ELSEVIER

Comput. Methods Appl. Mech. Engrg. 190 (2001) 5465–5479

Computer methods in applied mechanics and engineering

www.elsevier.com/locate/cma

Design sensitivity analysis and optimization of steady fluid-thermal systems

Dinesh Balagangadhar [a], Subrata Roy [b,*]

[a] *Department of Mechanical Engineering, Michigan State University, East Lansing, MI 48824, USA*
[b] *Department of Mechanical Engineering, Kettering University, Flint, MI 48504, USA*

Received 18 May 2000; received in revised form 29 November 2000

Abstract

Design optimization of fluid-thermal systems has been an area of significant research interest for the aerospace and automotive industry. The subject studies the modification of internal and external flow passages under certain specified objective constraints while satisfying the governing flow equations. Amongst various available optimization procedures the analytical sensitivity analyses-based optimization is arguably the most efficient design tool for complex multi-dimensional practical problems. In this paper, we augmented the analysis capabilities of the computational fluid dynamics (CFD) code with design sensitivity analysis (DSA). The design sensitivities are computed efficiently via analytical differentiation methods. The CFD–DSA codes are then combined with numerical optimization schemes. Finally, CFD–DSA design optimization algorithm is applied to the optimization of heat exchanger fin and HVAC duct systems. © 2001 Elsevier Science B.V. All rights reserved.

Keywords: Design sensitivity analysis; Study state; Shape optimization; Finite element; Weak statement; Fluid thermal systems

1. Introduction

The development of both commercial and research software for the study of computational fluid dynamics (CFD) problems has been an area of continuous endeavour for over two decades. These efforts have improved computational modeling techniques to a level where considerable complexity can be included in the modeling of different physical phenomena. Further, the optimal design of various systems can now be achieved by coupling the CFD codes with numerical optimization schemes.

In numerical optimization methods, fluid flow properties like pressure distribution, temperature, heat flux are calculated for a baseline geometric configuration and are used in defining an objective function to be minimized or maximized. The objective function must relate geometric shape changes to comparable improvements in fluid flow characteristics of the optimum design. However, the cost involved with such optimization schemes are very high due to the computational times involved in the CFD analysis for practical problems.

To elevate this prediction capability to a design tool, it has to be integrated with optimization algorithms and the gradients have to be computed efficiently. In the recent years, there has been significant effort to infuse design sensitivity analysis (DSA) into CFD analysis; gradient-based nonlinear programming algorithms then utilize the sensitivities to perform the optimization efficiently [1–9].

These developments have received considerable attention in the aerospace, automotive, and biomedical industry due to the enormous potential of DSA as a design tool. Led by the studies at NASA Langley

---

[*] Corresponding author. Tel.: +1-810-762-9949.
E-mail address: sroy@kettering.edu (S. Roy).

0045-7825/01/$ - see front matter
PII: S0045-7825(01)00224-9

Research Center in the mid-1980s, DSA has been utilized in aerospace arena for designing various aerodynamic configurations. The sequential implementations of design optimization method was shown to be effective for the design of airfoils, wings, wing-bodies, and complex aircraft configurations using both the potential equation and the Euler equations [1,2]. In [1], the use of advanced, unstructured-grid CFDs in multi-disciplinary analyses and multi-disciplinary sensitivity analyses within design optimization process is proposed. A synergistic two-step design optimization process employing features of structural, aerodynamic, propulsion and flight control systems is developed in [3].

Recently, an optimum aerodynamic design workshop was organized by European Computational Aerodynamics Research Project [4]. Several techniques including inverse methods [5], discrete and continuous sensitivity analysis methods, gradient-based numerical optimization techniques, adjoint methods, multi-point designs, and genetic algorithms with stochastic searches, were applied to a wide range of complex aerodynamic problems. Another part of the workshop [4] documented contributions for a set of benchmark Navier–Stokes problems on parallel distributed-memory computer architecture. Most software were based on multiple-instruction-multiple-data (MIMD) coding using upwind-biased discretization method and unstructured grid.

Similar rapid evolution of design optimization-based CFD analysis is also reported in the automotive, biomedical and electronics industry literature. A design optimization methodology for compact heat exchangers used in the automotive industry, like the radiators and condensers, for two-dimensional fin louver is presented in [6]. Another detailed study of CFD design optimization of finned heat sinks for impingement cooling of electronic packages was done by Kondo et al. [7]. The model allows cost-effective optimization analysis considering 16 design parameters, like fin thickness, spacing and height, and flow orifice dimensions. In [8], numerical optimization has been documented for practical biomedical design constraints. A multi-disciplinary shape optimization using generic algorithms has also been reported for aerodynamic and electro-magnetic applications [9].

From these documented contributions in the literature, it is evident that CFD can and should be used for the purpose of design in order to cut down the cycle time for a new or improved product. However, for a practical problem involving large matrix of candidate designs or design variables, the numerical optimization procedure may require prohibitively expensive computational resources. CFD-based design optimization, without recourse to sensitivity analysis, is thus limited to small-scale problems due to enormous computational costs.

As a means of achieving cost-efficient design optimization using CFD codes with reasonable computational time, we develop CFD codes that analytically compute design sensitivities and integrate them with numerical optimization methods along with the analysis. The algorithm thus developed is then applied to the design optimization of two practical multi-dimensional fluid-thermal systems.

This paper is organized into the following sections. Section 1 is a brief introduction of the state of the art. Section 2 starts with a general description of a multi-dimensional problem statement and its subsequent development of weak statement algebra. Section 3 expands the DSA for the general multi-dimensional governing equation and describes the differences between direct and adjoint formulation. Numerical results are documented in Section 4. An axi-symmetric fin of a heat exchanger has been optimized and compared with genetic algorithm result in Section 4.1. Section 4.2 elaborates the design optimization of a HVAC duct for minimum pressure drop. Finally, conclusions are presented in Section 5.

2. Governing equation for fluid-thermal systems

The Navier–Stokes conservation law system for each state variable $u = u(x_j)$ is of the form [10]

$$\frac{\partial}{\partial x_j}\left(f_j - f_j^v\right) - s = 0 \quad \text{on } \subset \mathbb{R}^d,\ 1 \leqslant j \leqslant d, \tag{1}$$

where $f_j = f(a_j, u)$ and $f_j^v = f(\epsilon(\partial u/\partial x_j))$ are the kinetic and dissipative flux vectors, respectively, the convection velocity is a_j, $\epsilon > 0$ is the diffusion coefficient that varies parametrically, u can be density, velocity components, temperature, kinetic energy and dissipation, and s is a source. Appropriate initial and boundary conditions close system (1) for the well-posed statement.