Heat transfer in gas filled pipes with closed warm end under different orientations

Hieke M., Haberstroh Ch., Quack H.

Technische Universitaet Dresden, Lehrstuhl fuer Kaelte- und Kryotechnik, D 01062 Dresden, Germany

In cryogenics it is a standard arrangement, that pipes filled with a stagnant gas, which lead from a high temperature to a low temperature, are installed with the warm end up. But sometimes the overall configuration does not allow this and pipes have to be arranged e.g. horizontally or even with the warm end down. How large is the additional heat transport by internal natural convection in such lines? To find this out, CFD-simulations in straight pipes have been done. The heat transport was evaluated from the simulations as function of orientation, diameter, length and wall thickness.

SETUP AND BOUNDARY CONDITIONS

Based on a straight pipe with a closed warm end and an open cold end, a model (Fig. 1) for a CFD-simulation is created. The model contains the fluid region as well as the solid walls. The warm and cold temperatures are applied to the pipe ends. The other outer walls are specified as adiabatic walls. The inner walls are set as domain interfaces between fluid and solid. This means there can be a heat flux but no flow. To simulate the cold open end, the length of the tube is extended by a section with fixed cold wall temperature.

A structured grid is applied to the model and the basic parameters are set. The warm end is connected to the surrounding temperature at 298 K, the cold end corresponds to a liquid hydrogen tank at 28 K and 6 bar absolute pressure. The pipe is filled with gaseous para-hydrogen and the wall consists of stainless steel.

The geometrical parameters of the pipe are varied as follow:

inner diameter
 D = 8, 12 mm
wall thickness
 s = 0.4, 0.8 mm
pipe length
 L = 500, 1000 mm
orientation
 α = - 90° ... + 90°

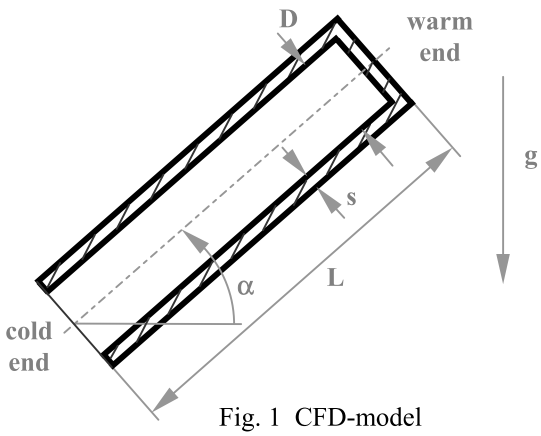

Fig. 1 CFD-model

A negative orientation means that the warm end is below the cold end. An orientation of 0° stands for a horizontal pipe. The inclination is varied in 15° steps for the short pipe and in 45° steps for the long pipe. The simulations are made with the commercially available software ANSYS-CFX v5.6. All calculations are steady state, 3-dimensional and use a combined turbulence model of k-ε and k-ω.

The physical properties of para-hydrogen are calculated with GASPAK v3.20. Because of the small pressure range within the pipe the properties are specified at constant pressure as a function of temperature. For the solid wall the properties are taken from CRYOCOMP v3.01 as a function of temperature.

MODEL VERIFICATION

To verify the abilities of the CFD-software for solid and fluid heat transfer a simple one-dimensional problem is modeled. A vertical pipe with the warm end up is simulated by CFD as well as calculated analytically. For the pipe aligned vertically a stable temperature stratification is expected. The heat is transported exclusively by axial heat conduction in gas and solid. Radial temperature gradients will be neglected. The axial temperature profile can be evaluated in a one-dimensional model assuming a constant axial heat flux $\dot{Q}$.

The pipe is discretized in axial direction and the local temperatures are calculated from

$$T_{x+dx} = T_x + \frac{\dot{Q} \cdot dx}{\left(A_{solid} + A_{fluid}\right) \cdot \lambda_{eff}}$$

with an effective heat conduction coefficient of

$$\lambda_{eff} = \frac{\lambda_{solid} \cdot A_{solid} + \lambda_{fluid} \cdot A_{fluid}}{A_{solid} + A_{fluid}}.$$

The local temperatures are then calculated iteratively by varying the axial heat flux until the warm end temperature has reached its given value.

In comparison with the CFD-simulation the axial temperature distributions are almost identical. Near the cold end there is a slight temperature difference as an effect from different temperature dependencies of the thermal conductivities of wall and fluid. There is a small radial temperature gradient in the simulation giving the divergence between simulation and analytical solution. For this reason the heat fluxes are taken from the warm side. The comparison will show a difference smaller than 1 % between analytical and CFD-solution:

	analytical	CFD	
heat flux (warm end)	0.7582	0.7512	W
difference		- 0.93	%

CFD-SIMULATIONS

A general converging problem is the relatively small cross area of the wall compared to the pipe length. For the axial heat transport it takes a long real time until a stable temperature profile is formed in the wall and later in the fluid. This has a strong influence on the overall computing time. On one hand a very small timestep is necessary for an accurate solution of the flow field, on the other hand a long timestep is important for the temperature field. To reduce the computing time a good start condition is very important. At negative orientations the solution from - 45°, once calculated with appreciable processing time, is used to start. The analytical solution for the vertical pipe is used for positive orientations instead.

The extrema + 90° and - 90° converge very badly. At + 90° a nearly stable temperature stratification exists. The only stimulation for a flow comes from different thermal conductivities of wall and fluid respectively. The flow velocities are nearly zero. For - 90° we find a fully convective flow. But there is no stable solution because of the symmetries in the pipes. The software can not solve the problem exactly. In the outer gas region a heated flow heading upwards and in the center a downward straight flow would be expected. The stable solution for - 90° could possibly being forced by an appropriate starting condition.

For all other orientations a stable converged solution could be found.

RESULTS

To post-process the results, an averaged heat flux is calculated from the heat fluxes through the warm and the cold end. In the chart (Fig. 2) the results are shown for different geometric parameters. Beside the heat fluxes through the standard pipe the heat fluxes through deviating geometries are plotted in extra curves.

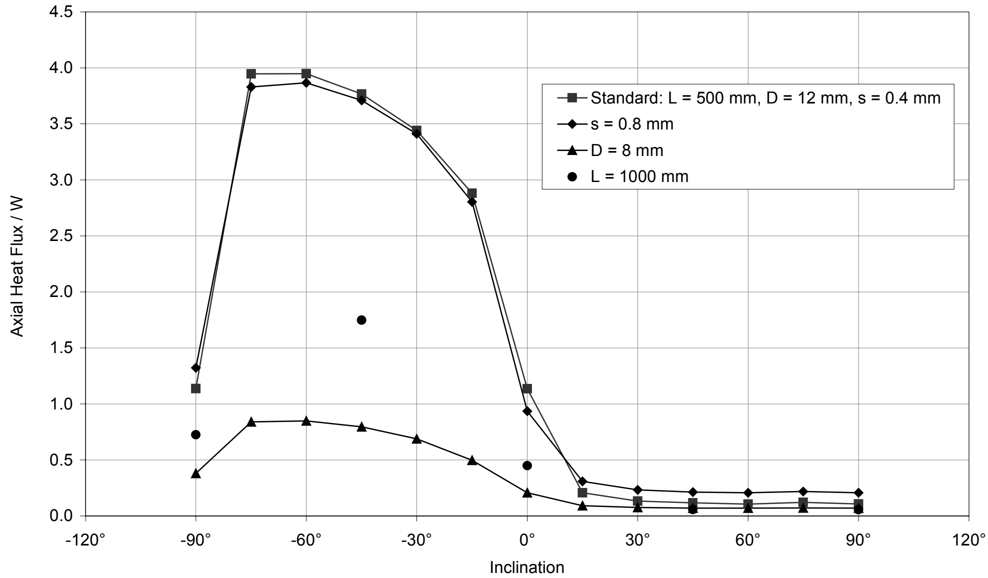

Fig. 2 Heat fluxes through pipes at different orientations

As an overall view all variants have a similar character with different absolute values. At positive inclinations the heat flux is small. Only heat conduction through solid and gas occur. With decreasing inclinations lower than + 30° the heat flux increases rapidly. A convective flow starts within the pipe and increases the heat flux through the gas region. The maximum heat flux occurs around - 75°. At lower inclinations no exact values are known because no stable solution could be found.

As a rough estimation three regions can be defined:

I.	- 90° ... 0°	free convection
II.	0° ... + 30°	free convection / heat conduction
III.	+ 30° ... + 90°	heat conduction

Some detailed local flow and temperature behavior are given in the following pictures. It can be shown (Fig. 3), that at lower inclination angles after start of the convection the temperature differences across the pipe will increase.

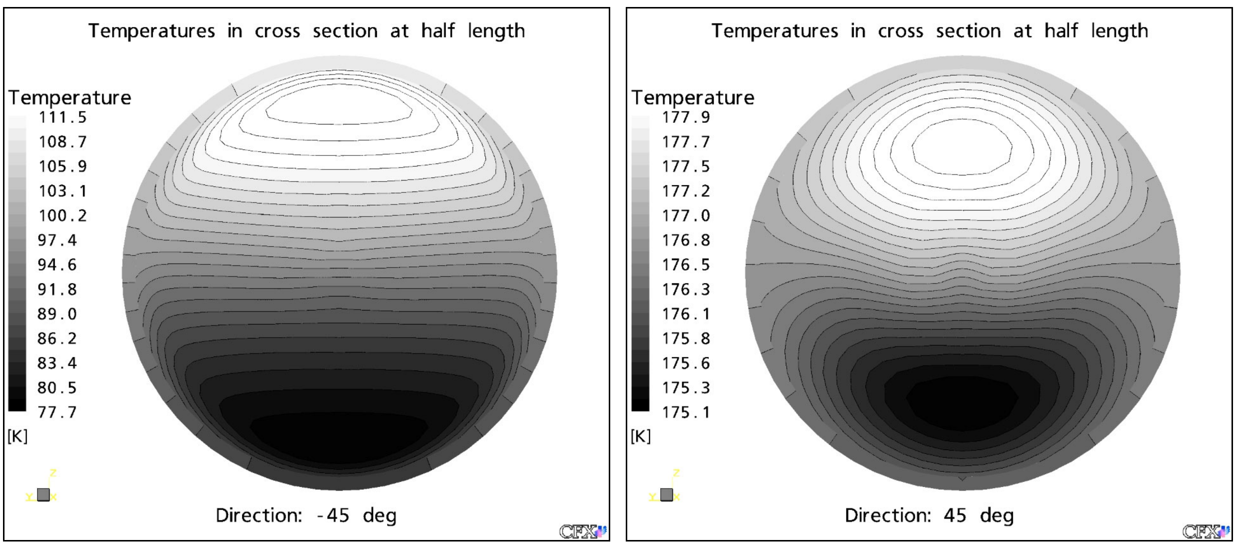

Fig. 3 Temperatures in cross section at half length (note: different scales)

In the upper half of the pipe a flow will occur from the hot to the cold end (Fig. 4). In the pictures the warm end is on the right side. A second flow from cold to hot end will be in the lower half of the pipe. These flows are responsible for the increase in heat transport. At positive orientations they are not fully developed. There are mainly local eddies negligible to the overall heat transport.

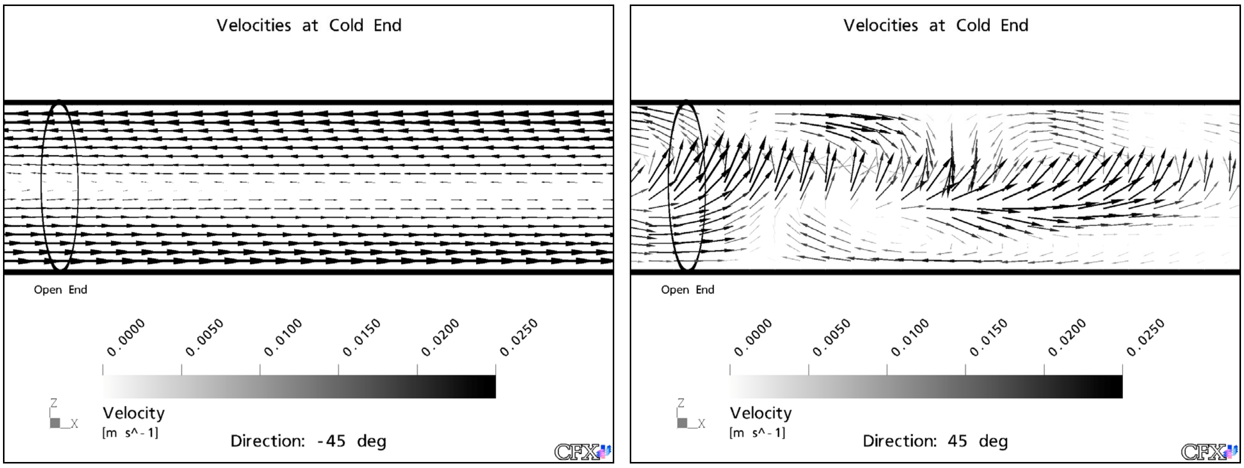

Fig. 4 Velocity vectors at cold end

Close to the warm end the gas is heated and ascends (Fig. 5). For positive inclinations the velocities are one order of magnitude smaller than at negative inclinations.

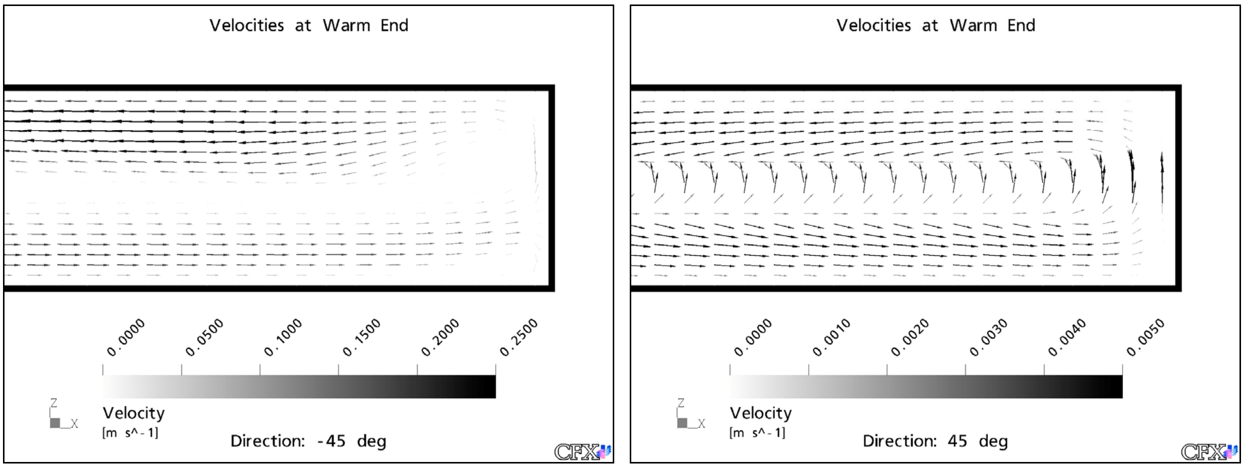

Fig. 5 Velocity vectors at warm end

CONCLUSIONS

The following dependencies are found by the CFD-simulations:

- The heat flux through pipes by convection compared to conduction can be up to 30 times higher.
- The doubling of the pipe length will reduce the heat flux by 50 %.
- A doubling of the wall thickness increases the heat flux for positive inclinations. The cross section for heat conduction becomes higher.
- A doubling of the gas cross section gives a more than 4 times higher heat flux if convection occurs.

As a result from these CFD-simulations more detailed hints for construction can be deduced. Usual design rules (as long as possible, minimum wall thickness, low heat conductivity...) can be optimized more precisely. Moreover the influence of the inclination can be taken into account.

As an option for future work the results could be cumulated into simple equations. Then the heat transport through complex pipe systems can be easily estimated with approximations of bent pipes by straight pipes.