Numerical optimization of scavenging in two-stroke engines with transfer ducts, an exhaust port and a moving piston

L. Klassen, A. Klimmek, D. Kröner and D. Trescher*

Institute for Applied Mathematics, University of Freiburg, Hermann-Herder-Straße 10, D-79104 Freiburg, Germany, e-mail: dietmar@mathematik.uni-freiburg.de,
* Fa. Andreas Stihl AG & Co, D-71336 Waiblingen, Germany

Abstract. For the benefit of the environment, the HC–emission of two-stroke engines has to be reduced. This can be done by reducing the losses of scavenging by improving the geometry of the transfer ducts and the exhaust port. Numerical simulations of the flow through the two-stroke engine should be performed for different geometries in order to reveal the geometry with an optimal scavenge process. The simulations can help to accelerate the development of new two-stroke engines. The underlying mathematical model consists of the compressible Navier-Stokes equations in the cylinder with a moving piston. For the discretization we use a stabilized finite volume scheme on a hexahedral mesh. Up to now we have developed a numerical code for computing the flow in the cylinder and the most important integral quantities such as trapping efficiency and the percentage of exhaust gas at the exhaust port. Now we are able to analyze quantitatively the scavenge process and to estimate the quality of different drafts for the geometrical design.

1 Description of the problem

The main purpose of this project is the improvement of the geometry of two-stroke engines in order to reduce the pollutant emission.

Currently, two-stroke engines are used in great quantities for motorcycles, marines and ship drives, for motorcar and aircraft models, and especially in hand-operated power tools such as chainsaws, brushcutters, etc. Advantages of two-stroke engines over the four-stroke ones consist in favourable ratios of their efficiency and petrol consumption to scavenge volumes. However, the disadvantage is worse exhaust data. To attain a drastic lowering of the exhaust gas, new concepts for improvement of the design have to be developed and employed. One approach is to optimize the geometry of two-stroke engines.

The fresh gas enters the cylinder of the engine (Figure 1a) when the moving piston opens the transfer ducts. The gas is pushed inside the cylinder by the excess pressure in these ducts, achieved as a result of compression of the fresh charge below the piston by its downward motion. The process of inducing the fresh gas into the cylinder is called the scavenge process. With a bad geometry, the fresh gas streams directly into the exhaust port (see Figure 2, right stream line). It is, therefore, totally lost for the next ignition, it diminishes the power and burdens the environment. In case of a good geometry, the deflagrated gas is ejected into the exhaust port

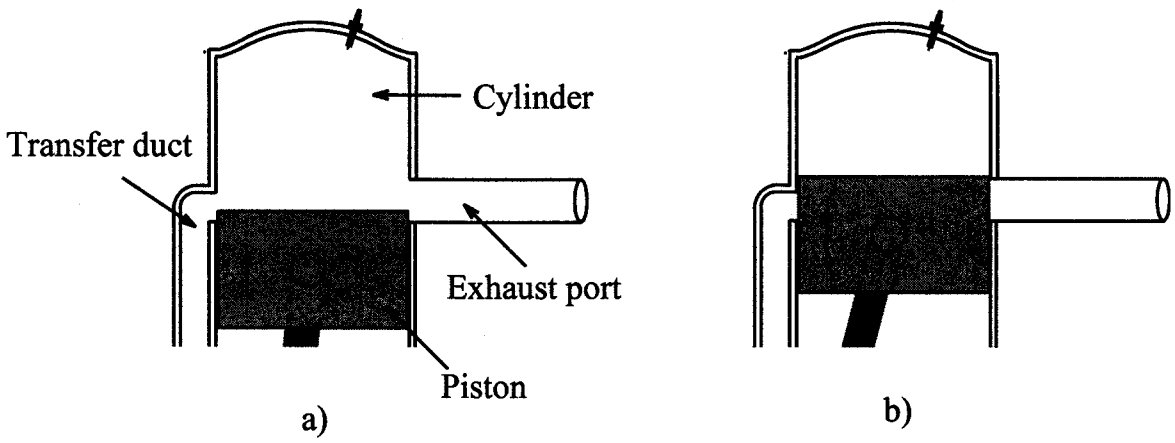

Figure 1: Motion of the piston in the two-stroke engine

by the fresh charge rising from the transfer duct, and the cylinder is filled with a mixture of the fresh gases. In this case, the fresh charge can also reach the exhaust port, and some part of the deflagrated gas can remain in the cylinder. After the transfer duct and the exhaust port have both been closed by the rising piston (Figure 1b), the compression process in the cylinder begins. As the piston moves downwards, the exhaust port opens, and the deflagrated gas streams in the form of a pressure shock-type wave from the cylinder into the exhaust port [1, p. 52]. The reflection of the pressure wave at the tapered shape of the exhaust pipe can be used to return the fresh gas, which has already arrived at the exhaust port, back into the cylinder (the download effect). The most efficient possibility for reducing the pollutant emission is to minimize the loss of the scavenging. By means of the numerical simulation for the gas motion in the cylinder, one can determine which changes of the geometry of the cylinder, transfer ducts and exhaust port can improve the composition of the gas mixture and minimize the losses of scavenging. In order to analyze the quality of new geometries, the fresh and deflagrated gas fluxes have to be computed in the transfer ducts and in the exhaust pipe and have to be compared for these geometries.

2 Mathematical model

The mixture of the fresh and deflagrated gases is considered as a compressible fluid characterized by the total mass density ρ and the velocity vector $\mathbf{v}$. If σ, $\mathbf{v}_\sigma$ and τ, $\mathbf{v}_\tau$ denote respectively the densities and the velocities for the fresh and deflagrated gases, then we have by definition: $\rho = \sigma + \tau$, $\mathbf{v} = (\sigma\,\mathbf{v}_\sigma + \tau\,\mathbf{v}_\tau)/\rho$. Hence, the mixture consists of two components filling one and the same volume. The concentration of the fresh gas in the mixture can be characterized by the ratio $c := \sigma/\rho$. We assume that, in each fixed point of the space, the change of the concentration c occurs only due to convective effects. This means that diffusion of the components in the mixture is neglected. The latter implies $\mathbf{v}_\tau = \mathbf{v}_\sigma = \mathbf{v}$. The combustion effects are also ignored. A further assumption is that, for each gas, the thermodynamic parameters and the equations of state can be introduced independently of the presence of the other component. In accordance with this assumption, the energy of the interaction between the gases is neglected, and the density of the total internal energy is equal to the sum of the internal energy densities for the components. It is assumed that both components of the mixture are perfect gases. In each particle of the mixture, the temperatures of the components are assumed to be equal: $T_\sigma = T_\tau =: T$, while

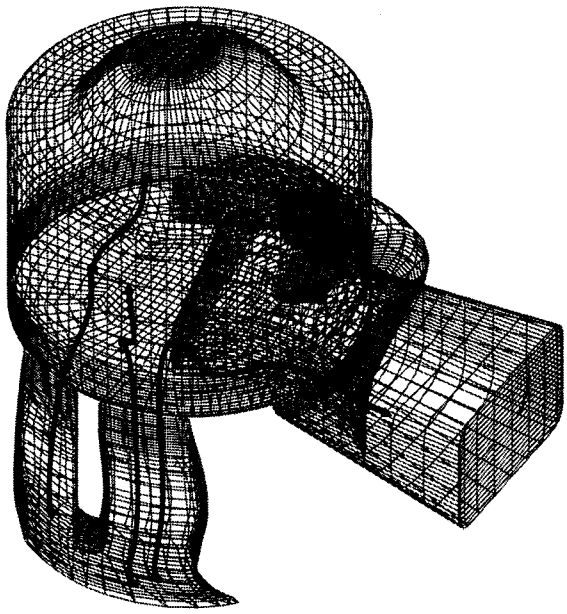

Figure 2: Possible motion of the fresh gas particles in the two-stroke engine

the pressure of the mixture is the sum of the partial pressures: $p = p_\sigma + p_\tau$ (Daltons law, see [21]). Under the above assumptions, the mathematical model for a motion of the mixture is given by the compressible Navier-Stokes equations for the mixture, complemented with the mass conservation equation for the fresh gas and with the perfect-gas equation of state for the mixture. This model can be written as follows (see [17], [21]):

$$\partial_t \mathbf{u} + \nabla \cdot \mathbf{f}(\mathbf{u}) - \nabla \cdot \mathbf{h}(\mathbf{u}) = \mathbf{0} \qquad \text{in } (\Omega_{cyl} \cup \Omega_{duct} \cup \Omega_{ex}) \times \mathbb{R}_0^+, \tag{1}$$

with

$$\mathbf{u} := \begin{pmatrix} \rho \\ \sigma \\ \rho v_1 \\ \rho v_2 \\ \rho v_3 \\ e \end{pmatrix}, \quad \mathbf{f} := \begin{pmatrix} f_1 \\ f_2 \\ f_3 \end{pmatrix}, \quad \mathbf{h} := \begin{pmatrix} h_1 \\ h_2 \\ h_3 \end{pmatrix}$$

and

$$f_k(\mathbf{u}) := \begin{pmatrix} \rho v_k \\ \sigma v_k \\ \rho v_1 v_k + \delta_{k1} p \\ \rho v_2 v_k + \delta_{k2} p \\ \rho v_3 v_k + \delta_{k3} p \\ (e+p) v_k \end{pmatrix}, \quad h_k(\mathbf{u}) := \begin{pmatrix} 0 \\ 0 \\ \tau_{k1} \\ \tau_{k2} \\ \tau_{k3} \\ \kappa \partial_k T + v_1 \tau_{k1} + v_2 \tau_{k2} + v_3 \tau_{k3} \end{pmatrix} \quad \text{for } k \in \{1,2,3\},$$

$$T = \frac{p}{\rho R}, \tag{2}$$

$$p = \left(\frac{\gamma_\tau c_{v\tau} (\rho - \sigma) + \gamma_\sigma c_{v\sigma} \sigma}{c_{v\tau} (\rho - \sigma) + c_{v\sigma} \sigma} - 1 \right) \left(e - \frac{\rho}{2} (v_1^2 + v_2^2 + v_3^2) \right). \tag{3}$$

Here Ω_{cyl}, Ω_{duct} and Ω_{ex} denote domains of the cylinder, transfer ducts and exhaust port respectively,

$$\tau_{ij} := \mu(\partial_i v_j + \partial_j v_i) - \tfrac{2}{3}\mu(\nabla \cdot \mathbf{v})\delta_{ij}, \quad i,j \in \{1,2,3\}$$

are the components of the viscous part of the stress tensor for the mixture, μ is the viscosity coefficient of the mixture, v_i are the components of the mixture velocity $\mathbf{v}$, δ_{ij} is the Kronecker symbol, e is the density of the total energy for the mixture (the sum of the internal and kinetic energies), κ is the heat transfer coefficient of the mixture, $\gamma_\sigma := c_{p\sigma}/c_{v\sigma}$, and $\gamma_\tau := c_{p\tau}/c_{v\tau}$ with $c_{p\sigma}$, $c_{p\tau}$, $c_{v\sigma}$, $c_{v\tau}$ denoting the specific thermal capacities of the gases under constant pressure and constant volume respectively. In the equation of state (2), $R := (\sigma R_\sigma + (\rho - \sigma) R_\tau)/\rho$ where R_σ and R_τ are the gas constants for the fresh and deflagrated gas correspondingly. The relationship (3) is a consequence of Daltons law and of the perfect-gas equation of state assumed for both gases.

Equation (1) contains six unknown functions, they are components of the vector $\mathbf{u}$. In the expressions for f_k, h_k, the functions T and p are determined by the components of $\mathbf{u}$ by means of equations (2) and (3).

It can be shown that under the assumption

$$|c_{v\tau}/c_{v\sigma} - 1| << 1, \qquad |\gamma_\tau/\gamma_\sigma - 1| << 1, \tag{4}$$

equation (3) is reduced to

$$p = (\gamma_\sigma - 1)\left(e - \frac{\rho}{2}(v_1^2 + v_2^2 + v_3^2)\right). \tag{5}$$

Assuming inequalities (4) are valid, we have used equation (5) instead of (3) for numerical simulations. The favourable property of using equation (5) is that the Navier-Stokes equations and the equation for the mass conservation of the fresh gas are not coupled any more.

For $t = 0$ (the piston is at top dead center, i.e. at its highest position) we assume

$$\begin{array}{rrrrrr}
\rho\,|_{\Omega_{duct}} = & 1.02\,kg/m^3, & \rho\,|_{\Omega_{cyl}} = & 5.55\,kg/m^3, & \rho\,|_{\Omega_{ex}} = & 0.41\,kg/m^3, \\
\sigma\,|_{\Omega_{duct}} = & 1.02\,kg/m^3, & \sigma\,|_{\Omega_{cyl}} = & 0\,kg/m^3, & \sigma\,|_{\Omega_{ex}} = & 0\,kg/m^3, \\
(\rho\mathbf{v})\,|_{\Omega_{duct}} = & \mathbf{0}\,kg/m^2\,sec, & (\rho\mathbf{v})\,|_{\Omega_{cyl}} = & \mathbf{0}\,kg/m^2\,sec, & (\rho\mathbf{v})\,|_{\Omega_{ex}} = & \mathbf{0}\,kg/m^2\,sec, \\
p\,|_{\Omega_{duct}} = & 9.199 \times 10^4\,Pa, & p\,|_{\Omega_{cyl}} = & 4.9 \times 10^6\,Pa, & p\,|_{\Omega_{ex}} = & 1.088 \times 10^5\,Pa.
\end{array}$$

The values for ρ and p are the outcome of calculations performed in a one-dimensional model of the scavenge process, that incorporates also the gas motion effect in the crankcase located below the piston. The temperature T and the total energy density e are calculated with the help of the initial values for ρ, p and $\mathbf{v}$ given above.

On the fixed solid boundaries Σ_{duct}, Σ_{cyl} and Σ_{ex}, the no-slip and adiabatic-wall assumptions hold

$$\mathbf{v} = \mathbf{0}, \quad \partial T/\partial n = 0,$$

where $\partial/\partial n$ is the partial derivative in the direction normal to the boundary. On the piston surface Σ_{pist} moving with the time-dependent velocity $\mathbf{v}_{pist}(t)$, we have

$$\mathbf{v} = \mathbf{v}_{pist}(t), \quad \partial T/\partial n = 0.$$

In accordance with equations (2) and (5), the internal energy per unit mass of the mixture (that is $e/\rho - 0.5(v_1^2 + v_2^2 + v_3^2)$) depends linearly on the temperature T. Therefore the adiabatic boundary conditions imply $\partial(e/\rho)/\partial n = 0$. On the inflow boundary Σ_{in} we use the time-dependent values

for ρ, p and $\mathbf{v}$ determined by the one-dimensional model of the scavenge process, and we put $\sigma = \rho$ (T and e follow from the relationships (2) and (5)). On the outflow boundary Σ_{out} we use again p from the one-dimensional scavenge-process simulations (ρ, σ, $\mathbf{v}$ are calculated by extrapolation from the inner domain).

Note finally that the Reynolds number calculated by use of the maximum inflow velocity ($58.3\,m/s$) and the maximal diameter of the transfer duct ($0.03\,m$) as characteristic velocity and length scales respectively, has found to be approximately equal to 1.2×10^5.

3 Numerical methods

For the discretization of the compressible Navier-Stokes equations we use an explicite time dependent upwind finite volume scheme on an irregular hexahedral 3D mesh. The stabilization of the numerical scheme is necessary since the Reynolds number is of the order 10^5. The grid is generated on the basis of CAD data of the geometry of the two-stroke engine with the help of [11]. The CAD data are produced by the Fa. Stihl. For the convective terms we use the AUSMDV Riemann solver, which has been proved to be the most effective one compared to several others [6]. The discretization of the diffusive terms is performed by use of the gradients in the direction of the normals to the cell-faces. The complete algorithm has been validated in [6] for examples with known explicit solutions in 2D and 3D.

The moving piston cuts the cells of the grid and the remaining fractions of the cells are considered as ordinary cells of the grid. If these cells become too small and imply a bad CFL condition, then these small parts are added to larger neighbouring cells [23].

This first-order scheme has been implemented on a shared memory parallel computer with 46 processors (SGI ORIGIN 2000), and the second-order scheme is being implemented now. We expect the computations to run relatively fast, so that it will be possible to simulate the flow through different engine geometries in reasonable time and to optimize the geometry iteratively.

Up to now we have performed computations with 400000 hexahedrons and 200000 time steps.

Adaptive local grid refinement [14], [18]–[20] with a dynamical load balancing [5], [22] and higher order discretizations [12], [16] on nonconformal unstructured hexahedral mesh have been implemented and tested for the unsteady Euler equations in 3D [2], [3], [8], [23]–[25] and will be extended to this Navier-Stokes solver.

For the visualization we have used GRAPE [9], [10], [10], [26].

4 Numerical results

In Figure 2, the cylinder geometry and the particle paths are presented for three particles of the fresh gas entering the cylinder. One particle is transported directly to the exhaust port. Such a kind of motion, called short circuit streaming, has to be avoided. The other two particles flow into the cylinder. Our numerical simulations show that the flow is governed by the pressure difference between the inflow of the ducts and the outflow of the exhaust pipe.

Up to now comparative numerical simulations have been performed for the two different geometries of the engine: "open geometry" and "handle geometry" (see Figure 3). For the qualitative and quantitative study of these geometries, we have calculated the percentage of the fresh gas

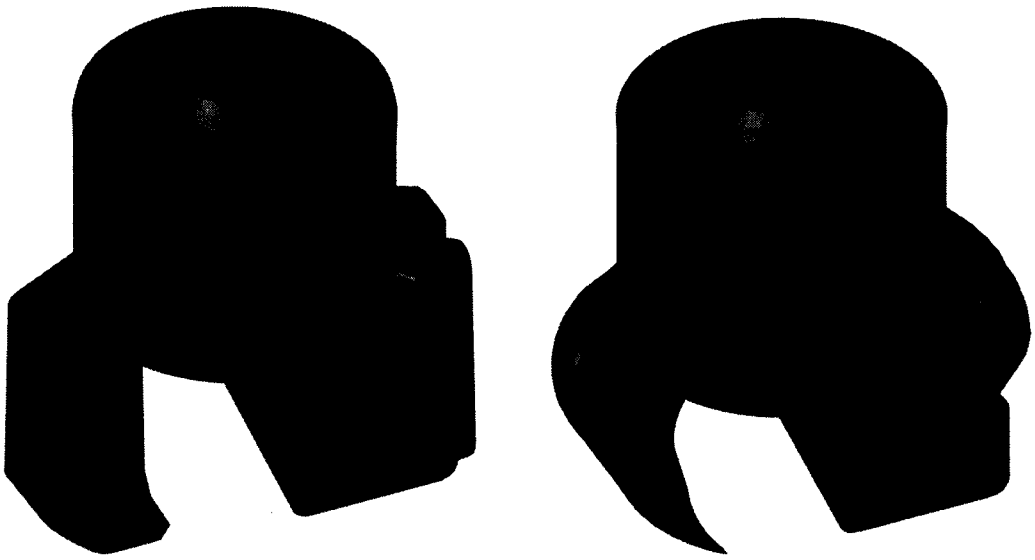

Figure 3: Two-stroke engine: open geometry (left) and handle geometry (right)

inside the cylinder as well as the mass fluxes at the inflow of the transfer ducts and at the outflow of the exhaust pipe. At the outflow of the exhaust pipe, we have calculated also the mass flux for the deflagrated gas. This allowed us to determine the dimensionless parameter

$$\text{percentage of exhaust gas} := \left(1 - \frac{\text{fresh gas mass flux at outlet}}{\text{total gas flux at outlet}}\right) \times 100\%$$

which is a very important measure for the fraction of the fresh gas reaching the exhaust during one scavenge period. If two geometries have equal fluxes of the fresh gas at the inlet, then the better geometry is that one which has the larger percentage of exhaust gas.

The results for the "open geometry" are shown in Figure 4 and those for the "handle geometry" in Figure 5. These numerical calculations have been performed on grids with 250000 elements. Presented are respectively the mass fluxes at the cylinder inflow and outflow, the fresh gas portion in the cylinder, and the percentage of exhaust gas as functions of the crank angle, CA (that defines the piston position). It is seen that for the "open geometry", the total flux of the fresh gas (that is its inflow for the scavenge period) is larger than that for the "handle geometry" (compare Figure 4a and 5a, solid lines). The fresh gas portions in the cylinder are, however, nearly the same. That is, because, for the "handle geometry", less fresh gas reaches the exhaust port. The latter is also a cause for a larger percentage of exhaust gas for this geometry.

Typical dimensionless parameters [1, p. 44] for comparison with the physical experiment are

$$\text{scavenge ratio} := \frac{m}{m_{ref}}$$

and

$$\text{trapping efficiency} := \frac{m_{trap}}{m},$$

where m is the mass of the fresh gas scavenging through the cylinder inflow, m_{ref} is the reference mass (the mass of the air filling the cylinder under the normal atmospheric pressure with the largest cylinder volume), and m_{trap} is the fresh gas mass that is in the cylinder at the end of the scavenging period. The following relationship holds: $m{=}m_{trap} + m_1$ where m_1 is the mass of the fresh gas, that flows in the exhaust pipe by short circuit streaming.

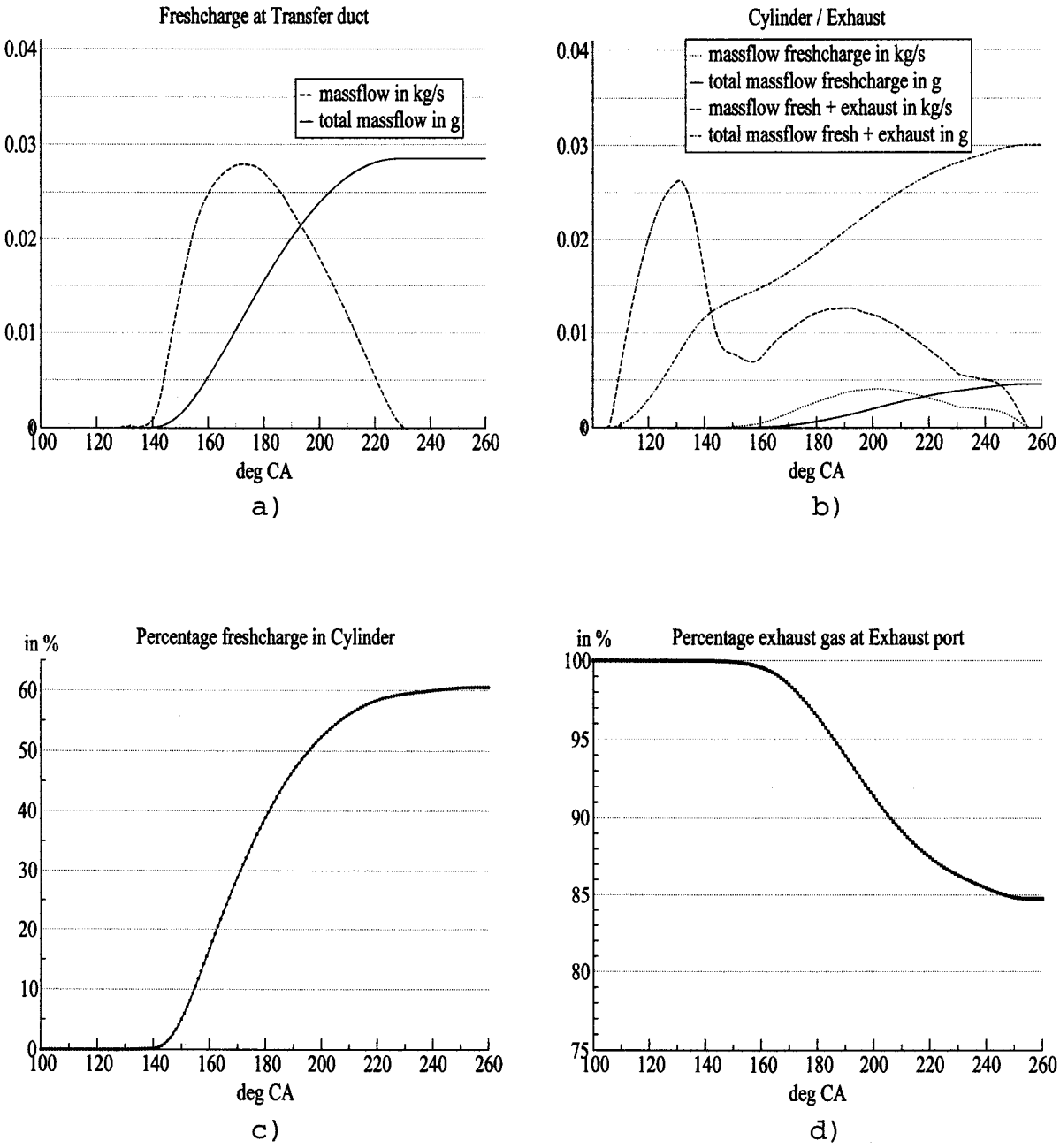

Figure 4: Numerical results for the open geometry

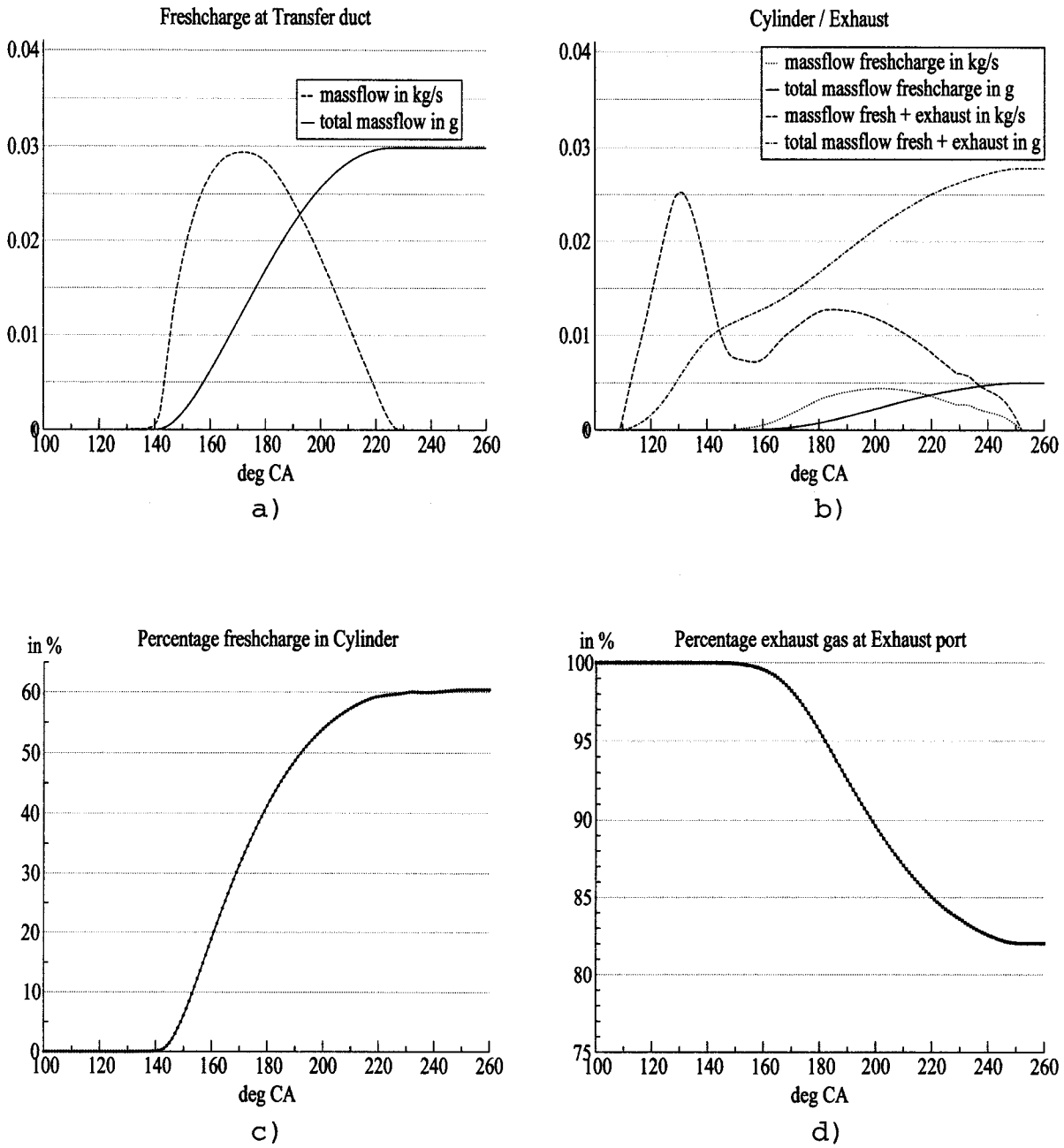

Figure 5: Numerical results for the handle geometry

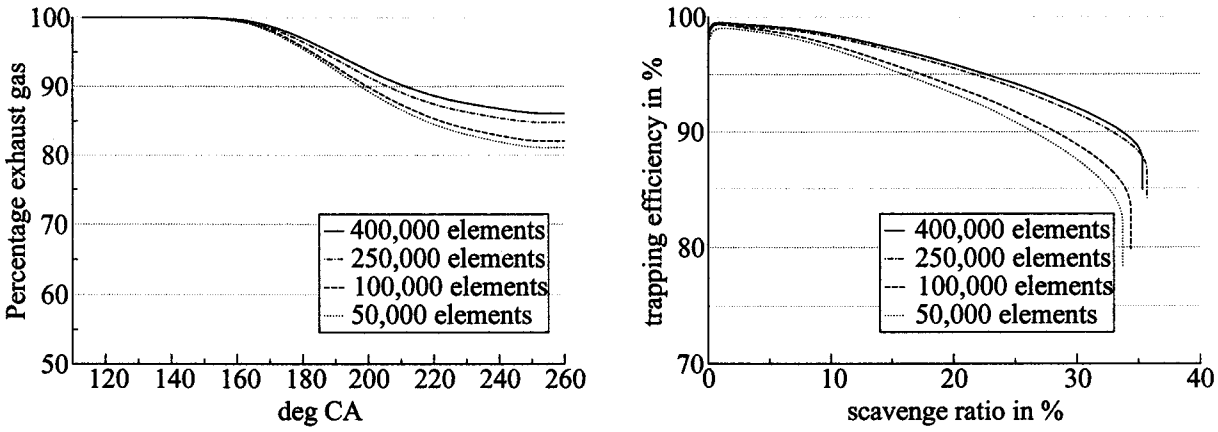

Figure 6: The percentage of exhaust gas and the trapping efficiency for the handle geometry

In Figure 6, the percentage of exhaust gas and the trapping efficiency for the "handle geometry" are presented. They are compared here for calculations performed respectively at grids with 50000, 100000, 250000, and 400000 elements. The results for the percentage of exhaust gas show yet some dependence of the solution from the grid, so that further refinement of the resolution is still needed. The trapping efficiency is presented as function of the scavenge ratio. This dependence shall be needed for comparison with physical experiments (compare [1, p. 44]). Some small dependence on the grid is also seen here. Yet an increase of the number of cells would result in long computing times. An effective possibility to improve the resolution on grids with a moderate number of elements is a higher-order discretization. In our test simulations [12] we used the reconstruction of Durlovsky, Engquist, Osher [4], [7], [13, p. 213] and showed that in order to reach the same accuracy, the calculation can be done on a much coarser grid in considerably less time.

References

[1] C.P. Blair, *Design and simulation of two-stroke engines*, Warrendale, PA: Soc. of Automotive Engineers 623 p. (1996).

[2] A. Dedner, D. Kröner, C. Rohde, and M. Wesenberg, *Godunov-Type Schemes for the MHD Equations*, submitted for conference proceedings, Universität Freiburg (1999).

[3] A. Dedner, C. Rohde and M. Wesenberg, *A MHD-Simulation in Solar Physics*, in R. Vilsmeier, F. Benkhaldoun, D. Hänel (Eds), Finite Volumes for Complex Applications II: Problems and Perspectives, Hermès Science Publ., Paris, pp. 491–498 (1999).

[4] L.J. Durlofsky, B. Engquist and S. Osher, *Triangle based adaptive stencils for the solution of hyperbolic conservation laws*, J. Comp. Phys. 98, pp. 64–73 (1992).

[5] A. Dedner, C. Rohde, B. Schupp and M. Wesenberg, *A parallel, load balanced MHD Code on locally adapted, unstructured grids in 3D*, in preparation.

[6] A. Egelja, D. Kröner, R. Schwörer, N. Lanson, M. Mancip and J.P. Vila, *Combined finite volume and smoothed particle method*, CNRS-DFG Collaborative Research Programme, Results 1996 – 1998, E. H. Hirschel (ed.), Vieweg, 50–74 (1999).

[7] B. Engquist and S. Osher, *One-sided difference approximations for nonlinear conservation laws*, Math. Comput. 36 pp. 321–351 (1981).

[8] T. Geßner, *Adaptive explizite und implizite numerische Simulation reaktiver Str"omungen*, Dissertation, Freiburg University, in preparation.

[9] *GRAPE. GRAphics Programming Environment: Reference Manual*, http://www.mathematik.uni-freiburg.de/Grape/DOC/HTML/manual.html, Institut für Angewandte Mathematik, Universität Freiburg (1996).

[10] R. T. Happe and M. Rumpf, *Characterizing global features of simulation data by selected local icons*, In Virtual Environments and Scientific Visualization '96, (1996).

[11] *ICEM CFD Hexa*, ICEM CFD Engineering, (1999).

[12] L. Klassen and D. Kröner, *Discretization of higher order for conservation laws on nonconformal unstructured rectangular grids*, in preparation (2000).

[13] D. Kröner, *Numerical schemes for conservation laws*, Wiley und Teubner, 508 p. (1996).

[14] D. Kröner und M. Ohlberger, *A posteriori error estimates for upwind finite volume schemes for nonlinear conservation laws in multi dimensions* Math. Comput. 69, No. 229, 25–39 (2000).

[15] D. Kröner, M. Ohlberger and Ch. Rohde (Eds.) *An Introduction to Recent Developments in Theory and Numerics for Conservation Laws. Proc. of the international school, Freiburg, Littenweiler, Germany, Okt. 1997*, Lecture Notes in Computational Science and Engineering, Vol. 5, Springer (1998).

[16] D. Kröner and M. Rokyta, *Higher order error estimates for time-dependent convection dominated diffusion equations in 2-D*, Manuscript, Freiburg University (1999).

[17] L. D. Landau and E. M. Lifschitz, *Theoretical Physics*, Vol. VI (1995).

[18] M. Ohlberger, *Adaptive mesh refinement for single and two phase flow problems in porous media*, In Proc. of the 2nd International Symposium on: Finite volumes for complex applications - problems and perspectives, Duisburg, 1999, pp. 761–768, Hermes Science Publications, Paris (1999).

[19] M. Ohlberger, *A posteriori error estimate for finite volume approximations to singularly perturbed nonlinear convection-diffusion equations*, Preprint 03-99, Freiburg (1999), accepted for publication in Num. Math. (2000).

[20] M. Ohlberger, *A posteriori error estimates for cell centered finite volume approximations of convection-diffusion-reaction equations*, in preparation.

[21] L. I. Sedov, *A course in continuum mechanics*, Vol. 2, Wolters - Noordhoff Publ. (1972).

[22] B. Schupp, *Entwicklung eines effizienten Verfahrens zur Simulation kompressibler Strömungen in 3D auf Parallelrechnern*, Dissertation, Universität Freiburg (2000).

[23] M. Wierse, *Higher Order Upwind Schemes on Unstructured Grids for the Compressible Euler Equations in Timedependent Geometries in 3D*, Dissertation, Universität Freiburg (1994).

[24] M. Wierse and D. Kröner, *Higher order upwind schemes on unstructured grids for the nonstationary compressible Navier-Stokes equations in complex time-dependent geometries in 3D*, Preprint No. 2, Freiburg University (1996).

[25] M. Wierse, D. Kröner, A. Müller, B. Schupp and R. Schwörer *Simulation of a 3-D piston driven flow*, R. Friedrich (ed.) et al., Computation and visualization of three-dimensional vortical and turbulent flows. Proc. of the 5th CNRS-DFG workschop on Numerical flow simulation, München, Germany, Dec. 6–7, 1996. Wiesbaden: Vieweg. Notes Numer. Fluid Mech. 64, 333–349 (1998).

[26] M. Wierse and M. Rumpf, *GRAPE, Eine interaktive Umgebung für Visualisierung und Numerik*, In Informatik, Forschung und Entwicklung, Vol. 7, pp. 145–151 (1992).