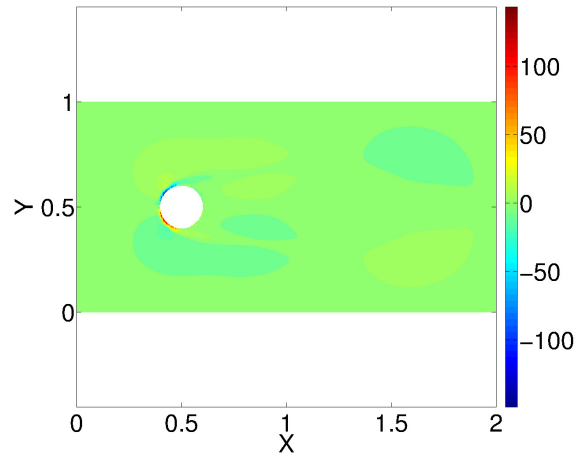
c) Continuum sensitivity result for u-velocity sensitivity



d) Complex step result for u-velocity sensitivity



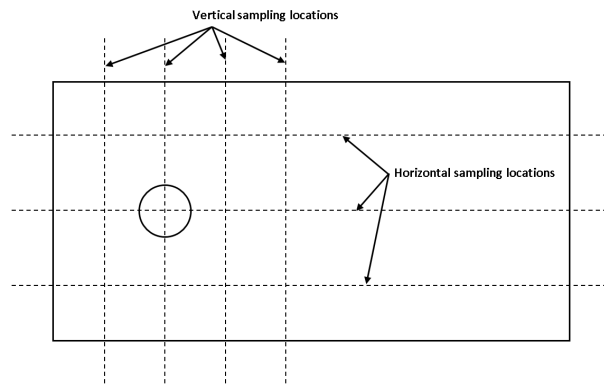
e) Continuum sensitivity result for v-velocity sensitivity



f) Complex step result for v-velocity sensitivity

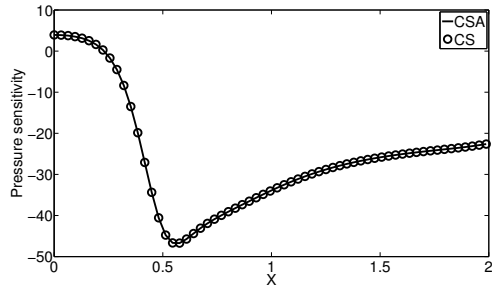
Figure 11. Comparison of sensitivity contours for flow over cylinder,  $Re = 1000$ .

For a better comparison between the sensitivity results of continuum sensitivity analysis and complex step methods, we looked at different locations in the computational domain. These locations are indicated in Figure 12.

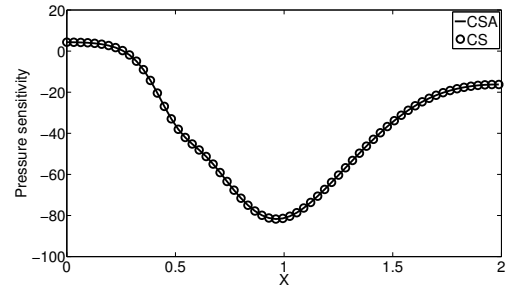


**Figure 12. Sampling locations for the sensitivities**

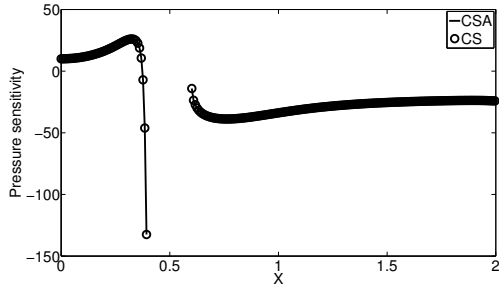
We compared the sensitivity of u-velocity and pressure on a sample location for different values of Reynolds numbers. As shown in Figures 13 and 14, the continuum sensitivity and complex step results agree well with each other.



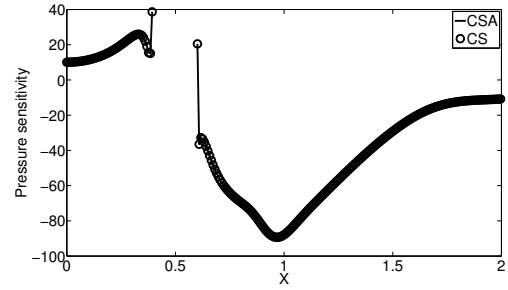
a)  $y = 0.25$ ,  $Re = 100$



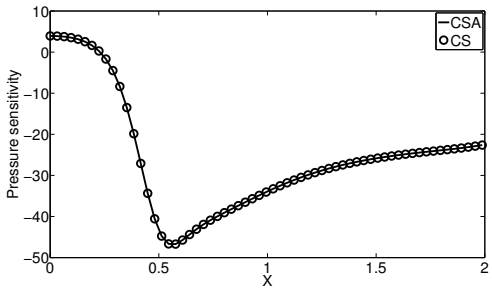
b)  $y = 0.25$ ,  $Re = 1000$



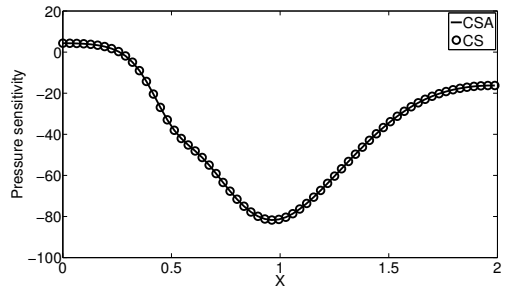
c)  $y = 0.5$ ,  $Re = 100$



d)  $y = 0.5$ ,  $Re = 1000$

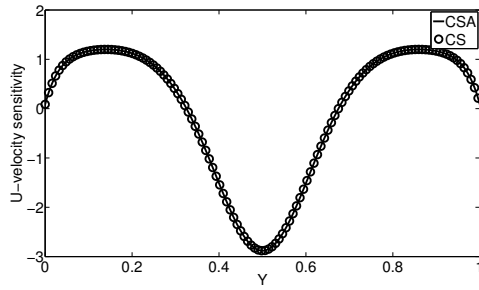


e)  $y = 0.75$ ,  $Re = 100$

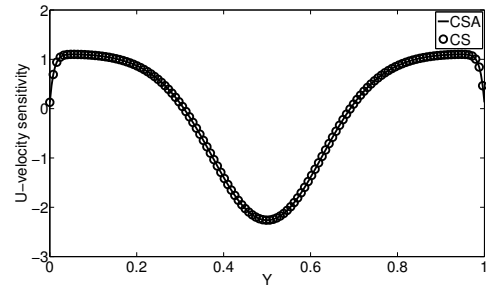


f)  $y = 0.75$ ,  $Re = 1000$

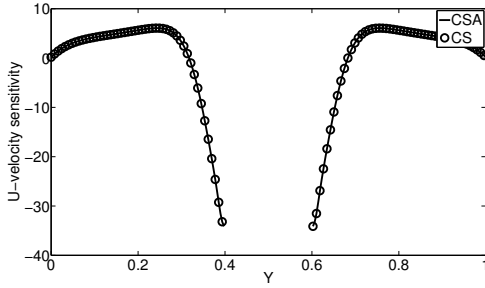
Figure 13. Pressure sensitivity to cylinder radius for different Reynolds numbers on horizontal sampling line.



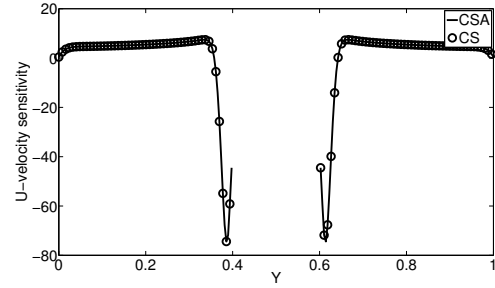
a)  $x = 0.25$ ,  $Re = 100$



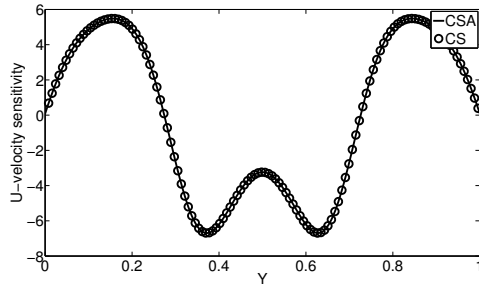
b)  $x = 0.25$ ,  $Re = 1000$



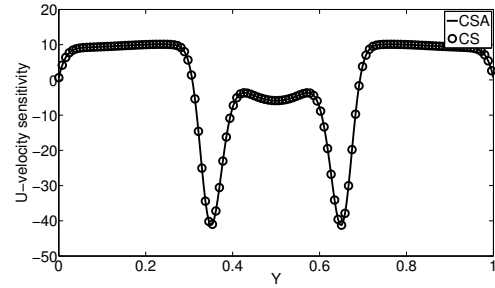
c)  $x = 0.5$ ,  $Re = 100$



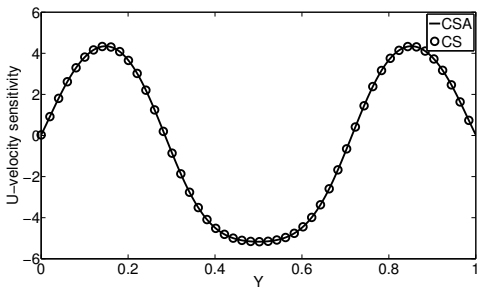
d)  $x = 0.5$ ,  $Re = 1000$



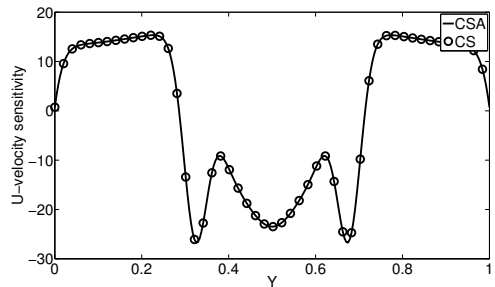
e)  $x = 0.75$ ,  $Re = 100$



f)  $x = 0.75$ ,  $Re = 1000$



g)  $x = 1.00$ ,  $Re = 100$



h)  $x = 1.00$ ,  $Re = 1000$

Figure 14. U-velocity sensitivity to cylinder radius for different Reynolds numbers on vertical sampling line.

## B. Linear Beam with an Airfoil

Laminar flow over a simplified wing is selected as the second demonstration problem. The physical model for this problem is shown in Figure 15. The one-way fluid-solid interaction is defined by mounting the airfoil on an elastic sting. The load and moment from the aerodynamic analysis using CFD are transferred to the structure through the mounting point. The beam is 4  $m$  in length, with a cross-sectional area of  $0.002 \text{ m}^2$ , and modulus of elasticity of 200 GPa. The initial angle of attack is selected at 2 degrees. The Reynolds number for the flow is selected as 100. The boundary conditions are defined as the inflow velocity and zero gradient of velocity at the outlet. The top and bottom walls are modelled as free-slip boundary conditions. The structure is modelled as a linear Euler-Bernoulli beam. We are interested in calculating the sensitivity of the tip displacement of the structure to the shape of the airfoil.

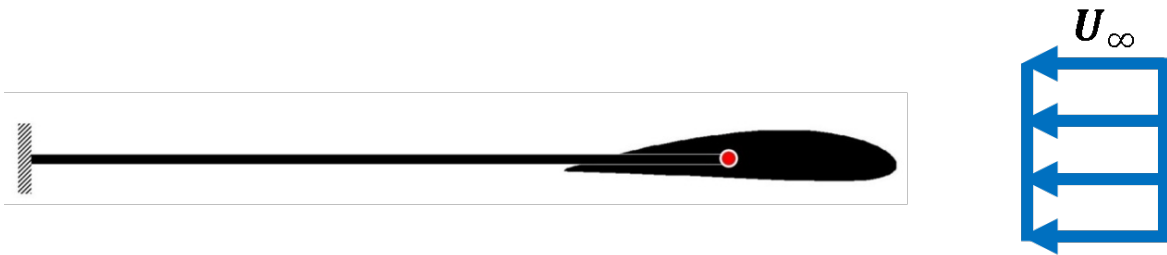


Figure 15. Simplified wing model.

The airfoil is defined using the Joukowski transformation shown in Equation (14); it is possible to map a circle passing through  $z_1 = 1$  and containing the point  $z_2 = -1$  to a curve shaped like the cross section of an airplane wing. The Joukowski transformation is done in a complex plane.

$$J(z) = z + \frac{1}{z} \quad (14)$$

By changing the location of the original circle, it is possible to change the airfoil camber. More importantly, the airfoils have analytical definition that will be used for imposing the force terms to the mesh cells. The effect of change in the location of the original circle to the camber of the airfoil is shown in Figure 16. It should be noted that after generation, the airfoils are normalized such that the chord length is always equal to one.

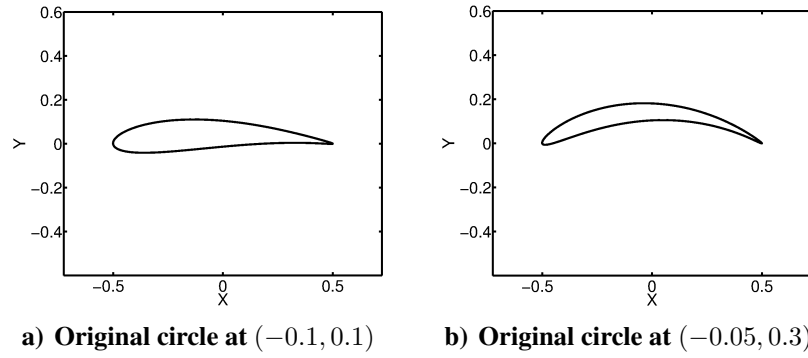


Figure 16. Airfoil definition by Joukowski transformation (not normalized).

The pressure and u-velocity contour plots for the flow over the airfoils shown in Figure 16 are shown in Figure 17. We refer to airfoil shown in Figure 16a as  $J_1$  and We refer to airfoil shown in Figure 16b as  $J_2$ .

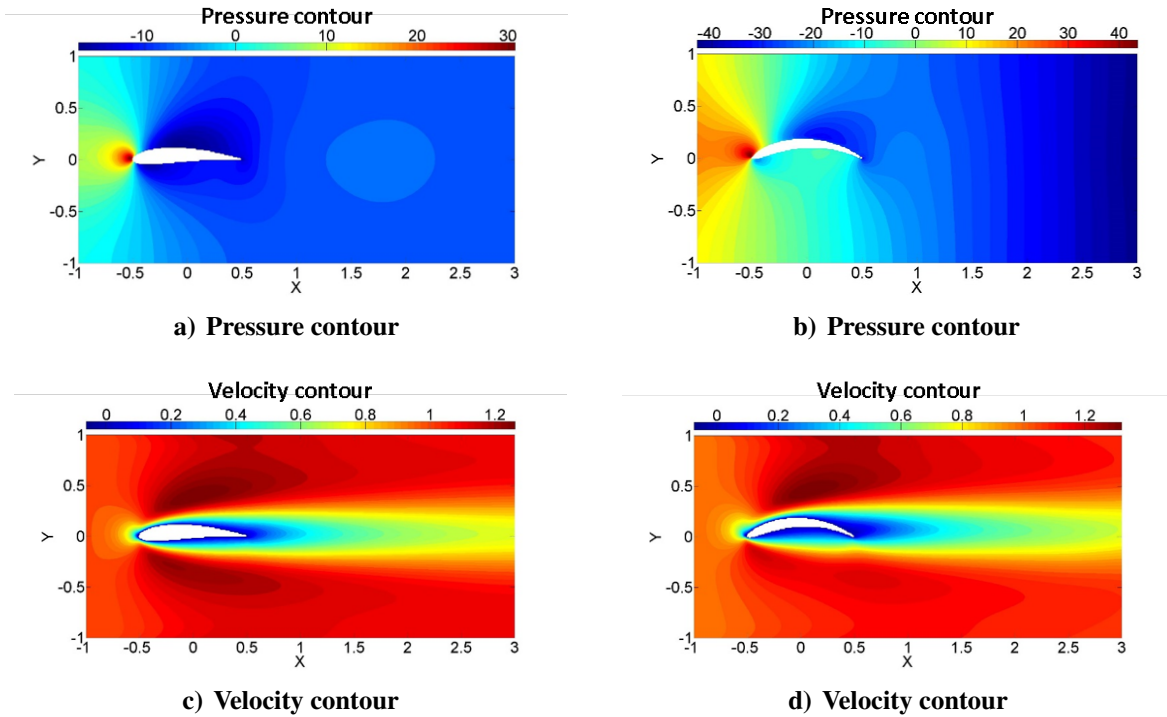
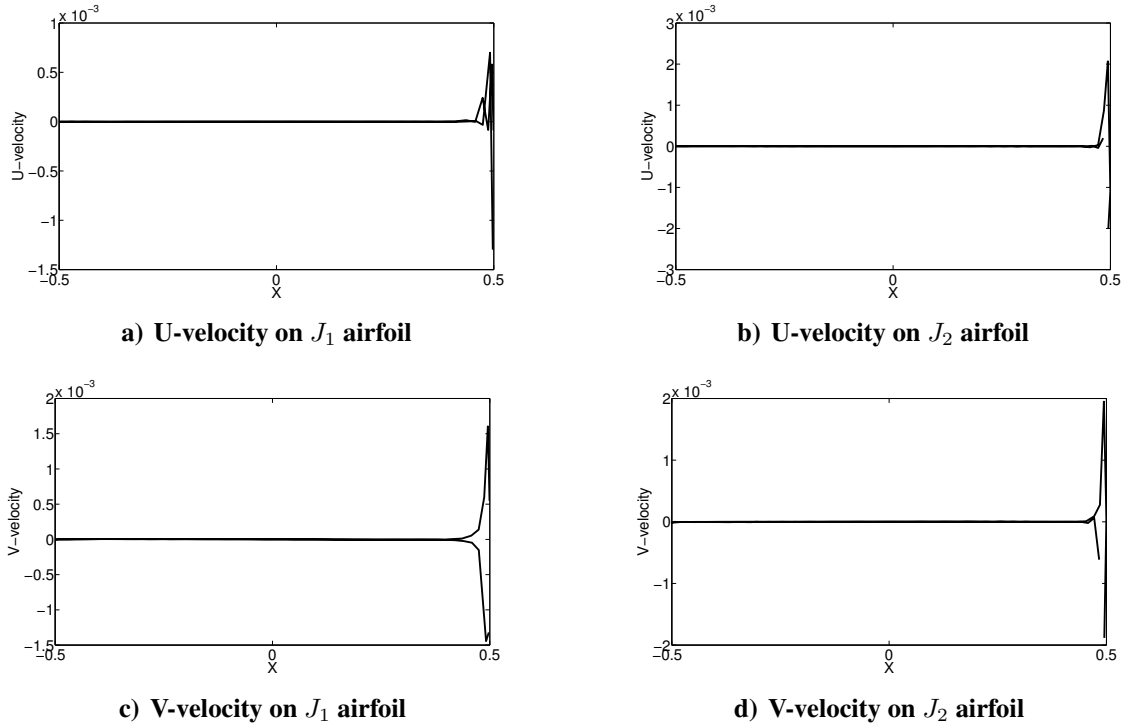


Figure 17. Airfoil definition by Joukowski transformation (not normalized).

The feedback forcing function is able to bring the velocity to zero on the surface of the airfoil. This is shown in Figure 18.

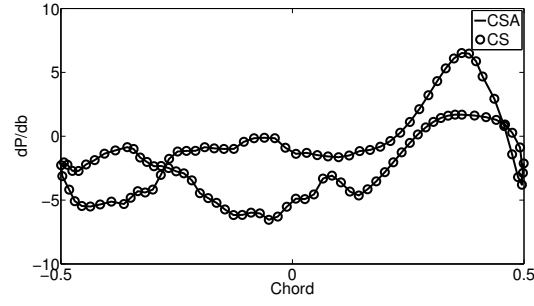


**Figure 18. Flow velocity on the surface of airfoils defined in Figure 16.**

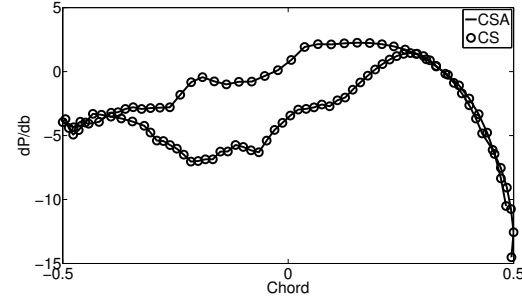
The governing equations, along with the boundary conditions, are differentiated to derive the CSEs as described in the previous section. The resulting system of equations is solved to get the sensitivity response of the system. As shown in Figure 16, the  $y$  coordinate of the center of the original circle defines the camber. The tip displacement is proportional to the load on the structure, which itself depends on the integral of the pressure over the surface of the airfoil. Therefore, to calculate the tip displacement sensitivity, it is required to calculate the pressure sensitivity and integrate it over the airfoil boundary.

The pressure field sensitivity on the top and bottom surfaces of the airfoil to camber line variation is shown in Figure 19. The integral of these functions are used to calculate the sensitivity of tip displacement to the change in airfoil camber. We used complex step method to verify the sensitivity results. As shown in Table 1, the results of the continuum sensitivity analysis agrees well with the complex step results.





a)  $J_1$  airfoil



b)  $J_2$  airfoil

**Figure 19. Pressure sensitivity on the surface of the airfoils.**

	CSA	Complex step	airfoil
$d\delta/dy_c$	0.441	0.441	$J_1$
$d\delta/dy_c$	0.68	0.68	$J_2$

**Table 1. Tip displacement sensitivity to camber**

## IV. Conclusions

In this paper, a continuum sensitivity analysis formulation is developed for the shape sensitivity analysis for the fluid-solid interaction problems. The flow is modelled as an incompressible, laminar, and viscous fluid where the Navier-Stokes equations are used to calculate the fluid response. The solid boundaries are modelled using the continuum formulation of immersed boundary method with an addition of feedback forcing function to the Navier-Stokes equations to represent the effect of solid boundaries. Using this approach we were able to decouple the fluid mesh from the shape of the solid boundary. This results in improving the robustness of the method since no mesh deformation is needed to handle the solid region movement. Moreover, the sensitivity analysis is simplified since the local and total sensitivities are equal for this formulation. A nonlinear mapping

function is used to couple the fluid and solid domain (Eulerian and Lagrangian) grids and to map the data between these domains. As a requirement for continuum sensitivity analysis, the mapping function used need to be  $C^1$  continuous. This is satisfied by proposing a novel regularized delta function that has continuous derivatives. The applicability of the proposed function is showed in this paper.

This methodology is applied to two different demonstration problems. For the first problem. The sensitivity of flow to change in the radius of an immersed cylinder is calculated. The results show a good comparison with the complex step results. It was shown that the method can handle flow with different Reynolds numbers. For the second demonstration problem, the one-way coupling of structural response and the fluid domain is investigated. The sensitivity of the displacement of simplified wing model to the shape of the lifting surface calculated. For this problem, the method was able to calculate accurate sensitivities.

## Acknowledgements

The authors would like to acknowledge the support provided by the Air Force Research Laboratory through contract FA8650-09-23938, the Collaborative Center for Multidisciplinary Sciences.

## References

- [1] Martins, J. R. and Hwang, J. T., “Review and unification of methods for computing derivatives of multidisciplinary computational models,” *AIAA journal*, Vol. 51, No. 11, 2013, pp. 2582–2599.
- [2] Cross, D. M. and Canfield, R. A., “Local continuum shape sensitivity with spatial gradient reconstruction,” *Structural and Multidisciplinary Optimization*, Vol. 50, No. 6, 2014, pp. 975–1000.
- [3] Peter, J. E. V. and Dwight, R. P., “Numerical sensitivity analysis for aerodynamic optimization: A survey of approaches,” *Computers & Fluids*, Vol. 39, No. 3, 2010, pp. 373–391.
- [4] Arora, J. and Haug, E., “Methods of Design Sensitivity Analysis in Structural Optimization,” *AIAA Journal*, Vol. 17, No. 9, 1979, pp. 970–974.

- [5] Dems, K. and Mroz, Z., “Variational approach to first- and second-order sensitivity analysis of elastic structures,” *International Journal for Numerical Methods in Engineering*, Vol. 21, No. 4, 1985, pp. 637–661.
- [6] Stanley, L. G. D. and Stewart, D. L., *Design Sensitivity Analysis: Computational Issues of Sensitivity Equation Methods*, Frontiers in applied mathematics, Society for Industrial and Applied Mathematics (SIAM, 3600 Market Street, Floor 6, Philadelphia, PA 19104), 2002.
- [7] Etienne, S. and Pelletier, D., “A general approach to sensitivity analysis of fluid–structure interactions,” *Journal of fluids and structures*, Vol. 21, No. 2, 2005, pp. 169–186.
- [8] Liu, S. and Canfield, R. A., “Equivalence of continuum and discrete analytic sensitivity methods for nonlinear differential equations,” *Structural and Multidisciplinary Optimization*, Vol. 48, No. 6, 2013, pp. 1173–1188.
- [9] Liu, S. and Canfield, R. A., “Boundary velocity method for continuum shape sensitivity of nonlinear fluid–structure interaction problems,” *Journal of Fluids and Structures*, Vol. 40, No. 0, 2013, pp. 284 – 301.
- [10] Mittal, R. and Iaccarino, G., “Immersed boundary methods,” *Annu. Rev. Fluid Mech.*, Vol. 37, 2005, pp. 239–261.
- [11] Choi, J.-I., Oberoi, R. C., Edwards, J. R., and Rosati, J. A., “An immersed boundary method for complex incompressible flows,” *Journal of Computational Physics*, Vol. 224, No. 2, 2007, pp. 757–784.
- [12] Fadlun, E., Verzicco, R., Orlandi, P., and Mohd-Yusof, J., “Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations,” *Journal of Computational Physics*, Vol. 161, No. 1, 2000, pp. 35–60.
- [13] Mittal, R., Dong, H., Bozkurtas, M., Najjar, F., Vargas, A., and von Loebbecke, A., “A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries,” *Journal of computational physics*, Vol. 227, No. 10, 2008, pp. 4825–4852.

- [14] Vanella, M., Rabenold, P., and Balaras, E., “A direct-forcing embedded-boundary method with adaptive mesh refinement for fluid–structure interaction problems,” *Journal of Computational Physics*, Vol. 229, No. 18, 2010, pp. 6427–6449.
- [15] Challis, V. J. and Guest, J. K., “Level set topology optimization of fluids in Stokes flow,” *International journal for numerical methods in engineering*, Vol. 79, No. 10, 2009, pp. 1284–1308.
- [16] Pingen, G., Waidmann, M., Evgrafov, A., and Maute, K., “A parametric level-set approach for topology optimization of flow domains,” *Structural and Multidisciplinary Optimization*, Vol. 41, No. 1, 2010, pp. 117–131.
- [17] Kreissl, S. and Maute, K., “Levelset based fluid topology optimization using the extended finite element method,” *Structural and Multidisciplinary Optimization*, Vol. 46, No. 3, 2012, pp. 311–326.
- [18] Goldstein, D., Handler, R., and Sirovich, L., “Modeling a no-slip flow boundary with an external force field,” *Journal of Computational Physics*, Vol. 105, No. 2, 1993, pp. 354–366.
- [19] Shin, S. J., Huang, W.-X., and Sung, H. J., “Assessment of regularized delta functions and feedback forcing schemes for an immersed boundary method,” *International Journal for Numerical Methods in Fluids*, Vol. 58, No. 3, 2008, pp. 263–286.
- [20] Gobal, K., Grandhi, R. V., and Kolonay, R. M., “Continuum Sensitivity Analysis for Structural Shape Design Variables Using Finite-Volume Method,” *AIAA Journal*, Vol. 53, No. 2, 2014, pp. 347–355.
- [21] Brown, D. L., Cortez, R., and Minion, M. L., “Accurate projection methods for the incompressible Navier–Stokes equations,” *Journal of computational physics*, Vol. 168, No. 2, 2001, pp. 464–499.
- [22] Martins, J. R., Sturdza, P., and Alonso, J. J., “The complex-step derivative approximation,” *ACM Transactions on Mathematical Software (TOMS)*, Vol. 29, No. 3, 2003, pp. 245–262.