Model Order Reduction for Linear Convective Thermal Flow

Christian Moosmann, Evgenii B. Rudnyi, Andreas Greiner, Jan G. Korvink

IMTEK, April 2004

Abstract

Simulation of the heat exchange between a solid body and a neighboring fluid is a common task in IC and MEMS design. In this paper, we demonstrate that model order reduction can be successfully applied for thermal flow problems with a specified flow velocity. We present the approach with an example of two devices: an anonemeter and IC board. For both cases, good agreement between the results of the full model and the reduced model has been observed. The total simulation time, that is, the model reduction time plus the simulation time of the reduced model is comparable with the time needed for the stationary solution. As a result, the inclusion of the flow with given velocity profile to the computational domain does not increase the computational time significantly and can be used efficiently when it is difficult to use convection boundary conditions.

1 Introduction

Many thermal problems require the description of heat exchange between a solid body and a flowing fluid. The most elaborate approach to this kind of 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 when setting these boundary conditions, especially in selecting the film coefficient.

An intermediate level is to include a flowregion with a given velocity profile, that adds convective transport to the model. Compared to convection boundary conditions this approach has the advantage that the film coefficient has not to be specified. 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. A formulation for this dependency is the grid Peclet number (1.1).

$$Pe = \frac{v \cdot L \cdot c}{2\kappa} \tag{1.1}$$

where v is the fluid speed, L is the element length in direction of the fluid flow c is the specific heat of the fluid and κ is the thermal conductivity of the fluid

To ensure correct simulation results the grid Peclet number must be lower than 1.0 [1, 2]. Therefore, with increasing fluid speed, the element size has to be decreased, accordingly. This results in a larger number of elements and therefore in longer computations time.

However, the given velocity profile leads to a linear system of differential equations. Then, by means of model order reduction, a model can be generated that describes the system with sufficient accuracy, but that is of much smaller order than the original one.

Two examples are used to demonstrate the advantages of model reduction. The first one is a 2D model of an anemometer, a flow meter based on convective thermal flow [3] (see Sensirion AG[1] for comercially available anemometers). The second model is a simple 3D model of a heat source, e.g. a chip, acting as a constant heat source, that is cooled by air flow, a typical problem in the heat management in IC industries [4].

2 Considered problem type

The considered models are generated with ANSYS®[2] using the PLANE55 and the SOLID70 element types, since these allow for computing convective heat flow. However, the velocity can only set constant in a defined region, so it is necessary to approximate a flow profile by step functions. While it would be possible to generate a large number of regions, to approximate the flow profile very well, the regions should not become too thin, as this would generate problems with meshing the region. For the rather small problems presented here only a coarse approximation has been used. This is depicted in figure 2.1.

Figure 2.1 a) shows the ANSYS model of an anemometer. The heater is included in the upper wall. Approximation of the flow profile is done by three zones. The model consists of approximately 19 282 elements and 9710 nodes, i.e. degrees of freedom (DOF).

In figure 2.1 b) a simple 3D structure is depicted. The heat source, e.g. a chip, is located in the middle of a baseplate, over which air flows with constant velocity. Only half of the flow field is computed here, it is approximated by four steps. Here 107 989 elements were used, resulting in 20542 DOF.

[1]Sensirion AG, Switzerland, www.sensirion.com
[2]ANSYS®, Ansys, Inc., www.ansys.com

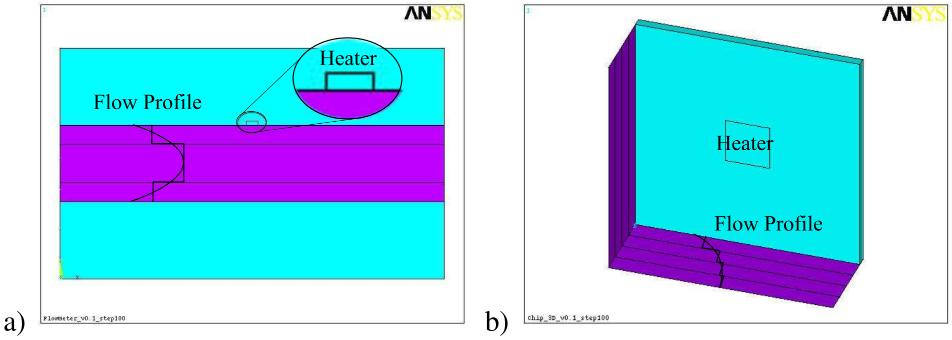


Figure 2.1: Computed models, a) 2D anemometer model, b) 3D chip cooling model

2.1 Mathematical formulation of the problem

Energy is a conserved quantity following a balance equation. Everywhere in the domain this means, that the heat generation rate in a volume ($\dot{q}$) has to equal the sum of the change of the total heat ($\rho c \cdot \partial T/\partial t$), the conductive heat flow ($-\kappa \nabla T$) and the convective heat flow ($\rho c \cdot \vec{v} \nabla T$) across its boundary. The partial differential equation for the temperature T reads:

$$\rho c \left(\frac{\partial T}{\partial t} + \vec{v} \nabla T \right) + \nabla \cdot (-\kappa \nabla T) = \dot{q} \tag{2.1}$$

where ρ denotes the mass density, c is the specific heat of the fluid, $\vec{v}$ is the fluid speed and κ is the thermal conductivity

For the spazially discretized form one obtains a system of ordinary differential equations:

$$C \cdot \dot{T} + K \cdot T = b \tag{2.2}$$

where the conductivity matrix K changes from a symmetric matrix for the case of mere conductive heat flow to an unsymmetric matrix due to the additional convection term. C is the heat capacitance.

When computing large models (i.e. models with many degrees of freedom) or performing parameter variations usually not all nodes of the model are of interest. Actually, in most cases, information from only a few nodes is needed. These are referred to as 'output nodes' in the following. An output matrix E can be defined to link the output nodes y to the state vector T:

$$y = E \cdot T \tag{2.3}$$

3 Introduction to Model Order Reduction with mor4ansys

Model reduction has been already applied succesfully to thermal problems without convection flow [5, 6, 7]. It has also been used for ground water flow problems [8, 9, 10], where the partial differential equation is quite similar to (2.1), except that instead of the temperature the concentration is the variable.

Consider a model, i.e., a system of ordinary differential equations of order n. The number of output nodes (k) is much smaller than the total number of nodes (n). There exists a reduced model of order m ($k \leq m \leq n$) that solves for the output nodes with sufficient accuracy. The task is to find this reduced model. The connection between the original state vector (T) and the reduced state vector (z) is given by the reduction matrix (V), resulting in the reduced form of the discretized equation:

$$T = Vz \tag{3.1}$$

$$C_r \cdot \dot{z} + K_r \cdot z = b_r \tag{3.2}$$

Since the focus of this article is the automated reduction with the *mor4ansys*[3] program, for the theory of generating reduced models the reader also is referred to [11].

3.1 The mor4ansys program

Mor4ansys is software developed at IMTEK[4]. It requires as input a model's system matrix, which it derives from an ANSYS®binary file. Also, information on the output nodes and boundary conditions is needed. With this information, the program performs the Arnoldi algorithm to obtain a reduced model of specified order. For a particular model, all data needed to perform the model order reduction can be directly obtained from the ANSYS®files.

The simulation with the reduced model is performed with common linear algebra software such as Mathematica®[5] or Matlab[6].

4 Results

In figure 4.1, plots of the temperature field are shown for the last timestep. One can clearly see the impact of the flow on the temperature. We would like to stess once more, that the description of the same problem with a convection boundary condition would be quite difficult, because information on the temperature profile in the flow cannot be obtained.

4.1 Computational Time

Performing transient simulations of large models is a costly task. The simple examples presented above needed 3m1s in the case of the 2D model and 26m16s in the case of the 3D model to compute only 100 steps on a dual Intel®Xeon™3.06GHz PC. But these examples are small

[3] http://www.imtek.uni-freiburg.de/simulation/mor4ansys

[4] http://www.imtek.uni-freiburg.de

[5] Mathematica®, Wolfram Research, www.wolfram.com

[6] Matlab, The MathWorks, www.mathworks.com

a) b)

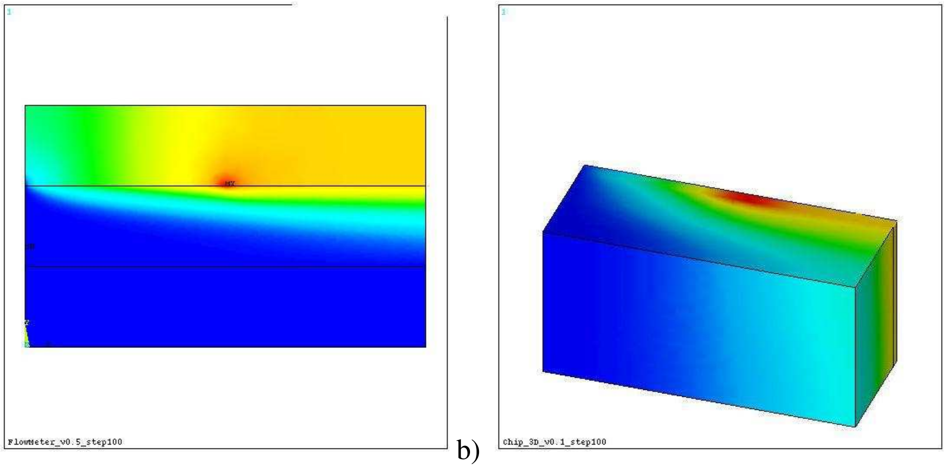


Figure 4.1: Temperature profiles of the computed models, a) 2D anemometer model, b) Cut through the 3D chip cooling model

compared to models for productive simulations, and only few timesteps were computed. So computational times can be expected to be much higher for a realistic simulation task.

Table 4.1 shows a comparison of the computational time needed to compute the model reduction and the steady state ANSYS solution. It shows that the reduction step takes about 2-3 times longer than a steady state solution on the same mesh.

Geometry	Architecture	Reduction step in *mor4ansys*	ANSYS steady state	Factor
Flow Meter, 2D, 9710 DOF	Sun UltraSPARC-II"450 MHz CPU	3.3s	1.8s	1.83
Chip Cooling, 3D, 20542 DOF	Sun UltraSPARC-II"450 MHz CPU	56.8s	19.9s	2.85

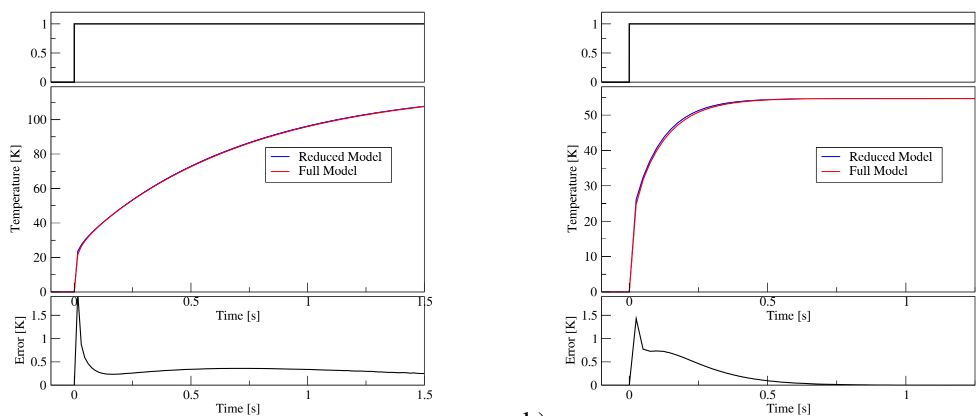
Table 4.1: Comparison between computational time needed for the reduction step and the steady state solution in ANSYS

However, computing 100 time-steps of a reduced model of order 30 (which is used for both examples here) typically needs less than 1s. So with model order reduction the only costly process is the reduction step, and it still is very cheap compared to a transient simulation of the full model.

4.2 Computational Results

The most important question of course is, whether the results computed with the reduced model fit well enough to the results of the full model. For the transient simulation a step response is computed, where the heat source is switched on, while the flow remains constant.

Figures 4.2 show the temperature progression at the heater, as well as the input function and the difference between the simulation with the full model and the reduced model. As expected, the temperature rises very steeply at the beginning to saturate after some time. Comparing the simulation of the full model with the reduced model, small deviations can be seen. However, while the overall error remains very small, a peak can be seen in the error at the first time-step, where the response curve is steepest.



a) b)

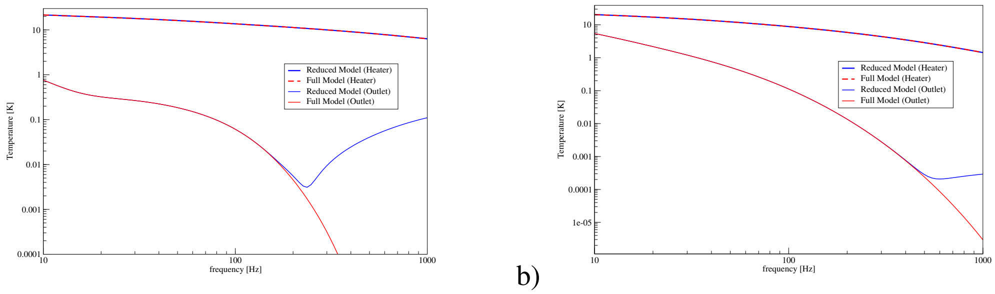
Figure 4.2: Step response, a) 2D anemometer, b) 3D chip cooling

Choosing a step response has one major advantage; since a step consists of the whole harmonic spectrum the error introduced for a specific frequency will be notified here also. Or, stated differently, all other input functions tend to show a smaller error than that of the step response.

Since the process of model order reduction basically is a moment matching approach, i.e., an approximation of the transfer function the most appropriate way to rate the quality of the reduced model is to compare the frequency sweep of the reduced and the complete model (figure 4.2).

While very good agreement can be seen at the heater positions, the results at the outlets show a clear deviation above 200Hz (anemometer) i.e. 400Hz (chip cooling). This fits very well with the results from the transient analysis, where the biggest disagreement could be found for rapid changes (i.e., high frequencies).

However, if very fast (i.e., high frequency) effects have to be taken into account, this behaviour can be further improved with multi-point matching [5, 11].



a) b)

Figure 4.3: Harmonic analysis a) 2D anemometer b) 3D chip cooling

5 Conclusion

The presented paper describes model order reduction for convective thermal flow, as it can be found in cooling systems and anemometers. Very good results could be obtained for the transient and harmonic analysis at the heater position, the results at the output positions also show very good agreement up to high frequencies. Sufficient accuracy in the transient results could be obtained with resources usually needed for the stationary solution.

Therefore, we could show that the approach of model order reduction is an effective tool to speed up transient and harmonic simulations in the case of convective thermal flow. Due to the subsequent reduction step it becomes affordable to increase the dimension of a model by including a stationary flow field with a given profile. The reduced models can then be used to perform parameter variations of the input functions very cost-effectively.

Unfortunately the heat generation rate is the only available input function for the reduced model in this case. Hence, changing the velocity, geometrical or material data makes it necessary to repeat the discretization and thus the reduction step. However, in principle it is possible to define the velocity on a per element basis, so this depends on the used discretization tool.

Figure 5.1 shows the typical workflow for such a variation as well as the states where input (red) is applied or output (blue) is obtained. The approximated flow profile (by steps, in our case) is applied in the design process, to be included in the discretization with the othe model data, like geometry and material properties. The resulting system of differential is reduced and computed with defined input function ($U(t)$, heat generation rate at the heater). Results or output of this simulation is the heat at the specified output nodes. In the presented examples especially the heat at the source, the sensors and at the outlet are of interest. With the same reduced model parameter variations on the input function ($U(t)$) can be performed. The results of this tests are used to perform design changes. As these changes make a new discretization necessary, the reduction step has to be repeated also. With the new reduced model the parameter variation is performed again and so on.

However, the cost of the reduction step is comparable to the cost of a stationary solution, that usually is affordable nowadays, so that a larger number of designs can be evaluated with

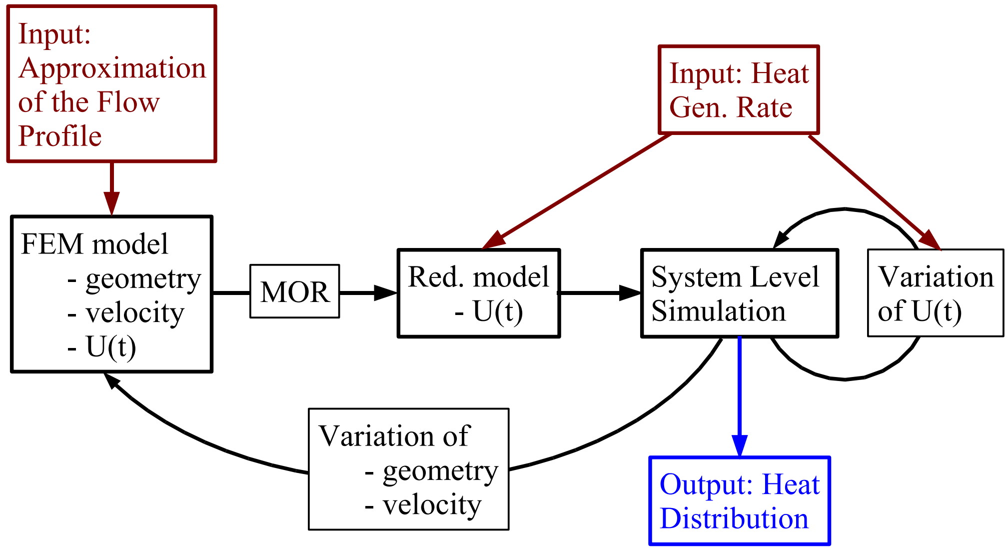


Figure 5.1: Process flow

reasonable effort.

6 Acknolegements

The authors wish to thank the German ministry for science and education (BMBF) for funding in the framework of the project SIMOD.

References

[1] C.A.J. Fletcher. *Computational Techniques for Fluid Dynamics*. Springer Verlag, 1991.

[2] Ansys Incorporated. *Theory Reference*. Ansys 8.0.

[3] Herbert Ernst. *High-Resolution Thermal Measurements in Fluids*. PhD thesis, Albert-Ludwigs-Universität Freiburg, 2001.

[4] C. A. Harper. *Electronic packaging and interconnection handbook*. New York McGraw-Hill, 1997.

[5] D. D'Amore L. Codecasa and P. Maffezzoni. An Arnoldi based thermal network reduction method for electro- thermal analysis. *IEEE Transactions on Components and Packaging Technologies*, 26:186–192, 2003.

[6] E. B. Rudnyi T. Bechtold and J. G. Korvink. Automatic generation of compact electro-thermal models for semiconductor devices. *IEICE Transactions on Electronics*, E86C:459–465, 2003.

[7] Yao-Joe Yang and Che-Chia Yu. Extraction of heat-transfer macromodels for MEMS devices. *Micromechanical Microengineering*, 14:587–596, 2004.

[8] A. Woodbury and K. N. Zhang. Lanczos method for the solution of groundwater flow in discretely fractured porous media. *Advances in Water Resources*, 24:621–630, 2001.

[9] A. Woodbury H. N. Li and P. Aitchison. Application of the unsymmetric Lanczos method to radionuclide decay-chain transport in porous media. *International Journal For Numerical Methods in Engineering*, 44:355–372, 1999.

[10] G. Pini and G. Gambolati. Arnoldi and Crank-Nicolson methods for integration in time of the transport equation. *International Journal For Numerical Methods in Fluids*, 35:25–38, 2001.

[11] Jan G. Korvink Evgenii B. Rudnyi. Review: Automatic model reduction for transient simulation of MEMS-based devices. *Sensors Update*, 2002.