International Journal of Heat and Mass Transfer 85 (2015) 181–194

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Coupled inviscid-viscous solution method for bounded domains: Application to data-center thermal management

Ethan Cruz, Yogendra Joshi *

Systems and Technology Group, IBM, 11400 Burnet Rd., Austin, TX 78758-3493, USA
School of Mechanical Engineering, Georgia Institute of Technology, 771 Ferst Dr., Atlanta, GA 30332, USA

ARTICLE INFO

Article history:
Received 27 October 2014
Received in revised form 11 January 2015
Accepted 12 January 2015

Keywords:
Inviscid
Viscous
Computational fluid dynamics (CFD)
Data-center

ABSTRACT

Computational fluid dynamics and heat transfer (CFD/HT) models have been employed as the dominant technique for the design and optimization of both new and existing data-centers. Inviscid modeling has shown great speed advantages over the full Navier–Stokes CFD/HT models (over 20 times faster), but is incapable of capturing the physics in the viscous regions of the domain. A coupled inviscid-viscous solution method (CIVSM) for bounded domains has been developed in order to increase both the solution speed and accuracy of CFD/HT models. The methodology consists of an iterative solution technique that divides the full domain into multiple regions consisting of at least one set of viscous, inviscid, and interface regions. The full steady, Reynolds averaged Navier–Stokes equations with turbulence modeling are used to solve the viscous domain, while the inviscid domain is solved using the Euler equations. By combining the increased speed of the inviscid solver in the inviscid regions, along with the viscous solver's ability to capture the turbulent flow physics in the viscous regions, a faster and potentially more accurate solution can be obtained for bounded domains that contain large inviscid regions, such as data-centers.

1. Introduction

The pervasive trend of increasing heat flux and power dissipation of Information Technology (IT) equipment [1], has created a significant challenge for air cooled data-center facilities, which can contain as many as several thousand pieces of IT equipment. In order to maintain high reliability, this equipment must be supplied adequate cooling air according to the manufacturers' specifications. As the power dissipation increases, so do the cooling requirements, necessitating higher IT equipment and cooling supply air flow rates. This has led to complex flow patterns and temperature distributions within these data-centers. In order to better understand these and to reduce mixing of hot and cold air streams, which degrades cooling efficiency, computational fluid dynamics and heat transfer (CFD/HT) modeling has been employed as the dominant technique for the design and optimization of both new and existing data-centers. With as much as 1.5% of the world's and 2.2% of the United States' electrical power going to data-centers, and roughly half of that used for cooling [2], optimizing the cooling systems of data-centers for minimized power consumption has become an industry imperative.

1.1. Reynolds-averaged Navier–Stokes

A number of researchers [3–14] have used the standard k–ε turbulence model to simulate data-centers using different CFD/HT Reynolds-Averaged Navier–Stokes (RANS) solvers. It has become the most commonly used turbulence model in data-center analysis [15], although it has neither proven to be the most accurate [11–13,15] nor the most computationally efficient [11–13]. Patel et al. [3] found an error ranging from 7 to 12% in temperature predictions for an overhead return, raised floor data-center test lab. Shrivastava et al. [6] compared a CFD/HT model to an actual 690 m^2 (7400 ft^2) data-center housing over 130 IT equipment racks and found a mean absolute rack inlet air temperature error of 4 °C, with a standard deviation of 3.3 °C. Schmidt et al. [7] also compared a CFD/HT model to measured temperature data from a data-center. While the general trends in the IT equipment inlet air temperatures were reasonably predicted, there were deviations of more than 30 °C between the measured and predicted values. Amemiya et al. [8] using a three-dimensional temperature mapping tool [16] found that transients within the data-center affected the temperature mapping results due to the differing time scales. Temperature

* Corresponding author at: School of Mechanical Engineering, Georgia Institute of Technology, 771 Ferst Dr., Atlanta, GA 30332, USA.

E-mail addresses: ethanc@gatech.edu (E. Cruz), yogendra.joshi@me.gatech.edu (Y. Joshi).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.01.077
0017-9310/

spikes of over 2 °C were discovered at the both the IT equipment and the chiller units, raising the uncertainty of the temperature measurements.

1.2. Reduced order modeling

In order to increase computational efficiency, reduced order modeling techniques have been used such as ad hoc methods [17–20], Proper Orthogonal Decomposition (POD) [21], and various inviscid and potential flow methods [13,14,22,23]. In general, these methods increase the computational efficiency relative to traditional CFD/HT RANS modeling methods, but at the cost of modeling accuracy. Combinations of these methods are also being studied in order to increase modeling accuracy [24–28].

The reduced order models based on ad hoc methods rely on measurements in order to predict air temperatures at specific locations within a data-center. These methods require the compilation of large amounts of experimental data in order to create a computational model. While able to help predict how the specific data-center will react to changes in certain parameters, these methods are not able to provide timely insight into the construction of new, or rearrangement of existing data-centers [17–20].

POD based algorithms have proven quite effective in reducing the computational effort, along with providing reasonably accurate results at the rack scale, but have not been able to provide accurate results at the room scale [21]. In combination with ad hoc methods, a POD model has shown both increased computational speed and accuracy for a specific data-center layout [24]. Although future improvements to this method could reduce or eliminate these limitations, the combined method is currently incapable of predicting the effects of rearranging existing data-centers, or in optimizing the construction of new ones without new observations.

1.2.1. Inviscid methods

Inviscid and potential flow reduced order modeling techniques are the most similar to the full Navier–Stokes CFD/HT models and in some cases are simply a subset of the same equations. The inviscid equations (or Euler equations) are a subset of the full Navier–Stokes equations with the elimination of the viscous terms. The potential flow models add the requirement that the flow is not only inviscid but also irrotational. Both of these simplified modeling techniques allow for significantly smaller grid counts and therefore faster solution times, but do not necessarily capture all of the physics in a complex, turbulent flow field. For large rooms with high Reynolds number flows such as those found in many data-centers, it may be worth the trade-off of loss of accuracy for the guaranteed reduction in solution time.

Toulouse et al. [22] explored the use of a finite difference solver based on the potential flow equations and found significant reductions in solution time. An experimental validation was performed on a modified version of this solver [23], which showed large deviations between predicted and measured temperatures. A later study [14] used the method of vortex superposition to modify the original model to account for the effects of buoyancy which were thought to be the cause of the previous inaccuracies. This new model was then "tuned" using temperature data collected in the modeled data-center. Three models were used to compare measured temperatures in a data-center: the new "optimized" model with superimposed Rankine vortices, the original "basic" potential flow model, and a CFD/HT model using the k–ε turbulence model. The CFD/HT model produced the lowest overall temperature deviations, while the "optimized" model produced the lowest rack inlet temperature deviations. The low deviations at the inlet were not unexpected for the "optimized" model since that was the location the model was "tuned" to produce the best results. While providing a significant reduction in computational effort, the "optimized" model may need to be "tuned" for different geometries of data-centers in order to be able to limit the deviations.

1.2.2. Combination of multiple methods

There are a few other examples of combining potential flow modeling with other techniques. Most notable are the combinations of potential flow and ad hoc methods [25–28] for the modeling of data-centers. Hamann et al. [26] explored the zones that each CRAC creates within the data-center using sensor data, along with a potential flow model. Das et al. [27] expanded on this potential flow modeling, incorporating sensor data to predict and optimize CRAC thermal zones in order to minimize power consumption. Lopez and Hamann [28] considered the number of ad hoc data points needed for an accurate potential flow model.

In order to evaluate the performance of helicopter rotors, [29,30] developed a closely coupled potential flow and full Navier–Stokes modeling technique. When compared to a full Navier–Stokes model, this technique reduced the computational effort by 40–50% without reducing the accuracy. However, it relies heavily on the a priori knowledge of the shape and approximate location of the inviscid-viscous boundary for the flow around the rotor. Two regions are formed; a viscous region near the rotor that employs the full Navier–Stokes solver, and an inviscid region a distance away from the rotor that employs the potential flow solver. The two regions are solved iteratively, and are coupled via an interface "surface", which provides the boundary conditions for the two regions, and allows them to interact with one another.

To the best of the authors' knowledge, a coupled inviscid-viscous CFD/HT technique has not been executed on low-speed, internal (bounded) flow applications such as data-centers.

2. Coupled inviscid-viscous solution method (CIVSM)

We developed a coupled inviscid-viscous solution method (CIVSM) for subsonic flow in a bounded domain based on [31]. First, a high-level description of the method is presented. This is followed by a delineation of the governing equations for the different solution domains, along with guidelines on properly sizing the grid. The specific algorithms and procedures referred to in the high-level description are then more completely illustrated. These include a detailed explanation of the partitioning algorithm, boundary conditions and coupling procedure, mass and energy balance algorithms, various convergence criteria, and pressure–velocity coupling methods.

Once the CIVSM is fully described, results from a test case based on a data-center test cell are compared to traditional CFD/HT with multiple turbulence models. Fig. 1 shows a picture of most of the region of interest within the data-center test cell, as well as a plan-view and an isometric of the data-center test cell CFD/HT model. Further details on the values for the different parameters are enumerated and discussed. The results include a discussion of the partitioning algorithm, a grid study, the mass balance algorithm, modeling accuracy, and solution time.

2.1. Overall solution method

Fig. 2 shows a flow chart of the general solution approach. After a CFD/HT model is created with all of the appropriate boundary conditions, the model is solved using the inviscid equations on a coarse mesh. The second step in the process is to re-run the model using the solution to the initial solve (the first step) as the initial solution with a more refined "medium" mesh and using a basic turbulence model, such as the zero-equation mixing-length model [32,33].