Int. J. Engng Ed. Vol. 25, No. 6, pp. 1105–1113, 2009
Printed in Great Britain.
0949-149X/91 $3.00+0.00

Design Optimization of the Double Glass Problem: Application and Limitations of CFD*

R. J. M. BASTIAANS
Department of Mechanical Engineering, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands. E-mail: r.j.m.bastiaans@tue.nl

A simple teaching example of optimization in fluid mechanics is given, where use is made of Computational Fluid Dynamics (CFD). This was carried out on the basis of the assumption that students are trained at a basic level in CFD. In the future, automatic optimization procedures will become very important in design problems. However, it must be emphasized that optimization in fluid mechanics is mostly not trivial. With automatic optimization, the problem formulation must be kept within its applicability range. Moreover, with geometric parameter variation, the (automatic) grid generation and refinement must be appropriate to obtain accurate results. Furthermore, the accuracy of the applied models in the equations must be in general better than the gain in optimization function. In order to illustrate these issues to students, an exercise is compiled using the modeling of double glass windows. The analysis (student assignment) is performed with the aid of COMSOL 3.4 with MATLAB.

Keywords: CFD; optimization; design; natural convection; double glass

INTRODUCTION

WITHIN THE FIELD of fluid mechanics, automatic optimization of design using CFD is not very widespread. Indeed up to present, very often models had to be tuned to get solutions that are in line with experiments. Therefore many CFD solutions were not predictive at all. Thus, the solutions obtained by means of CFD were used in two ways, i) to investigate trends when varying a parameter, without the need to be accurate and ii) to see what actually can happen in a flow system, thus to get insight and on the basis of this changing the design. With the advance of models, numerical methods and last but not least computing capacity, a large range of problems can be solved with good accuracy. Especially in 2-dimensional cases with low forcing, the accuracy is generally no issue anymore if one is prepared to invest sufficient computer capacity. It is still much more difficult for 3-dimensional problems, and with higher forcing leading to more non-linear behaviour. This increase in forcing also implies that one enters non-stationary solutions and with a further increase in turbulent conditions. This generally results in multi-scale problems. Since relatively simple flow problems can be solved with sufficient accuracy, we are at the edge of a paradigm shift in application of CFD, where it can actually be used for more or less automatic design purposes. It is important to confront students already with this shift since they will be the designers and users of CFD tomorrow. Especially for multi-physics problems at low forcing, good packages are available to perform calculations. COMSOL Multiphysics is such a package that will be used in demonstrating optimization problems. This class of problems is also an important class since in technology there are many attempts to miniaturize systems in which some kind of transfer and/or conversion takes place, e.g. lab on a chip.

Besides the dynamics of the flow problem, and therefore i) the dimensionality, ii) temporal and iii) spatial resolution, also the validity of the applied governing equations are subject to limitations within the optimization procedure. For example, in case of optimization of a fluids velocity of a compressible medium in some kind of a setup, constrained to e.g. pressure and viscous losses together with an additional transfer process (otherwise trivial solutions arise). In this case, the incompressible Navier–Stokes equations hold, whereas at higher velocities the compressible version must be applied. For the latter case, the governing equations are of a fundamental different mathematical nature, and thus not only the equations have to be changed but also the solution strategy in a CFD environment is fundamentally different. However, it is well known that for Mach numbers lower than 0.3, flows can be treated as incompressible. This since the Mach number appears in quadratic form in the dimensionless equations.

* Accepted 12 August 2009.

This means that a 10% error is taken for granted. This may not be allowed for optimization of transport systems in which a 1% drag reduction is a significant gain. In the present example of natural convection, the validity of the Boussinesq approach to buoyancy has a similar effect.

Because of the limitations to CFD solutions, automatic optimization methods have to be applied with great care and a very critical attitude. However, optimization methods will become increasingly more important in the near future. Thus, the message of the student assignment that is proposed in the present paper is only partly about the skills in applying the technique but for the larger part on interpreting and critical assessment of the methods and the results. Since students are intelligent people, teachers probably need to tell them what to think about, rather then what to think (agenda setting). This is also much more effective and an important paradigm in political sciences related to the influence of mass media [1], now with a much more positive connotation.

Because of the complexity of CFD and optimization, a relatively low amount of really relevant publications can be found in literature. A general overview is given by Mohammadi and Pironneau [2], although this is of a more mathematical nature with respect to optimization, less focusing on relevant physical applications. A nice and relevant application concerning mixing, which directly becomes quite complex, is given by Hertzog et al. [3]. Focussing on the technique of optimization in CFD, there is the study of Slawig [4], using COMSOL as in the present case (COMSOL was formerly called Femlab). To the author's knowledge, there are no classical examples or test problems for the present context and hence the present paper is regarded as a contribution to this issue. For a recent article on teaching basic CFD, see Fraser et al. [5].

The problem that is put forward for optimization studies is the thermal isolation of a window system, consisting of double glass. These systems are widely used nowadays and they are very important in the minimization of energy consumption for buildings and therefore very relevant to study. There is a long tradition in modeling and optimization of these systems, but there are many (especially older) studies that do justice to the critical assessment as proposed in the beginning of this introduction. A recent publication with simulation and the proposition of an alternative physical design is given in Ismail et al. [6].

In the next sections, a description is given of the basic problem, the geometry, basic assumptions, the optimization parameters, and hypotheses about what will happen and how to setup the calculations in COMSOL [7]. Then, some results of basic simulations will be shown and the assignment to be tackled by students will be formulated. The paper ends with some conclusions.

PROBLEM DESCRIPTION

Setup and equations

The very basic physical problem is a very simple description of a double glass window system. Students should notice that this is already a first set of assumptions which might falsify the entire study at the end so that more research is required. Therefore, also the importance of dimensionless numbers as frequently used in fluid mechanics should be a very prominent aspect in teaching. It is assumed that there are two glass sheets of certain thicknesses, d_1,d_2, separated with a gap of some size, L, containing a fluid. Of course there should be a certain width, W, and a certain height, H, of the system of which the first one is ignored on the basis of the assumption of 2-dimensionality of the problem and the second one is a parameter that is subject of investigation. The implicit assumption is that a system of large lateral extent at relatively low forcing is considered. Then the flow will be laminar, steady and 2-dimensional for low forcing. It is known from theory and observations that for gradually higher forcing, the first unsteadiness will still be a solution in the 2-dimensional plane. In the 2-dimensional problem, a hot (inside) and a cold side (outside) is considered. In the basic form, only conduction will be considered in the glass sheets and both convection and conduction will be treated in the gap space. Boundary conditions are all no-slip and isothermal at the hot and cold side and adiabatic everywhere else. At the internal boundaries between glass sheets and the gap, continuity of the heat flux is supposed. A result showing the basic form of the solution is given in 'Results of the reference problem' section.

It must be noted that the isothermal conditions are not very representative. Convection heat transfer both at the outside and at the inside must be taken into account. However, this is a good exercise for students as well in the final assignment. Of course, the convection cannot be handled in detail (describing the transport of all instantaneous structures at the outside and inside). Here, statistical modeling with an effective heat transfer coefficient is often used. In that case, the heat flux at the hot and cold wall is described as a function of the room and outside temperatures. Of course, for a (statistically) steady state the heat flux at all cross sections should be equal. This is a property that will be used later in the estimation of the accuracy of results or validity of the steady state assumption. Also radiation is very important and very frequently coatings are used to obtain a desired behaviour, not only minimizing the heat flux to the outside but also maximizing the irradiative transport inside. This is very interesting from a technical viewpoint but beyond the scope of the present goals. An example of radiation modeling is discussed in Ismail and Carlos Salinas [8].

Thus, for the physical description, the Boussinesq approximation will be used, and thus, using the summation convention, the system of equations is given by, mass, momentum and energy conservation, respectively as:

$$\frac{\partial u_i}{\partial x_i} = 0,$$
$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho_0}\frac{\partial p}{\partial x_i} - g\frac{\rho}{\rho_0}\delta_{i2} + \nu\frac{\partial^2 u_i}{\partial x_i^2},$$
$$\frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} = \kappa\frac{\partial^2 T}{\partial x_i^2},$$

with x_i and u_i the position and velocity vector respectively, t is the time (but most calculations will be performed using the assumption that the solution is stationary, so the terms involving time will be dropped), p is the pressure, g the acceleration of gravity, ρ the density, δ_{ij} the Kronecker delta (here indicating that buoyancy only works in the 2nd dimension) and ν is the kinematic viscosity. The thermal diffusivity of heat is given by κ,

$$\kappa = \frac{\lambda}{\rho c_p},$$

with λ, the conduction coefficient and c_p the heat capacity at constant pressure.

The buoyancy force and analysis

Since any system of equations must be used with care and often assume some constraints, here, some detail about the final version of the frequently applied system is presented. In the present situation, all physical properties are implicitly taken at a reference state given at the reference temperature, T_0, except for the density, which has a local value in the Boussinesq approximation. Therefore, in the buoyant forcing of the momentum equation, the local density and the reference value both appear. For small temperature deviations, $(T - T_0)$, this can be expressed in terms of the volumetric expansion coefficient, β*, by:

$$-\frac{\rho}{\rho_0} = \beta(T - T_0) - 1,$$

based on the ideal gas assumption and constant (thermodynamic) pressure (hence more specifically Charles's law that $\rho T = C$). However, the definition of β also allows for descriptions of liquids. Specifically for gases, the volumetric expansion coefficient is equal to $\beta = 1/T_0$. From this exercise it is clear that T_0 can be taken at an arbitrary reference, but that for best accuracy of the Boussinesq approximation, a representative value for the final solution is required. By taking out the hydrostatic situation (with zero velocity and constant density, ρ_0) of the momentum equation, an equation for the deviatoric pressure $p' = p - p_h$ is obtained. The solution for the hydrostatic pressure yields: $p_h = p_s - \rho_0 g x_2$, in which the subscript s denotes values at the assumed free surface and x_2 of which the origin is taken at the surface and with positive direction opposite to the direction of g. This results in a momentum equation formulated as,

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho_0}\frac{\partial p'}{\partial x_i} + g\beta(T - T_0)\delta_{i2} + \nu\frac{\partial^2 u_i}{\partial x_i^2}$$

Note here that for consistency, T_0 should be defined at the surface as well, or that the pressure has to be redefined again!

Now to get some idea about the governing parameters in the problem, it is important to non-dimensionalize the problem, to see what processes are important and how to influence these. Therefore, one needs to identify dimensionless quantities, which can be defined quite arbitrarily. Here, a balance between convection and conduction in the energy equation is taken, to define a velocity scale,

$$U = \frac{\kappa}{L}.$$

This gives a timescale as well, $\tau = L/U = L^2/\kappa$, and by using dimensionless quantities indicated by a superscript asterisk, the following can be written down: $u = Uu^*$, $x = Lx^*$ $t = \tau t^*$, $T - T_0 = \Delta T T^*$ and $p' = \rho_0 U^2 p^*$, in which a characteristic temperature difference ΔT, is assumed, imposed by the boundary conditions. In this way, the system of equations becomes,

$$\frac{\partial u_i^*}{\partial x_i^*} = 0,$$
$$\frac{\partial u_i^*}{\partial t^*} + u_j^* \frac{\partial u_i^*}{\partial x_j^*} = -\frac{\partial p^*}{\partial x_i^*} + Ra\,\mathrm{Pr}\,T^*\delta_{i2} + \nu\frac{\partial^2 u_i^*}{\partial x_i^{*2}},$$
$$\frac{\partial T^*}{\partial t^*} + u_j^* \frac{\partial T^*}{\partial x_j^*} = \frac{\partial^2 T^*}{\partial x_i^{*2}},$$

with the Prandtl and Rayleigh numbers which are the two important parameters of the system,

$$Pr = \frac{\nu}{\kappa},$$
$$Ra = \frac{g\beta\Delta T L^3}{\nu\kappa}.$$

This system can be solved alternatively as well, with the correct adjustments on the definition of the length scales and boundary conditions. This is ideal for systems in which all dimensions change with the characteristic dimension L. Then, there is similarity of the solution: if for two systems Pr and Ra are the same and the geometry is similar then the solution can be scaled back from the dimensionless solution. However, as will be seen, only a

* The formal definition of the volumetric expansion coefficient β, which is used, is:

$$\beta \equiv \frac{1}{V}\left(\frac{\partial V}{\partial T}\right)_p = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p.$$

Table 1. Properties of air, glass and problem properties. The hot and cold side have temperatures $T_0 + dT$ and $T_0 - dT$ respectively

Media properties	ρ kg/m^3	$\eta = \rho\nu$ $Pa\ s$	c_p $W/kg\ K$	λ W/mK
Fluid	1.2	$1.7 \cdot 10^{-5}$	1006	0.025
Solid	2500		840	1.1
Problem properties	g m/s^2	T_0 K	dT K	$\beta = 1/T_0$ K^{-1}
	9.81	300	20	0.0033

single length scale will be changed, then one cannot make use of similarity.

Optimization, hypotheses and procedure

The design of a double glass window system can be subject to many goals. Besides minimization of the outgoing heat flux and maximization of the ingoing heat flux, one might think of minimization of acoustic energy transfer, maximization of the mechanical strength and resistance to impact. This is all under the constraint of minimization of the associated costs. Here, minimization of the heat flux will be looked at, either inward or outward, which depends on the phase during the day and season and the local climate conditions. So, in principal, all these conditions have to be considered. Here, a base case will be taken and variations of certain parameters are studied. The gap size, L, is a good object of study and this parameter will be investigated with respect to the heat flux.

Before starting with CFD calculations and in order for them to be efficient, it is very useful to think about the outcome first. Therefore, the study is started with a short analysis of the problem. In the basic configuration, air is taken as the medium in the gap between the glass sheets. Now, one can argue that having a larger gap size leads to a larger insulation. However, this is under the condition that the gas is stagnant. For a non-stagnant flow there are two important parameters, the Rayleigh number and the Prandtl number of which the latter is fixed for a certain medium. Since air is taken as

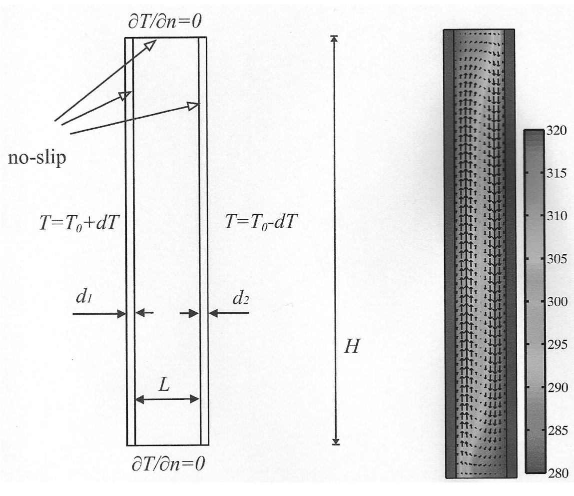

Fig. 1. Basic setup (left) and solution (right) of the temperature distribution in the window gap (indicated by pseudocolors) and velocity distribution with vectors. The maximal vector length is 6.2 cm/s.

the medium, only the influence of the Rayleigh number appears. The Rayleigh number determines the strength of the flow that is induced by the buoyancy. Therefore, it also determines the heat transfer at the glass walls. Now, the length scale that is incorporated in the Rayleigh number has a quite strong effect, since it goes with a 3rd power. So probably the heat transfer decreases with the gap size al lower values and increases with larger values. Then, probably, there is an optimal value.

In order to study the optimal situation, the system can be implemented in COMSOL. For later optimization, it is convenient to use the combination COMSOL with MATLAB. In the present study, version 3.4 was used. To that end, a basic setup is considered with a height of 0.1 m, thicknesses of the glass windows of 2 mm and gap size of 1 cm and defined properties for the windows and the fluid. These properties are given in Table 1, and the geometry and boundary conditions are shown in Fig. 1, together with results in the chapter on Basic results and optimization. To set up the problem in COMSOL, it is easiest to first use a geometry with certain dimensions, export it to MATLAB [9] and then change everything later in MATLAB.

In order to construct the case, COMSOL with MATLAB has to be started. For the models, one has to choose both the application modes of "Incompressible Navier-Stokes" and "Convection and Conduction" for heat transfer. The best is to choose the transient versions because these can be applied at steady state as well. The latter one can check for unsteadiness by changing the solver parameters. Then, first the geometry has to be specified, consisting of 3 rectangles. Boundary conditions and physical properties have to be specified in the Physics section. A mesh has to be initialized in the standard COMSOL way and in the Solver section, adaptive mesh refinement must be enabled. Here, 2 refinements are used in a default way. Furthermore, in constructing the case, it is important to fix the pressure to a certain value at a single point. The value itself is not important for the solution of velocities and temperature since the governing equations only contain a pressure gradient. It is just to prevent ill-conditioning of the system. The resulting basic MATLAB file is given in appendix A. This file can be run both in MATLAB and in COMSOL. The latter can be convenient for making additional changes in the setup, analyses of the result etc.

First, this test case has to be run to check whether a converged solution is obtained with a certain accuracy. Then a parameter variation can be carried out to study the influence of this parameter with respect to the optimization function (heat transfer). Each time it must be checked whether there is an accurate result over the entire parameter space. To that end, the accuracy is defined by having a very low value of the error, ε, defined as:

$$\varepsilon \equiv 2\frac{|q_1 + q_2|}{q_2 - q_1},$$

which is the mean absolute relative difference of the left (1) and right (2) heat fluxes, q, expressed in W/m, relative to the mean heat flux, $q_m = (q_2 - q_1)/2$*. With the hot side at the left, the heat flux is from left to right, and therefore, it is positive. However, the boundary integration in COMSOL employs a flux normal to the wall (outside direction). Therefore, at the hot side, the heat flux is negative, and at the cold side, it is positive. These integrals can be recognized in the script in appendix A, as `I1` and `I2`. Note that the default convergence of COMSOL is used which is equal to 10^{-6}. In the script in the appendix A it can be seen that a basic mesh is constructed, with 2 automatic refinements, applying the default refinement method of COMSOL.

BASIC RESULTS AND OPTIMIZATION

Results of the reference problem

For the base case, the stationary solution is given in Fig. 1. It can be observed that a large recirculation zone appears in the cavity. A mean heat flux of $q_m = 12.5$ W/m is found with an error of $\varepsilon = 5 \cdot 10^{-5}$, which seems sufficiently accurate. For a single grid refinement, the heat flux does not change significantly and the error is only one order of magnitude higher, which is allowable. For the basic calculation, the two grid refinements is on the safe side. Also, application of a method to calculate unsteady behaviour does not show any change with time. Of course there would have been a transient solution, when the system is initialized with an arbitrary temperature and velocity field, e.g. $(u_1(x_1, x_2, t = 0), u_2 \quad (x_1, x_2, t = 0), T(x_1, x_2, t = 0)) = (0, 0, T_0)$. From this basic calculation, it can be easily seen that with the conductivity, density and heat capacity of a typical glass it does not matter what the thickness of it is. Therefore, only the air layer is responsible for the heat transfer. This observation supports the hypothesis that the size of the gap is a parameter which is of importance in optimizing the system, the glass thickness is of course of importance for the mechanical strength and the cost of the system.

A very important aspect of CFD is validation and verification. The first one involves the comparison with some sort of truth, obtained from a real physical experiment or other simulations that are validated themselves. The verification consists of checking whether a solution is obtained that is correct for the governing system of equations and the problem definition. This generally requires a check of the numerical algorithms and the coding. Sometimes good results can be obtained for the wrong reason! This is beyond the scope of the present paper, and it is considered to be sufficient with the observation that smooth solutions are obtained which give heat flux results that are in the range of values that are observed experimentally as well, although there is the fact that not all physics is taken into account.

The optimization assignment

Depending on the prior knowledge of the students, the assignment can include the setting up of the basic problem, or the MATLAB dgbasic.m file as in the appendix can be provided. Also, the formulation of the optimization, constraints and the critical assessment of the results can be the subject of an assignment formulated for students. At the author's institution, an introductory course on CFD with COMSOL is requested. This course comprises first some discretization exercises for one-dimensional convection-diffusion problems in MATLAB. Here, Taylor series in space and time are introduced to arrive at discrete numerical schemes. The Euler forward, Euler backward and Crank-Nicholson methods are treated to discuss the issues of accuracy and stability. The same problem is then treated in COMSOL, after which setups of fluid mechanic basic problems are treated. To be specific i) stationary entrance flow in a channel and in a pipe, ii) unsteady von Kármán street behind a cylinder, iii) turbulent mixing of two passive markers in a T-junction with different angles of connection (this is based on turbulence modeling using the standard k-ϵ turbulence model), iv) compressible Euler flow over a wing as function of the angle of attack and v) natural convection flow in a differentially heated cavity. In this sense, setting up of the present problem is a smooth continuation of what they learned.

The paper continues with the optimization of the gap size and the solutions that will be encountered. The richness in the results will provide enough space for a critical assessment and ways of continuing the investigation. This can, therefore, also be left to the imagination of the students.

In the MATLAB script for COMSOL (appendix A), one can easily add a loop in which the gap size is varied. Then, results can be stored as function of the gap size and investigated afterwards. In this way, a very important result like the heat flux and accuracy as function of gap size can be evaluated. Such a result is displayed in Fig. 2, where the gap size is varied from 1 to 2 mm with a step size of 1

* Note that the heat flux dimension is per meter in the third direction. It would be even more convenient to define the mean heat flux over the height as well, if one wants to study the effect of the window height on the performance, which would be an important investigation in the present example.

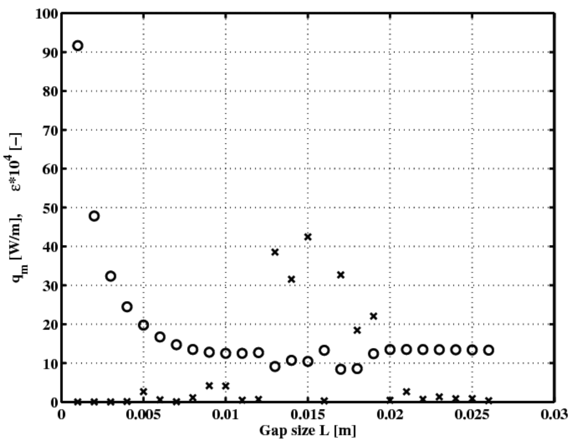

Fig. 2. Heat flux (o) and error (x) as function of the gap size.

mm. It can be observed clearly that the heat flux decreases with the gap size. However, starting at a gap size of 10 mm, the decrease is levelled of and there is almost no gain anymore. Also, it can be observed that there are some irregularities, specifically at 13, 14, 15, 17, 18 and 19 mm. At these values, also the error is relatively high and it was observed that no convergence was reported by MATLAB. So, these solutions are not valid. Looking a little closer to Fig. 2, it seems that a minimum is present at around 10–11 mm. Now, one can try to find the optimal situation by refining the step size and start at a gap size below the optimal value. Also one might want to know what is happening at the unconverged points. How does the unconverged solution look like? Should the grid be more refined in these points? Or is there an unsteady solution at these points, which would be remarkable, since there is a stable accurate solution at a gap of 16 mm?

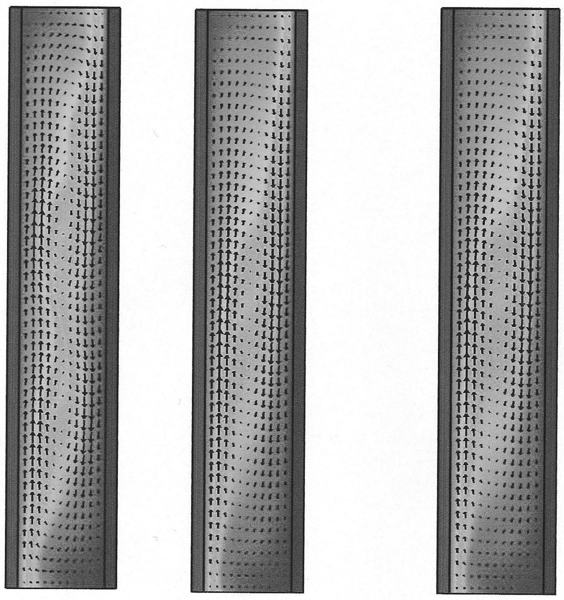

Fig. 3. Results for gap size of 15 (left two figures) and 16 mm (right), left unconverged coarse grid solution, (heat flux is 10.4 W/m, max. velocity is 13.1 cm/s, error is 0.0042), middle converged solution at comparable grid (heat flux is 13.2 W/m, max. velocity is 10.0 cm/s, error is $2{\cdot}10^{-5}$), right: 16 mm case (heat flux is 10.2 W/m, max velocity is 10.2 cm/s and error is $1.6{\cdot}10^{-4}$). Colorbar as in Fig. 1.

One might want to have a look at the gap size at which the error is largest, $L = 15$ mm. It is found that for this situation, the initial mesh is too coarse to find a first solution. Therefore, COMSOL can not perform the subsequent grid refinements. This is the case for all the cases in which no convergence could be found. Therefore, the outcomes in Fig. 2 for these cases are outcomes defined at the initial coarse grid. In principle, one would like to have a less strict convergence requirement for the coarse grid, and with the adaptive refinement of the grid also this requirement should decrease. In order to find a solution, a calculation was started at a refined grid and performed only a single grid refinement to obtain results that can be compared to the reference case and all other converged solutions. For this situation, the counterpart of Fig. 1 is displayed in Fig. 3, for the unconverged coarse grid case, the converged refined grid case and the (regularly) converged $L = 16$ mm case. Here, it can be observed that a solution with marginal differences can be obtained for cases with marginal changes in gap size, as is expected.

An additional analysis can be performed with respect to the hypotheses of 'Optimization, hypotheses and procedure' section. The question was whether the heat loss as function of the gap size would behave linear or at another rate due to the increasing degree of freedom with gap size and the associated Rayleigh number which scales with the gap size to the third power. To investigate this issue, a plot was made on a log-log scale to identify the behaviour. This is given in Fig. 4. From this figure, it can be clearly seen that at low gap sizes, there is a linear decrease of the heat flux as supported by the correlation with the line that is drawn. Indeed for small gap size, $L \leq 8$ mm, the velocities go down and asymptotically a stagnant air layer is approached. For larger gap size, the decrease levels of very quickly and a small increase can be observed. As stated before, an optimal value of the gap size is at about 10 mm.

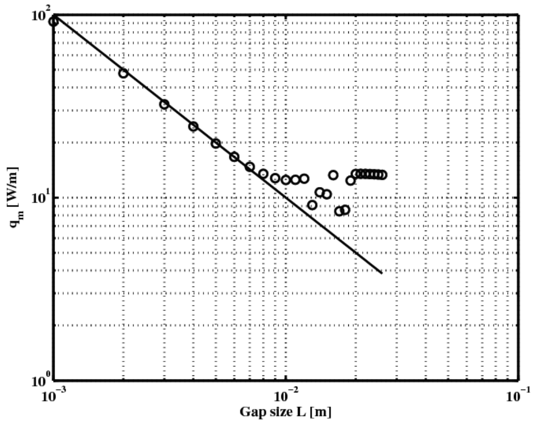

Fig. 4. Comparison of the heat flux found with COMSOL (o) with a $q_m \approx cL^{-1}$ behaviour (solid line, $c = 0.1$ W).

Table 2. Properties of argon and water

Media properties	ρ *kg/m*^{3}	$\eta = \nu\rho$ *Pa s*	c_p *W/kg K*	λ *W/mK*	β K^{-1}
Argon	1.45	$2.3 \cdot 10^{-5}$	520	0.018	0.0033
Water	996	$8.94 \cdot 10^{-4}$	4179	0.58	$2.8 \cdot 10^{-4}$

Possible extensions

The analysis, as started here, could be extended much further. The height could be varied as well as the material properties of the glass and the fluid. As an exercise, a suggestion would be to see what improvements can be expected when using argon or water (the latter for the sake of argument) as a fluid. See for properties Table 2. Moreover, more realistic temperature differences could be applied. This all potentially leads to different Rayleigh and Prandtl numbers, eventually leading to unsteady behaviour. Especially for low Prandtl numbers, higher mesh resolutions are required. Also, the scaling properties and related to that the accuracy could be studied by solving the dimensionless system of equations.

When progressing to more advanced methods, one could make more extensive use of optimization theory, although even more care has to be taken with the interpretation of results then was the case up to here. For that matter, the optimization toolbox of MATLAB can be used. For finding the minimum of a constrained nonlinear multi-variable function, as what is presently the case, the tool fmincon might be a very interesting option for further investigations.

CONCLUSIONS

In the present paper, optimization is considered for problems in fluid dynamics from a perspective of teaching students. A non-trivial example involving the isolation of a double glass window system, that can be used in an educational environment, is partly worked out. The example approaches the level of contemporary research in the associated field.

It is concluded that optimization will become a very important tool in modern optimal design technology. However, at all levels of the design analysis, care has to be taken about the assumptions and methods that are used as function of the parameters that are changed. This ranges from the validity of the physical modeling to the numerical implementation in all their aspects.

REFERENCES

1. B. C. Cohen, *The Press and Foreign Policy*. Princeton University Press, Princeton (1963).
2. B. Mohammadi, O. Pironneau, Shape Optimization in Fluid Mechanics, *Annual Rev. Fluid Mech.*, **36**, (2004), pp. 255–279.
3. D. E. Hertzog, B. Ivorra, B. Mohammadi, O. Bakajin, J. G. Santiago, Optimization of a microfluidic mixer for studying protein folding kinetics, *Anal. Chem.*, **78**(13), (2006), pp. 4299–4306.
4. T. Slawig, PDE-constrained control using FEMLAB—Control of the Navier–Stokes equations. *Numer. Algor.*, **42**, (2006), pp. 107–126.
5. D. M. Fraser, R. Pillay, L. Tjatindi, J. M. Case, Enhancing the learning of fluid mechanics using computer simulations, *J. Eng. Ed.*, **96**(4), (2007), pp. 381–388.
6. K. A. R. Ismail, C. T. Salinas, J. R. Henriquez, Comparison between PCM filled glass windows and absorbing gas filled windows, *Energy and Buildings*, **40**, (2008), pp. 710–719.
7. COMSOL Multiphysics. COMSOL 3.4 Multiphysics User's Guide. See also http://www.comsol.com.
8. K. A. R. Ismail, S. Carlos Salinas, Non-gray radiative convective conductive modeling of a double glass window with a cavity filled with a mixture of absorbing gases, *Int. J. Heat Mass Transf.*, **49**(17–18), (2006), pp. 2972–2983.
9. MATLAB: www.matlab.com.

APPENDIX A: OUTPUT OF BASIC COMSOL FILE TO MATLAB

```
% COMSOL Multiphysics Model M-file dgbas.m
% Generated by COMSOL 3.4 (COMSOL 3.4.0.248, $Date: 2007/10/10 16:07:51 $)

flclear fem

% COMSOL version
clear vrsn
vrsn.name = 'COMSOL 3.4';
vrsn.ext = '';
vrsn.major = 0;
```

```
vrsn.build = 248;
vrsn.rcs = '$Name: $';
vrsn.date = '$Date: 2007/10/10 16:07:51 $';
fem.version = vrsn;

% Geometry
g1=rect2(0.0020,0.1,'base','corner','pos',[0,0]);
[g2]=geomcopy({g1});
g2=move(g2,[0.012,0]);
g3=rect2(0.01,0.1,'base','corner','pos',[0.0020,0]);

% Analyzed geometry
clear s
s.objs={g1,g2,g3};
s.name={'R1','R2','R3'};
s.tags={'g1','g2','g3'};

fem.draw=struct('s',s);
fem.geom=geomcsg(fem);

% Constants
fem.const = {'rhoflu','1.2','kflu','0.025','cpflu','1006','etaflu','1.7e-5', ...
 'rhosol','2500','ksol','1.1','cpsol','840','g','9.81','T0','300', ...
 'dT','20','beta','1/T0'};

% Initialize mesh
fem.mesh=meshinit(fem,'hauto',5);

% (Default values are not included)

% Application mode 1
clear appl
appl.mode.class = 'FlNavierStokes';
appl.gporder = {4,2};
appl.cporder = {2,1};
appl.assignsuffix = '_ns';
clear prop
prop.analysis='static';
appl.prop = prop;
clear pnt
pnt.pnton = {0,1};
pnt.ind = [1,1,2,1,1,1,1,1];
appl.pnt = pnt;
clear bnd
bnd.type = {'int','walltype'};
bnd.ind = [1,1,1,2,2,2,2,1,1,1];
appl.bnd = bnd;
clear equ
equ.cporder = {{1;1;2}};
equ.eta = {1,'etaflu'};
equ.gporder = {{1;1;2}};
equ.usage = {0,1};
equ.F_y = {0,'g*beta*(T-T0)'};
equ.rho = {1,'rhoflu'};
equ.ind = [1,2,1];
appl.equ = equ;
fem.appl{1} = appl;

% Application mode 2
clear appl
appl.mode.class = 'FlConvCond';
appl.assignsuffix = '_cc';
clear prop
prop.analysis='static';
clear weakconstr
weakconstr.value = 'off';
weakconstr.dim = {'lm4'};
prop.weakconstr = weakconstr;
appl.prop = prop;
clear bnd
bnd.type = {'T','q0','cont','T'};
bnd.T0 = {'T0+dT',0,0,'T0-dT'};
```

```
bnd.ind = [1,2,2,3,2,2,3,2,2,4];
appl.bnd = bnd;
clear equ
equ.C = {'cpsol','cpflu'};
equ.rho = {'rhosol','rhoflu'};
equ.k = {'ksol','kflu'};
equ.v = {0,'v'};
equ.u = {0,'u'};
equ.ind = [1,2,1];
appl.equ = equ;
fem.appl{2} = appl;
fem.frame = {'ref'};
fem.border = 1;
clear units;
units.basesystem = 'SI';
fem.units = units;

% ODE Settings
clear ode
clear units;
units.basesystem = 'SI';
ode.units = units;
fem.ode = ode;
% Multiphysics
fem = multiphysics(fem);

% Extend mesh
fem.xmesh = meshextend(fem);

% Solve problem
fem = adaption(fem,'blocksize',1000,'solcomp',{'v','T','u','p'},...
'outcomp',{'v','T','u','p'},'hnlin','on','solver','stationary', ...
 'l2scale',[1],'l2staborder',[2],'eigselect',[1], ...
 'maxt',10000000,'ngen',2,'resorder',[0],'rmethod','longest', ...
 'tppar',1.7,'linsolver','pardiso','uscale','none', ...
 'geomnum',1);

% Save current fem structure for restart purposes
fem0=fem;

% Plot solution
postplot(fem,'tridata',{'T','cont','internal','unit','K'},'trimap','jet(1024)',
   ...
 'arrowdata',{'u','v'},'arrowxspacing',15,'arrowyspacing',50, ...
 'arrowtype','arrow','arrowstyle','proportional','arrowcolor',[0.0,0.0,0.0], ...
 'title','Surface: Temperature [K] Arrow: Velocity field');

% Integrate
I1=postint(fem,'ntflux_T_cc','unit','W/m','dl',[1],'edim',1)

% Integrate
I2=postint(fem,'ntflux_T_cc','unit','W/m','dl',[10],'edim',1)
```

Robert Johan Maria Bastiaans received his Ph.D. from the Department of Mechanical Engineering at Eindhoven University of Technology, Netherlands in 1997. The subject was on modeling and measuring turbulent natural convection flows. He obtained his M.Sc. on aero-acoustics with honors in 1991 at the same university. Rob is currently an assistant professor at the Eindhoven University of Technology. He teaches courses on Energy from Biomass, Numerical Simulation of Heat and Flow Problems, Design Optimization and Introduction to Mechanical Engineering. His research is in the field of modeling turbulent combustion processes. He published about 25 scientific journal papers.