16

Convective Thermal Flow Problems

Christian Moosmann and Andreas Greiner

Institute for Microsystem Technology, Albert Ludwig University
Georges Köhler Allee 103, 79110 Freiburg, Germany
{moosmann,greiner}@imtek.uni-freiburg.de

Summary. A benchmark for the convective heat transfer problem, related to modeling of a anemometer and a chip cooled by forced convection, is presented. It can be used to apply model reduction algorithms to a linear first-order problem.

16.1 Modeling

Many thermal problems require simulation of heat exchange between a solid body and a fluid flow. The most elaborate approach to this problem is computational fluid dynamics (CFD). However, CFD is computationally expensive. A popular solution is to exclude the flow completely from the computational domain and to use convection boundary conditions for the solid model. However, caution has to be taken to select the film coefficient.

An intermediate level is to include a flow region with a given velocity profile, that adds convective transport to the model. The partial differential equation for the temperature T in this case reads:

$$\rho c \left(\frac{\partial T}{\partial t} + \mathbf{v} \nabla T \right) + \nabla \cdot (-\kappa \nabla T) = \dot{q} \tag{16.1}$$

where ρ is the mass density, c is the specific heat of the fluid, $\mathbf{v}$ is the fluid speed, κ is the thermal conductivity, $\dot{q}$ is the heat generation rate.

Compared to convection boundary conditions this approach has the advantage that the film coefficient does not need to be specified and that information about the heat profile in the flow can be obtained. A drawback of the method is the greatly increased number of elements needed to perform a physically valid simulation, because the solution accuracy when employing upwind finite element schemes depends on the element size. While this problem still is linear, due to the forced convection, the conductivity matrix changes from a symmetric matrix to an un-symmetric one. So this problem type can be used as a benchmark for problems containing un-symmetric matrices.

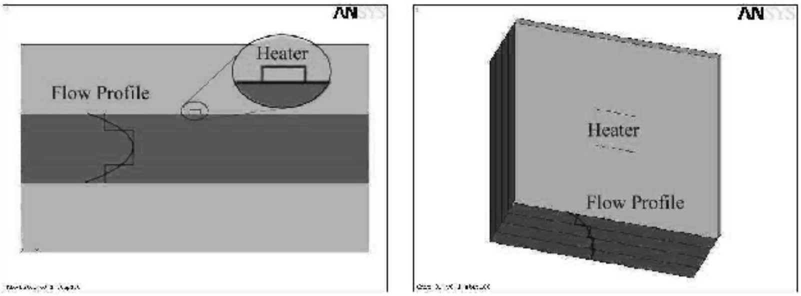

Fig. 16.1. Convective heat flow examples: 2D anemometer model (left), 3D cooling structure (right)

16.2 Discretization

Two different designs are tested: a 2D model of an anemometer-like structure mainly consisting of a tube and a small heat source (Figure 16.1 left) [Ern01]. The solid model has been generated and meshed in ANSYS. Triangular PLANE55 elements have been used for meshing and discretizing by the finite element method, resulting in 19 282 elements and 9710 nodes. The second design is a 3D model of a chip cooled by forced convection (Figure 16.1 right) [Har97]. In this case the tetrahedral element type SOLID70 was used, resulting in 107 989 elements and 20542 nodes. Since the implementation of the convective term in ANSYS does not allow for definition of the fluid speed on a per element, but on a per region basis, the flow profile has to be approximated by piece-wise step functions. The approximation used for this benchmarks is shown in figure 16.1.

The Dirichlet boundary conditions are applied to the original system. In both models the reference temperature is set to 300 K, Dirichlet boundary conditions as well as initial conditions are set to 0 with respect to the reference. The specified Dirichlet boundary conditions are in both cases the inlet of the fluid and the outer faces of the solids. Matrices are supplied for the symmetric case (fluid speed is zero; no convection), and the unsymmetric case (with forced convection). Table 16.1 shows the output nodes specified for the two benchmarks, table 16.2 shows the filenames according to the different cases.

Further information on the models can be found in [MRGK04] where model reduction by means of the Arnoldi algorithm is also presented.

16.3 Acknowledgments

This work is partially funded by the DFG project MST-Compact (KO-1883/6), the Italian research council CNR together with the Italian province of Trento PAT, by the German Ministry of Research BMBF (SIMOD), and an operating grant of the University of Freiburg.

Table 16.1. Output nodes for the two models

Model	Number	Code	Comment
Flow Meter	1	out1	outlet position
	2	out2	outlet position
	3	SenL	left sensor position
	4	Heater	within the heater
	5	SenR	right sensor position
cooling Structure	1	out1	outlet position
	2	out2	outlet position
	3	out3	outlet position
	4	out4	outlet position
	5	Heater	within the heater

Table 16.2. Provided files

Model	fluid speed (m/s)	Filenames
Flow Meter	0	flow_meter_model_v0.*
	0.5	flow_meter_model_v0.5.*
cooling Structure	0	chip_cooling_model_v0.*
	0.1	chip_cooling_model_v0.1.*

References

[Ern01] Ernst, H.: High-Resolution Thermal Measurements in Fluids. PhD thesis, University of Freiburg, Germany (2001)

[Har97] Harper, C. A.: Electronic packaging and interconnection handbook. New York McGraw-Hill, USA (1997)

[MRGK04] Moosmann, C., Rudnyi, E.B., Greiner, A., Korvink, J.G.: Model Order Reduction for Linear Convective Thermal Flow. In: Proceedings of 10th International Workshops on THERMal INvestigations of ICs and Systems, THERMINIC2004, 29 Sept - 1 Oct , Sophia Antipolis, France, p. 317-322 (2004)