TFFB coupling BEM/FVM algorithm for multi-dimensional conjugate heat transfer

E. Divo¹, E. Steinthorsson² & A.J. Kassab¹
¹Mechanical, Materials and Aerospace Engineering Department, University of Central Florida, USA
²Parker-Hannifin Corporation, Gas Fuel Systems Division, USA

Abstract

We develop a BEM-based temperature forward/flux back (TFFB) coupling algorithm to solve the conjugate heat transfer (CHT) which arises naturally in analysis of systems exposed to a convective environment. Here, heat conduction within a structure is coupled to heat transfer to the external fluid which is convecting heat into or out of the solid structure. There are two basic approaches to solving coupled fluid-structural systems. The first is a direct coupling where the solution of the different fields is solved simultaneously in one large set of equations. The second approach is a loose coupling strategy where each set of field equations is solved to provide boundary conditions for the other. The equations are solved in turn until an iterated convergence criterion is met at the fluid-solid interface. The loose coupling strategy is particularly attractive when coupling auxiliary field equations to computational fluid dynamics codes. We adopt the latter method in which the BEM is used to solve heat conduction inside a structure which is exposed to a convective field which in turn is resolved by solving the Navier-Stokes equations by finite volume methods. Interface of flux and temperature is enforced at the solid/fluid interface.

1 Introduction

As field solvers have matured, coupled field analysis has received much attention in an effort to obtain computational models ever-more faithful to the physics being modeled. The coupled field problem which we address is the conjugate heat transfer (CHT) problem arising commonly in practice: time dependent or time independent convective heat transfer over coupled to conduction heat transfer.
within a solid body. Examples of CHT include analysis of automotive engine blocks, fuel ejectors, cooled turbine blade/vanes, nozzle or combustor walls, or thermal protection system for re-entry vehicles. Applications of interest are then any thermal system in which multi-mode convective/conduction heat transfer is of particular importance to thermal design, and thus CHT arises naturally in most instances where external and internal temperature fields are coupled.

However, conjugacy is often ignored in numerical simulations. For instance, in analysis of turbomachinery, separate flow and thermal analyses are typically performed, and, a constant wall temperature or heat flux boundary condition is typically imposed for the flow solver. Convective heat transfer coefficients are then obtained from the flow solution, and these are provided to the conduction solver to determine the temperature field and eventually to perform further thermal stress analysis. The shortcomings of this approach which neglects the effects of the wall temperature distribution on the development of the thermal boundary layer are overcome by a CHT analysis in which the coupled nature of the field problem is explicitly taken into account in the analysis.

There are several algorithmic approaches which can be taken to solve the conjugate problem. Most methods are based on either finite elements (FEM) or finite volume methods (FVM), see for instance Comini et al.[1], Shyy and Burke [2], Patankar [3], and Kao and Liou [4], and in either case, require complete meshing of both fluid and solid regions while enforcing solid/fluid interface continuity of fluxes and temperatures.

A different approach taken by Li and Kassab [5,6] and Ye et al. [7] develops a CHT algorithm which does not require meshing of the solid region to resolve the heat conduction problem. In particular, the method couples the boundary element method (BEM) to an FVM Navier-Stokes solver to solve a steady state compressible subsonic CHT problem over cooled and uncooled turbine blades. Due to the boundary-only discretization nature of the BEM, the onerous task of grid generation within intricate solid regions is avoided. Here, the boundary discretization utilized to generate the computational grid for the external flow-field provides the boundary discretization required for the boundary element method. In cases where the solid is multiply-connected, such as a cooled turbine blade, the interior boundary surfaces must also be discretized; however, this poses little additional effort. Moreover, in addition to eliminating meshing the solid region, this BEM/FVM method offers an additional advantage in solving CHT problems which arises from the fact that nodal unknowns which appear in the BEM are the surface temperatures and heat fluxes. Consequently, solid/fluid interfacial heat fluxes which are required to enforce continuity in CHT problems are naturally provided by the BEM conduction analysis. This is in contrast to the domain meshing methods such as FVM and FEM where heat fluxes are computed in a post-processing stage by numerical differentiation.

We now present work which extends the loosely coupled BEM/FVM approach to solving CHT for steady state analysis of three dimensional problems. Here, we couple a BEM conduction solver with the NASA Glenn multi-block FVM Navier-Stokes convective heat transfer code, Glenn-HT. The methodology adopted in this work is given as well as the information passing process. Results are presented from a preliminary study carried out for a test-case configuration to
be used in a future laboratory experiment which will serve as experimental validation of the CHT solver.

2 Governing equations

2.1 Flow regime

The governing equations for fluid flow and heat transfer are the compressible Navier-Stokes equations, which describe the conservation of mass, momentum and energy. These can be written in integral form as

$$\int_\Omega \frac{\partial \mathbf{W}}{\partial t} \, d\Omega + \int_\Gamma \left( \mathbf{F} - \mathbf{T} \right) \cdot \mathbf{n} \, d\Gamma = \int_\Omega \mathbf{S} \, d\Omega$$  \hspace{1cm} (1)

where $\Omega$ denotes the volume, $\Gamma$ denotes the surface bounded by the volume $\Omega$, $\mathbf{n}$ is the outward-drawn normal, $\mathbf{W} = (\rho, \rho u, \rho v, \rho w, \rho e, \rho k, \rho \omega)$ is the vector of conserved variables, where, $\rho, u, v, w, e, k, \omega$ are the density, the velocity components in $x$-, $y$-, and $z$-directions, the specific total energy and $k$ and $\omega$ are the kinetic energy of turbulent fluctuations and the specific dissipation rate of the two equation $k$-$\omega$ Wilcoxon turbulence model adopted in Glenn-HT. The vectors $\mathbf{F}$ and $\mathbf{T}$ are convective and diffusive fluxes respectively, $\mathbf{S}$ is a vector containing all terms arising from the use of a non-inertial reference frame as well as production and dissipation of turbulent quantities. The fluid is modeled as an ideal gas. A rotating frame of reference can be adopted for rotating flow studies. The effective viscosity and thermal conductivity are given by a Prandtl number analogy where

$$\mu = \mu_l + \mu_t \hspace{0.5cm} k = \frac{\gamma}{\gamma - 1} \left[ \frac{\mu_l}{Pr_l} + \frac{\mu_t}{Pr_t} \right]$$  \hspace{1cm} (2)

where $\mu_l = \rho k/\omega$, $Pr$ is the Prandtl number, and the subscribes $l$ and $t$ refers to laminar and turbulent values respectively.

2.2 Heat conduction

In the CHT model, the NS equations are solved to steady state by a time marching scheme. As physically realistic time-dependent solutions are not sought, a quasi-steady heat conduction analysis using the BEM is performed at a given time level. As such, only steady-state heat conduction is considered. The governing equation is

$$\nabla \cdot [k(T_s) \nabla T_s] = 0$$  \hspace{1cm} (3)

where, $T_s$ denotes the temperature of the solid, and $k_s$ is the thermal conductivity of the solid material. If the thermal conductivity is taken as constant, then the above reduces to the Laplace equation for the temperature. When the thermal conductivity variation with temperature is an important concern, the nonlinearity
of the heat conduction equation can readily be removed by introducing the Kirchhoff transform, \( U(T) \), which is defined as

\[
U(T) = \frac{1}{k_o} \int_{T_o}^{T} k_s(T) dT
\]

where \( T_o \) is the reference temperature and \( k_o \) is the reference thermal conductivity. The transform and its inverse are readily evaluated, either analytically or numerically, and the heat conduction equation transforms to a Laplace equation for the transform parameter \( U(T) \). The heat conduction equation thus reduces to the Laplace equation in any case, and this equation is readily solved by the BEM.

In the conjugate problem, continuity of temperature and heat flux at the blade surface, \( \Gamma \), must be satisfied

\[
T = T_s \quad \text{and} \quad k \frac{\partial T}{\partial n} = k_s \frac{\partial T_s}{\partial n}
\]

Here, \( T \) is the temperature computed from the N-S solution, \( T_s \) is the temperature within the solid which is computed from the BEM solution, and \( \partial / \partial n \) denotes the normal derivative. Both boundary conditions transform linearly in the case of temperature dependent conductivity.

### 3 Solution Algorithm

A brief description of the Glenn-HT code is given in this section. Details of the code and its verification can be found in [8,9]. The heat conduction equation is solved using BEM.

#### 3.1 Navier-Stokes solver

A cell-centered FVM is used to discretize the NS equations. Eq. 1 is integrated over a hexahedral computational cell with the nodal unknowns located at the cell center \((i,j,k)\). The convective flux vector is discretized by a central difference supplemented by artificial dissipation as described in Jameson et al. [10]. The artificial dissipation is a blend of first and third order differences with the third order term active everywhere except at shocks and locations of strong pressure gradients. The viscous terms are evaluated using central differences. The resulting finite volume equations can be written as

\[
V_{i,j,k} \frac{d W_{i,j,k}}{dt} + q_{i,j,k} = \ell_{i,j,k}
\]

where \( W_{i,j,k} \) is the cell-volume averaged vector of conserved variables, \( q_{i,j,k} \) and \( \ell_{i,j,k} \) are the net flux and dissipation for the finite volume obtained by surface integration of Eq. (1), and \( \ell_{i,j,k} \) is the net finite source term. The above is solved using a time marching scheme based on a fourth order explicit Runge-Kutta time stepping algorithm. Local time-stepping and implicit residual
smoothing are used to accelerate convergence. A multi-grid option is available in the code, although it was not used in the results reported herein.

3.2 Heat conduction boundary element solution

The heat conduction equation reduces to the Laplace equation in the temperature or the Kirchhoff transform. It is solved using the BEM element method with isoparametric bilinear elements. Upon discretization the standard BEM equations are obtained: \([H]{T_s} = [G]{q_s}\). These are solved subject to either of the following boundary conditions:

(a) at the external bounding wall: \(T_s\Gamma_e = T\) \(\text{ (7)}\)
(b) at internal cooling hole surfaces: \(k_s \frac{dT}{dn}\big|_{\Gamma_c} = h\big(T_s - T_{\infty}\big)\big|_{\Gamma_c}\) \(\text{ (8)}\)

Here, \(T\) is the temperature computed from the N-S solution, \(T_s\) is the temperature computed from the BEM solution, \(\Gamma_e\) denotes the external boundary, while \(\Gamma_c\) denotes the convective internal cooling hole boundaries. In this preliminary study the flow solution within cooling holes is not simulated using the NS solver (although this is perfectly possible and would be consistent with the CHT philosophy). The advantage of the BEM formulation over finite difference or finite element formulations is that no interior mesh is generated and the surface heat flux is computed in the solution. A GMRES iterative solver with an ILU pre-conditioning is used to solve the BEM equations.

3.3. CHT algorithm

The Navier-Stokes equations for the external fluid flow and the heat conduction equation for heat conduction within the solid are interactively solved to steady state through a time-marching algorithm. The surface temperature obtained from the solution of the Navier-Stokes equations is used as the boundary condition of the boundary element method for the calculation of heat flux through the solid surface. This heat flux is in turn used as a boundary condition for the Navier-Stokes equations in the next time step. This procedure is repeated until a steady-state solution is obtained. In practice, the BEM is solved every few cycles of the FVM, say every two hundred, to update the boundary conditions, as intermediate solutions are not physical in this scheme. This is referred to as the temperature/flux back coupling (TFBC) algorithm as outlined below:

- FVM Navier-Stokes solver:

  1. Begins with initial adiabatic boundary condition at solid surface.
  2. Solves compressible NS for fluid region.
  3. Provides temperature distribution to BEM conduction solver after a number of iterations.
  4. Receives flux boundary condition from the BEM as input for next set of iterations.
BEM conduction solver:

1. Receives temperature distribution from FVM solver.
3. Provides flux distribution to FVM solver.

The transfer of heat flux from the BEM to the FVM solver is accomplished after under-relaxation.

\[ q = \omega q_{old}^{BEM} + (1 - \omega)q_{new}^{BEM} \]  

(9)

with \( \omega \) taken as 0.2. Furthermore, the FVM surface grid is much too fine to be used in the BEM. Thus a much coarser grid was used for the BEM. At each BEM node, temperatures were computed from a distance-weighted average of the nearest ten FVM surface nodes.

4 Numerical results

We now report on a preliminary simulation used to verify our algorithm. A 3-D model is made of a 2-D configuration used in an experiment set up to simulate heat transfer conditions in a cooled turbine blade tip and investigate the importance of conjugacy, see Fig. 1. Heated air at 319.5K enters horizontally at the left end of the channel, flows over the simulated tip gap, and out the bottom of the channel. The block is made of stainless-steel \((k = 14.9 \, W/mK, \rho = 8.03 \times 10^3 kg/m^3, c = 502.48 \, J/kgK)\) and is cooled by a laminar flow of water at 286K. The film coefficient in the cooling channels is calculated as 536 \( W/m^2K \). In this simulation, convective boundary conditions are used to model flow in the channel as a full external flow and internal flow CHT solution was not carried out, due to the fact that the CFD code is specialized for turbomachinery applications and can only model air as a working fluid. All walls not exposed to the flow are adiabatic. The height of inlet channel is 0.02m, the height of the outlet channel is 0.025m, width of passage between block and wall is 0.005m, the length of inlet channel up to block is 0.403m, the length of outlet channel measured from block to exit is 0.40615m. Total pressure at inlet is atmospheric and the back pressure at exit is 0.92 \times \text{inlet total pressure} \text{ (i.e., } p/P_{\text{inlet}} = 0.92\text{).} A 3-D model was constructed to model the centerline of the block. Four finite volume cells were used to model the width and the surface grid at the block are shown in Figs. 2 and 3. The FVM mesh uses 1104 surface cells, 200,000 total finite volume cells, the BEM grid used 946 uniformly spaced boundary elements. The CHT solution was run to reduce residuals in density to \(1.0 \times 10^{-6}\) and in energy to \(1.0 \times 10^{-9}\). Results for the CHT predicted block surface temperature, flow passage temperature and Mach number are displayed in Figs. 4-5. Residuals The Mach number in center of inlet channel is approximately 0.06, and typical Mach number in outlet channel is approximately 0.068 (variation over the height of the channel due to the flow over the block).
5 Conclusions

A combined FVM/BEM method has been developed to solve the conjugate problem in CHT analysis. As a boundary only grid is used by the BEM, the computational time for the heat conduction analysis is insignificant compared to the time used for the NS analysis. The proposed method produces realistic results without using arbitrary assumptions for the thermal condition at the conductor surface.

Acknowledgement

This research was carried out under funding from an NRA grant from NASA Glenn, NAG3-2311.

6 References


Figure 1: Cross-section of experiment set-up which is also used in numerical simulation. Upstream channel extends 10 hydraulic diameters upstream of the block.

Figure 2: Finite Volume surface mesh. Cell centered finite volumes, with four cells in z-direction and total of 1104 surface cells.
Figure 3: Surface BEM discretization for the block with 946 equally spaced bilinear elements distributed over the surface of the block.

Figure 4: Surface temperature distribution from the converged conjugate solution. Temperature distribution plotted from the BEM solution.
Figure 5: CHT predicted (a) flow passage temperatures and Mach numbers, (b) top corner Mach number, and (c) top corner temperatures.