

CONJUGATE HEAT TRANSFER: COMPUTATIONAL METHODS AND APPLICATIONS

Alain Kassab, alain.kassab@ucf.edu

Mechanical and Aerospace Engineering, University of Central Florida, Orlando, Florida 32816-2450

Eduardo Divo, divoe@erau.edu

Mechanical Engineering Department, Embry-Riddle Aeronautical University, DB Campus, LB-159, 386-226-7987

Abstract. The problem of conjugate heat transfer (CHT) refers to the coupling of convective heat transfer external to the solid body of a thermal component to conduction heat transfer within the solid body. Conjugate heat transfer is of importance in thermal analysis in which multi-mode convective/conduction heat transfer is critical to design of thermal systems. CHT analysis is typically carried out via: (1) hybrid loose-coupling procedure method or (2) the strongly-coupled homogeneous/monolithic method. We present results from a FVM/BEM hybrid approach and from a FVM monolithic approach and show the importance of CHT.

Keywords: Conjugate heat transfer, boundary elements, computational fluid dynamics.

1. INTRODUCTION

Engineering analysis of complex power producing mechanical devices requires ever-increasing fidelity in numerical models upon which designers rely in their efforts to attain demanding specifications placed on efficiency and durability of modern machinery. In order to capture more of the physics under investigation a coupled-field analysis is often incorporated in computational models for analysis and design. For gas turbine applications, as in many other heat transfer applications, it then becomes necessary to accompany the computation of the flow and associated heat transfer in the fluid with the heat conduction inside the adjacent solid surfaces, for example in the case of blade film cooling. The coupling of these two modes of heat transfer is termed as conjugate heat transfer (CHT). In many cases, the CHT problem itself includes coupling of several types of heat transfer problems. For example, in a typical cooled turbine airfoil/shroud, there are three heat transfer problems that are linked together: external convection, internal convection, and conduction within the metal.

The problem of conjugate heat transfer (CHT) thus refers to the coupling of convective heat transfer external to the solid body of a thermal component to conduction heat transfer within the solid body of that component. Conjugate heat transfer is of importance in thermal analysis in which multi-mode convective/conduction heat transfer is critical to design. CHT examples of external and internal coupled temperature fields include, automotive engine blocks, fuel ejectors, cooled turbine blade, vanes, and end-walls, thermal protection system (TPS) for re-entry vehicles, and cooled rockets nozzles. The continuity of temperature and heat flux at the solid/fluid interface must be satisfied:

$$T_s = T_f \quad \text{and} \quad -k_s \frac{\partial T_s}{\partial n} = k_f \frac{\partial T_f}{\partial n} \quad (1)$$

where the subscripts s and f denote the solid and fluid respectively and $\partial/\partial n$ denotes the normal derivative.

Generally, the approaches to solving a conjugate heat transfer problem can be divided up into two methods: (1) hybrid loose-coupling procedure method and (2) the strongly-coupled homogeneous/monolithic method. The hybrid method is performed using CFD solvers for internal or external flows coupled to a conventional FVM, FEM or BEM solver to predict the temperature distribution in the solid. The main disadvantages of this method are problems associated with handling interface boundary conditions between different calculation areas. Whereas, the homogeneous method consists of direct coupling of the fluid zone and the solid zone using the same discretization and numerical approach. This makes it possible to have an interpolation-free crossing of the heat fluxes between the neighboring cell faces. Additionally, the wall surface temperature as well as the temperatures in the solid are a direct results of this simulation. However, this method requires access to the source codes of both solid conduction and convective flow solvers and issues arise with regards to integration of the algebraic analog for the heat conduction solver to the convective heat transfer solver and the over-refined mesh for conduction if compatible fluid and solid meshes are used.

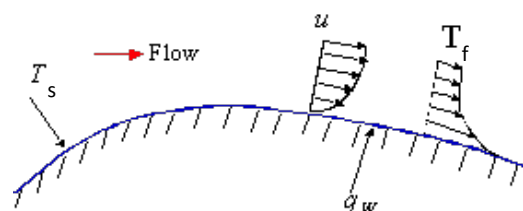


Figure 1. CHT: the coupling of the external and internal thermal fields.

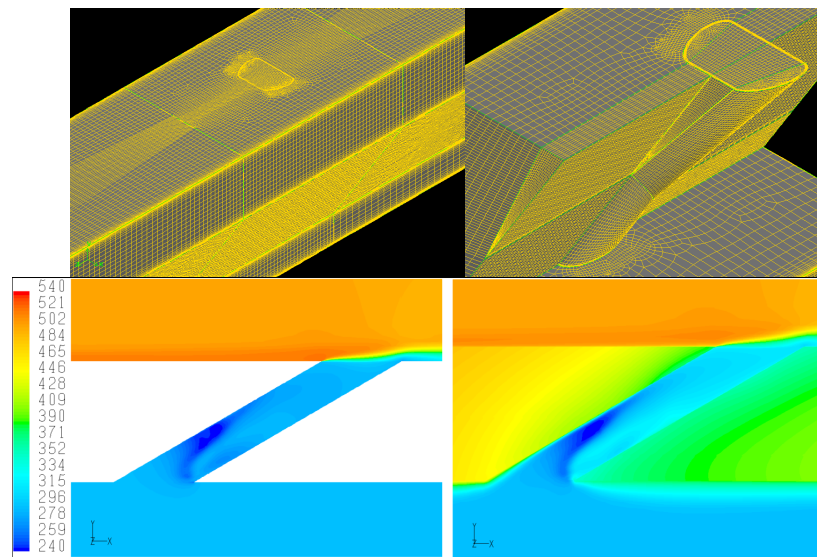
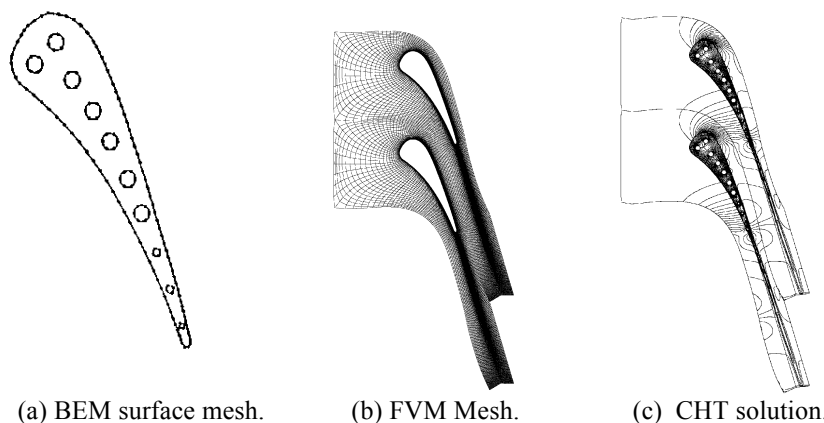


Figure 2. Details of grid used in an FVM monolithic CHT analysis of a film cooling study comparing temperatures along centerline plane for adiabatic and CHT cases (Silieti, Divo and Kassab, 2009a and 2009b).



(a) BEM surface mesh. (b) FVM Mesh. (c) CHT solution.
Figure 3. Loosely coupled BEM/FVM CHT model (a) BEM surface mesh, (b) FVM grid, and (c) CHT solution (Li and Kassab, 1981).

2. CONJUGATE HEAT TRANSFER SOLUTION VIA FVM/BEM LOOSELY COUPLED APPROACH

There are several algorithmic approaches which can be taken to solve the conjugate problem. Most methods are based on either finite elements (FEM), finite volume methods (FVM), or more recently meshless methods (Divo and Kassab; Pepper, Kassab and Divo, 2014) and, in such case, a complete meshing of both fluid and solid regions is required while enforcing solid/fluid interface continuity of fluxes and temperature. An attractive approach that does not require meshing of the solid region to resolve the heat conduction problem is the Boundary Element Method (BEM), and the CHT model is accomplished by coupling the BEM for heat conduction within the solid to the FVM solution of the Navier-Stokes equations in the fluid region while enforcing the continuity of heat flux and temperature at the solid/fluid interface. Interestingly, the nodal temperatures and heat fluxes appear as the boundary unknowns in the BEM.

Moreover, the homogenous/monolithic coupling or the so-called loose coupling strategy may be undertaken. In the former method the fields are solved simultaneously in one large set of equations. Monolithic coupling is mostly applicable for problems where time accuracy is critical, for instance, in aero-elasticity applications where the time scale of the fluid motion is on the same order as the structural modal frequency. However, this approach suffers from a major disadvantage due to the mismatch in the structure of the coefficient matrices arising from BEM, and the FEM and/or FVM solvers. That is, given the fully populated nature of the BEM coefficient matrix, the direct coupling approach would severely degrade numerical efficiency of the solution by directly incorporating the fully populated BEM equations into the sparsely banded FEM or FVM equations. A second approach which may be followed is a loose coupling strategy where each set of field equations is solved separately to produce boundary conditions for the other. The equations are solved in turn until an iterated convergence criterion, namely continuity of temperature and heat flux,

is met at the fluid-solid interface. The loose coupling strategy is particularly attractive when coupling auxiliary field equations to computational fluid dynamics codes as the structure of neither solver interferes in the solution process.

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 (Li and Kassab, 1981; Heidmann, et al., 2003). 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 to update the boundary conditions, as intermediate solutions are not physical in this scheme. In the calculations carried out in such cases, the BEM solution was run once for every decade or so cycles of the finite volume solver. This is referred to as the temperature forward/flux back (TFFB) coupling algorithm as outlined below:

- FVM or FEM Navier-Stokes solver:

1. Begins with initial adiabatic boundary condition at solid surface.
2. Solves 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.
2. Solves steady-state conduction problem.
3. Provides flux distribution to FVM solver.

The transfer of heat flux from the BEM to the FVM solver is accomplished after under-relaxation: $q^{BEM} = q_{old}^{BEM} + \beta(q_{new}^{BEM} - q_{old}^{BEM})$ with β as the relaxation parameter. The process continues until the NS solver, wall temperatures and heat fluxes converge, that is until

$$\|T_s - T_f\| \leq \varepsilon_T \quad \text{and} \quad \|q_s - q_f\| \leq \varepsilon_q \quad (2)$$

is satisfied within a set tolerances. The process is continued until the NS solver converges and wall temperatures and heat fluxes converge. In this steady-state application, the Kirchhoff transform is used to solve the non-linear heat conduction problem with the BEM. The alternative approach of specifying the flux as a boundary condition for the BEM code leading to a flux forward temperature back (FFTb) approach would lead to specifying second kind boundary conditions completely around the surface of a domain governed by an elliptic equation, resulting in a non-unique solution.

An issue arises in information transfer between CFD and BEM as there exists a significant difference in the levels of discretization between the two meshes in a typical CHT simulation. Accurate resolution of the boundary layer requires a FVM/FEM surface grid which is much too fine to be used directly in the BEM. A much coarser surface grid is typically generated for the BEM solution of the conduction problem. The disparity between the two grids requires a general interpolation of the surface temperature and heat flux between the two solvers as it is not possible in general to isolate a single BEM node and identify a set of nearest FVM/FEM nodes. Indeed in certain regions where the CFD mesh is very fine, a BEM node can readily be surrounded by tens or more FVM nodes. A radial basis function interpolation has been successfully used for transfer of temperature and flux values between the BEM and CFD grids in large scale turbomachinery cooled blade CHT computations.

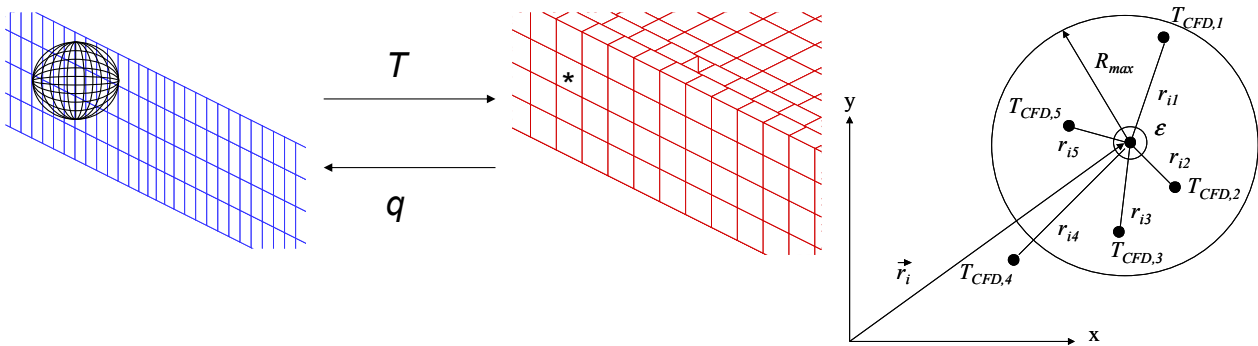
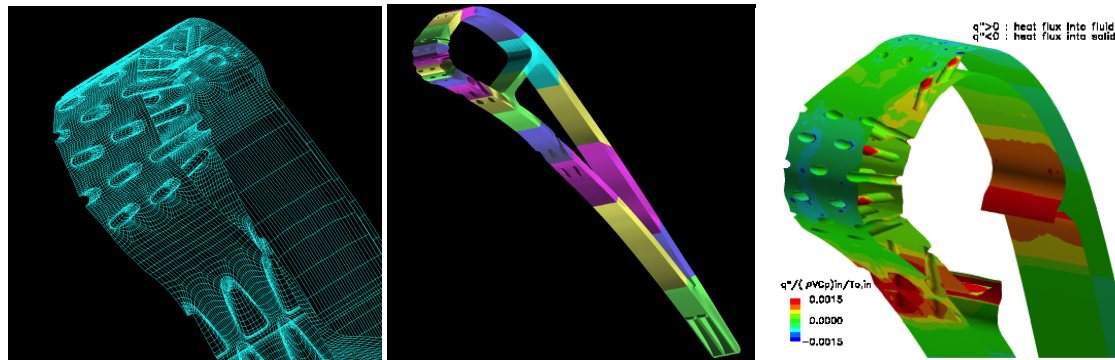


Figure 4. RBF's used to exchange information between disparately discretized solid (BEM) thermal analysis and fluid (FVM or FEM) analysis in a CHT simulation (Kassab et al., 2003).

An example of a CHT analysis utilizing the BEM/FVM coupled TFFB scheme of a film-cooled turbine blade is a study (Heidmann et al., 2003) where the authors coupled the NASA Glenn FVM multi-block turbomachinery code Glenn-HT to a 3D BEM code specially tailored through iterative domain decomposition for large-scale computation (Divo, Kassab, and Rodriguez, 2003). CHT analysis reveals that multidimensional conduction is important in the accurate determination of the thermal field of the blade and that, in general, the conjugate conduction reduces wall temperatures where heat flux is into the wall and increases them where heat flux is into the fluid. The indication is that thermal conduction is not simply one-dimensional across the thin wall of the blade, but, rather that heat also travels along the vane wall toward low temperature regions from the hotter adjacent regions. These results show that it is too simplistic to assume that increasing vane conductivity will monotonically drive all surface temperatures toward the vane mean temperature, especially for a two temperature mixing problem such as turbine film cooling.



(a) Film cooled blade. (b) BEM domain decomposition. (c) Surface heat flux.

Figure 5. CHT analysis of film-cooled blade: (a) surface mesh of film-cooled blade, (b) BEM domain decomposition, and (c) surface heat flux.

CHT solutions can also be obtained in a time accurate manner (Rahaim, Kassab, and Cavalleri, 2000). In such a case, a transient BEM solver must be used and the dual reciprocity BEM (DRBEM) approach is suitable for this purpose. The results comparing the temperature histories from CHT computations at two thermocouple locations inside a heated blunted wedge subjected to cooling in a supersonic wind tunnel are provided in Figure 6. The BEM mesh of an in-house DRBEM transient conduction solver and the FVM mesh utilized for two compressible NS flow solvers (GASP and NASFLO) are provided. These results were validated against experiments.

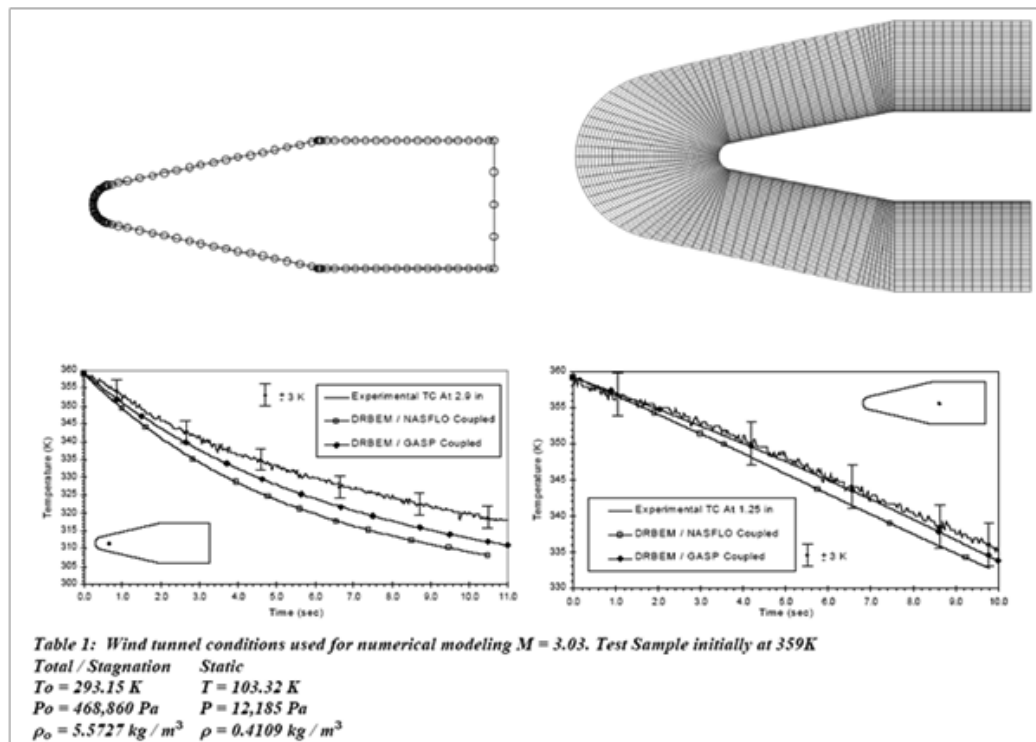
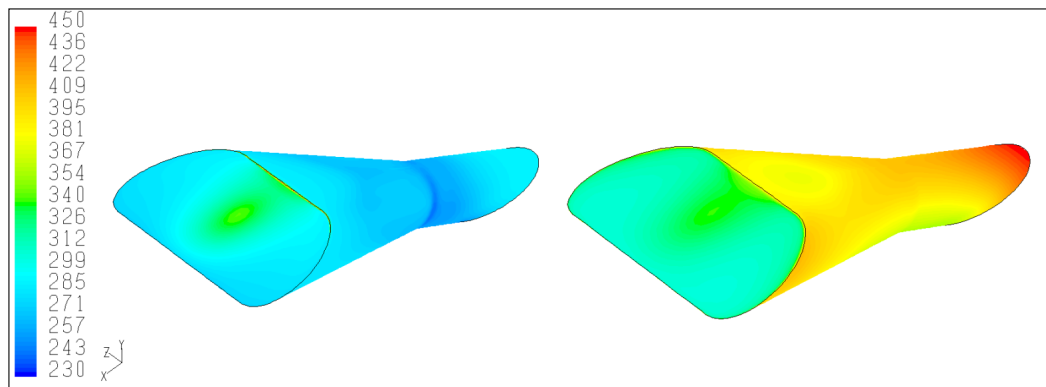


Figure 6. Time-accurate CHT solution based on DRBEM/FVM coupling (Rahaim, Kassab, and Cavalleri, 2000).

3. MONOLITHIC FVM CHT ANALYSIS

An example of the importance of CHT is the film cooling hole study of (Silieti, Kassab, and Divo, 2009a.) where an isolated film-cooling hole is studied in detail. Here, a monolithic FVM CHT analysis is carried out with Ansys Fluent. Referring to Figure 2 in this paper, where the temperature magnitude contours are shown along centerline plane in the film cooling hole region as predicted by the realizable $k-\epsilon$ turbulence model, it is apparent that the film-hole surface temperature predicted by a adiabatic analysis is quite different than that predicted by CHT analysis as seen in Figure 7. The film hole is hotter due to conduction from the hot gas path across the endwall to the cooling hole raising the cooling hole temperature and thereby reducing film effectiveness.



(a) Adiabatic surface film-hole temperature. (b) CHT film hole surface temperature.

Figure 7. Comparison of adiabatic and CHT analysis of a film-cooling hole (Silieti, Kassab, and Divo, 2009a.)

4. CONCLUSIONS

The field problem of conjugate heat transfer (CHT) refers to the coupling of convective heat transfer external to the solid body of a thermal component coupled to conduction heat transfer within the solid body of that component. Conjugate heat transfer is of importance in thermal analysis in which multi-mode convective/conduction heat transfer is critical to determining design parameters of thermal systems. CHT analysis is typically carried out via: (1) hybrid loose-coupling procedure method and (2) the strongly-coupled homogeneous/monolithic method. Results from a FVM/BEM hybrid approach and from a FVM monolithic approach and show the importance of CHT.

5. REFERENCES

- Divo, E.A. and Kassab, A.J., 2007, "An Efficient Localized RBF Meshless Method for Fluid Flow and Conjugate Heat Transfer". ASME Journal of Heat Transfer, Vol.129, pp.124-136.
- Divo, E., Kassab, A., and Rodriguez, F., 2003. "Parallel Domain Decomposition Approach for large-scale 3D Boundary Element Models in Linear and Non-Linear Heat Conduction". Numerical Heat Transfer, Part B, Fundamentals, Vol. 44, No.5, pp. 417- 437.
- Heidmann, J.D., Kassab, A.J., Divo, E.A., Rodriguez, F., and Steinthorsson, E., 2003. "Conjugate Heat Transfer Effects on a Realistic Film-Cooled Turbine Vane". ASME Paper GT2003- 38553.
- Li, H. and Kassab, A.J., 1981, "A Coupled FVM/BEM Solution to Conjugate Heat Transfer in Turbine Blades," AIAA Paper 94-1981.
- Kassab, A., Divo, E., Heidmann, J., Steinthorsson, E., and Rodriguez, F., 2003. "BEM/FVM Conjugate Heat Transfer Analysis of a Three-Dimensional Film Cooled Turbine Blade." International Journal for Numerical Methods in Heat and Fluid Flow, Vol. 13, No. 5, pp. 581-610.
- Pepper, D., Kassab, A., and Divo, E., 2014. An Introduction to the Finite Element, BoundaryElement, and Meshless Methods - with applications to heat transfer and fluid flow. ASME Press. New York.
- Rahaim, C.P., Kassab, A.J., and Cavalleri, R., 2000. "A Coupled Dual Reciprocity Boundary Element/Finite Volume Method for Transient Conjugate Heat Transfer". AIAAJournal of Thermophysics and Heat Transfer, Vol. 14, No. 1, pp. 27-38.
- Silieti, M., Kassab, A.J. and Divo, E.A., 2009a. "Film Cooling Effectiveness: comparison of adiabatic and conjugate heat transfer CFD models". International Journal of Thermal Sciences, Vol. 48, pp. 2237–2248.
- Silieti, M., Kassab, A.J. and Divo, E.A., 2009b. "The Effect of Conjugate Heat Transfer on Film Cooling Effectiveness," Numerical Heat Transfer, Part A: Fundamentals, Vol. 56, No. 5, pp. 335-350.

6. RESPONSIBILITY NOTICE

The authors are solely responsible for the printed material included in this paper.