Multiphysics Design Optimization Using an Adjoint Sensitivity Analysis

Stuart A. Walker, Ph.D.
Altair Engineering Inc.
Cincinnati, OH
swalker@altair.com

ABSTRACT

Optimal design methods involving the coupling of fluid and structural solutions are a topic of active research; particularly for aerospace applications. The paper presents a coupled fluid and structure approach to topology optimization using two commercial finite element solutions; AcuSolve and OptiStruct. A gradient based method is used to minimize the compliance of a structure subject to thermal loading. The optimal material distribution to minimize compliance is computed using the Solid-Isotropic Material with Penalty (SIMP) method available in OptiStruct. A volume fraction constraint is imposed in order to iteratively reduce the parts mass. Draw constraints are used to ensure manufacturability. The thermal loading is computed iteratively using a computational fluid dynamics (CFD) solution from AcuSolve. The optimization produces an innovative design which increases the heat rejection rate of the part while reducing the mass.

I. INTRODUCTION

As engineering designs become more complex, there is a need to incorporate multiphysics modeling in the design process. Multiphysics design optimization methods, specifically methods involving the coupling of fluid and structural solutions, are a topic of active research. As these techniques gain confidence, the industrial design process will benefit from a multiphysics optimization approach. The paper presents a coupled fluid and structure approach to topology optimization using a finite element based CFD solution in AcuSolve and a finite element based structural solution in OptiStruct.

II. BACKGROUND

Multiphysics design optimization is a current topic of research in many fields. In the aerospace industry, turbine design and air intake systems have been designed using multiphysics modeling [7,8]. The automotive industry has produced numerous studies citing multiphysics techniques [4,6]. Fluid and structural optimization techniques have been studied extensively [1,2,3,5]. While multiphysics and optimization solutions are widely used in industrial applications, the solutions are rarely coupled thus requiring additional design cycles.

Consider a design space connecting two heat sources with heat removal by a convecting fluid (Figure 1). The optimization problem is to minimize the thermal compliance while reducing the mass. Traditional design process begins by computing the heat removal from the device by the flow field using a CFD solution. The temperature distribution is applied as a thermal load in a topology optimization solution to minimize the thermal compliance. The heat removal by the fluid is then recomputed using the optimized topology of the design space and a new temperature distribution is found. The changes in topology which reduce the thermal compliance are not guaranteed to produce a converged material distribution subject to the convective heat removal constraint. The aim of this paper is to describe a scheme for producing topologic changes which minimize thermal compliance subject to a temperature constraint resulting from fluid convection and solid conduction of the thermal energy.

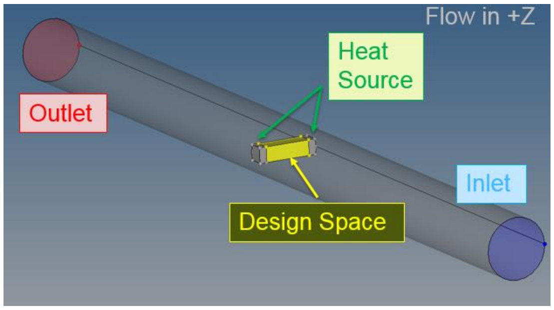

Figure 1: Conceptual multiphysics model.

III. DESIGN OPTIMIZATION METHODS

The opportunity is to minimize the thermal compliance subject to a temperature distribution, T. Define the thermal compliance, J, as [9,10]

$$J = {}^{1}\!/_{2}\, T^{T} f$$

Where f is the thermal loading. The temperature distribution is subject to the constraint

$$F(T,\rho) = \underline{K}T - f = 0$$

Where $\underline{K}$ is the penalized conductivity matrix. Given the constraint on the temperature field, the thermal compliance can be expressed as

$$J = {}^{1}\!/_{2}\, T^{T} \underline{K} T$$

The penalized conductivity varies with the element density according to the solid-isotropic material penalty (SIMP) method so that $\underline{K}$ is defined as

$$\underline{K} = \rho^{p} K$$

With K defining the conductivity matrix, ρ the element density, and p the penalization factor which is always greater than one. The SIMP method uses the density of each element as the design variable so that at each iteration a new density field is defined by

$$\rho_{n+1} = \rho_n + \delta\rho$$

A gradient method or "method of steepest decent" is used to compute the incremental elemental density change, $\delta\rho$, leading to a density field that minimizes the thermal compliance. The perturbation in the density field which acts to minimize a function, J, is given by

$$\delta\rho = -\gamma \frac{\partial J}{\partial \rho}$$

Taking a direct approach to solve the thermal compliance sensitivity function , $\frac{\partial J}{\partial \rho}$, requires a thermal loading solution and finite difference computation for each design variable, i.e. for each change in the continuous design variable, ρ. In a multiphysics context, creating CFD models to compute the temperature field for each design variable change is computationally prohibitive to the industrial design process.

Instead, an adjoint method is used to compute the objective sensitivity. The objective function is augmented by interpolating $F(T,\rho)$ over a dual vector space by the method of Lagrange multipliers. Define the augmented objective function as

$$J_{aug} = J + \int \varphi^T F \, dV$$

Perturbing the augmented objective function about a solution, T, of the constraint equation, F, gives

$$\delta J_{aug} = \left(\frac{\partial J}{\partial T} + \varphi^T \frac{\partial F}{\partial T}\right) \delta T + \int \varphi^T \frac{\partial F}{\partial \rho} \delta\rho \, dV$$

The coefficient of the temperature perturbation is set to zero so that the adjoint solution, φ, is found as the linear solution

$$\left(\frac{\partial F}{\partial T}\right)^T \varphi = -\left(\frac{\partial J}{\partial T}\right)^T$$

Solving the perturbed objective function for the sensitivity yields

$$\frac{\partial J}{\partial \rho} = \varphi^T \frac{\partial F}{\partial \rho}$$

All derivatives are computed analytically so that the objective sensitivity requires the solution of a single linear system. In contrast, recall, the direct method requires non-linear solutions for each design variable change; i.e. at least two non-linear CFD solutions for each model element due to finite differencing.

Special attention must be paid when computing the constraint sensitivity, $\frac{\partial F}{\partial \rho}$. The constraint is linearized about a solution, T, such that the perturbation is given as

$$\delta F|_T = \frac{\partial F}{\partial \rho} \delta\rho + \frac{\partial F}{\partial T} \delta T$$

Taking $\frac{\partial F}{\partial T} = 0$ and recalling the constraint definition, the sensitivity is computed as

$$\frac{\partial F}{\partial \rho} = \frac{\partial \underline{K}}{\partial \rho} T - \frac{\partial f}{\partial \rho}$$

The first derivative on the right hand side is computed analytically. The second derivative is the load sensitivity which is computed analytically for volume and surface loads. Special care must be taken to ensure that load sensitivity is consistent between the CFD and FEA models.

The typical FEA design optimization process is schematically presented in Figure 2. The process begins with a model definition including a design space and thermal loadings. The objective function sensitivity is computed using the adjoint solution to define a new search direction. The new search direction is used to define a new density field. The density field is refined iteratively. Convergence is achieved when the perturbation in the density field is small and the design constraints are met. OptiStruct provides the framework for this solution.

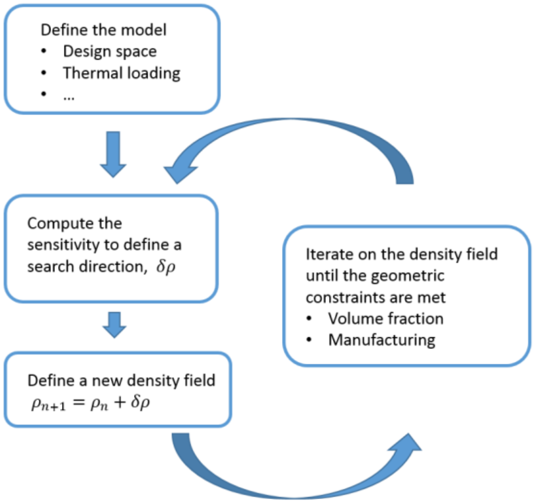

Figure 2: Schematic of a typical FEA design optimization process.

IV. SIMULATION

A schematic of the multiphysics solution is presented in Figure 3. The process begins with a model definition including a design space, thermal loadings, and CFD boundary conditions. The temperature distribution in the design space is computed in AcuSolve subject to volumetric heat sources and conjugate heat transfer with the flow field; recall Figure 1. The compliance is minimized subject to manufacturing constraints and the converged density field is computed. The optimized topology is converted to a new design space using an element wrapping technique. A CFD model is constructed automatically in HyperMesh to compute the new temperature distribution.

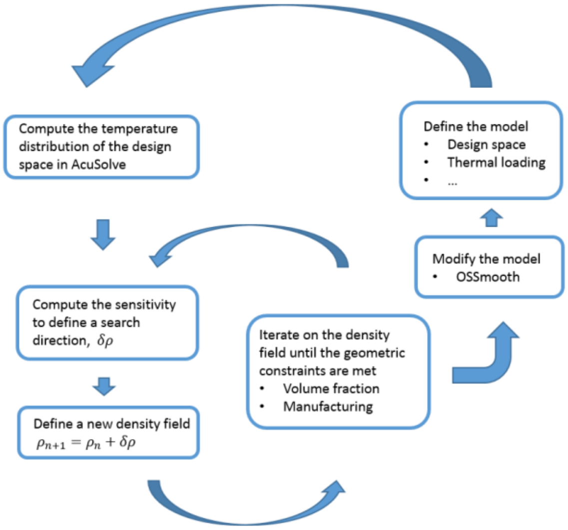

Figure 3: Schematic of the coupled fluid and structure design process.

A high volume fraction constraint is imposed to ensure small incremental changes to the density field in the compliance minimization loop. The volume fraction constraint and a single direction draw manufacturing constraint help to ensure that the temperature loading sensitivity is minimized thus ensuring that the coupled CFD and FEA solution will produce a converged gradient method.

V. RESULTS

The initial design space is shown in Figure 4 (top). The flow is from right to left in the isometric view and from bottom to top in the cross section view. The initial design is a rectangular block spanning two non-design space volumetric heat sources. The optimization is subject to a volume fraction constraint > 0.95 and a single direction draw manufacturing constraint in the cross stream direction. The iterative cross section of the optimized topology is given in Figure 5. After the first iteration, the rectangular block has become streamlined in the upstream direction to facilitate energy transfer in the downstream direction. After iteration 05, the streamlined design has maximized the energy that can be conducted downstream and pockets are formed to facilitate mixing and conjugate heat transfer at the upstream face. The final design is shown in Figure 4 (bottom) and Figure 5 (bottom). The mass reduction for each iteration is shown in Figure 6. The overall mass reduction was ~4%.

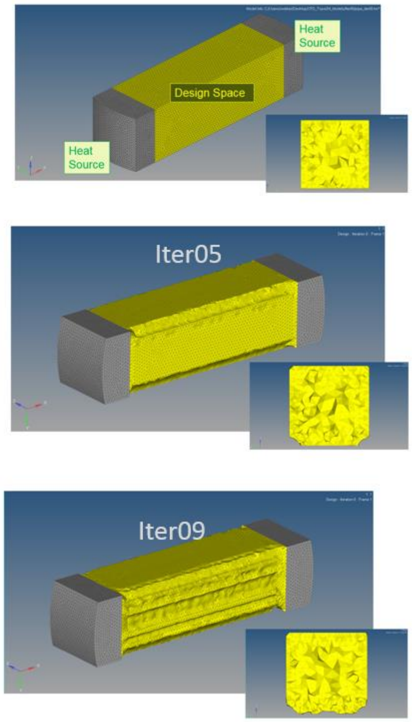

Figure 4: Topology changes of design space.

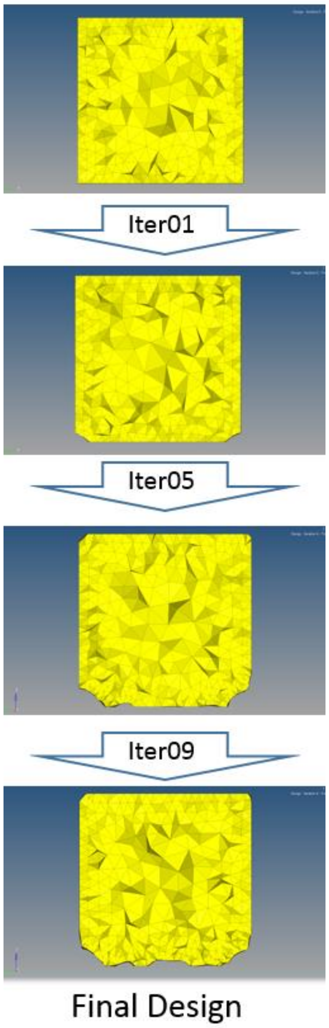

Figure 5: Topology changes of design space; cross section.

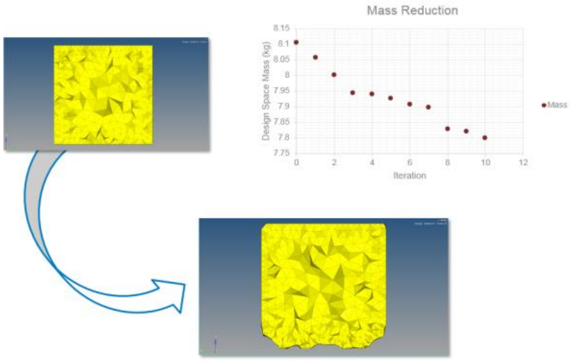

Figure 6: Mass reduction.

VI. CONCLUSION

The paper presents a coupled fluid and structure approach to topology optimization using two commercial finite element solutions; AcuSolve and OptiStruct. It is shown that with appropriate care to loading sensitivities a robust gradient method can be constructed. The FEA design optimization process is modified to include thermal loading from a CFD solution. The design process is demonstrated for a generic design space. The final design minimizes the thermal compliance while showing an overall mass reduction of ~4%.

[1] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, “Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle”.
[2] Arias, S. Automatic CFD Analysis Method for Shape Optimization. Eafit University Medellin 2007.
[3] Jameson, A. Optimum Aerodynamic Design using CFD and Control Theory. AIAA-95-1729-CP.
[4] Naik, S. Design & Shape Optimization of Connecting Rod Using FEA: A Review. International Journal of Engineering and Technical Research. ISSN: 2321-0869. August 2014.
[5] Chang, B., Chen, G., Issa, R. Miller, A. Review of Compsuter Aided Numerical Simulation in Wind Energy. University of New Haven. 2013.
[6] Gobbi, M. et.al. A Critical Review of Optimization Methods for Road Vehicles Design. AIAA 2006-6998.
[7] Choudhary, T. et.al. A Review on Turbine Design and Optimization “A State of the Art”. International Journal of Emerging Technology and Advanced Engineering. Feb. 2014.
[8] Chew, J. et. al. Use of CFD for Thermal Coupling in Aeroengine Internal Air Systems Applications. International Symposium on Fluid Machinery and Fluid Engineer ing. November 2008. 4ISFMFE-IL07.
[9] HyperMesh Reference Manual. Altair Engineering. 2016.
[10] Pagaldipti, N. Qu, X. Thermal Compliance in OptiStruct 14.0. Internal Altair Document . 2016.