INFLUENCE OF VAPOR FEED DESIGN ON THE FLOW DISTRIBUTION

M. Wehrli *, S. Hirschberg, R. Schweizer****

* Sulzer Chemtech AG, Postfach, CH-8404 Winterthur, Switzerland
** Sulzer Innotec AG, Postfach, CH-8401 Winterthur, Switzerland

ABSTRACT

Computational fluid dynamics (CFD) has been applied to investigate the flow field which develops above the vapor feed in packed separation columns. The uniformity of the flow below the packing is assessed by means of the coefficient of variation K_p. The resulting K_p for the standard inlet is input into an empirical correlation depending on the three most important, basically geometrical, dimensionless parameters. Other feed systems including the Schoepentoeter and the vapor horn are compared with the standard inlet.

INTRODUCTION

Optimum operation of packed separation columns requires even distribution of liquid films and gas flow. While the role of proper liquid distribution was never disputed the gas initial distribution has been neglected for a long time. Its importance becomes evident as large column diameters and packings with lower pressure drop are considered. The purpose of the vapor feed system is to introduce process gas or vapor, coming for example from the reboiler, into the column and distribute it evenly over the whole cross section. To achieve this, the vapor velocity needs to be reduced over a short distance. At the same time, the inlet should not unduly block the column cross section or lead to excessive pressure drops. For economical reasons, a minimum distance between the nozzle and the packing is desired.

Computational fluid dynamics (CFD) has become a widely accepted design tool, and its capability to solve process engineering problems is now being realized [1, 2]. While major challenges within this field, like the simulation of multiphase flows, are still awaiting better understanding and models of general applicability, many problems of lower complexity can now be addressed by CFD. Among them the vapor feed flow into columns is a good example. CFD offers the opportunity to complete valuable experimental and empirical research, e.g. [3-6], by more rigorous but less expensive numerical experiments. First numerical investigations related to gas inlets date back to the early nineties [7] and this subject has also been addressed a few times since [1, 8].

This paper presents CFD simulations which have been performed to assess the quality of vapor velocity profiles at the packing entry. A correlation for the coefficient of variation for simple nozzle inlets (standard inlet) is established. Its predictions are then compared with coefficients of variation calculated for more sophisticated systems.

CFD MODEL AND VALIDATION

The goal of the CFD simulation is to determine the uniformity of the vapor velocity profile right under the lower packing edge. Therefore the study will concentrate on the open space between the entry and the packing. The packing has to be included in the model because it influences the flow below it. It is modeled as a porous body with a resistance factor f_p leading to a pressure drop in agreement with design data for the liquid load of interest:

$$\frac{\Delta p}{\Delta h} = f_p \frac{\rho_g \overline{w}^2}{2} \qquad (1)$$

The influence of the liquid on the vapor velocity distribution in the open space is small and is therefore negligible. The vapor flow is assumed to be incompressible. Temperature variations are neglected. The effect of turbulence is taken into account using the standard k-ε turbulence model. The boundaries and boundary conditions in the model used are described in Tab. 1 and illustrated in Fig. 1.

Tab. 1: Boundaries and boundary conditions. The colors refer to the surfaces in Fig. 1.

Boundary	**Position**	**Boundary Condition (BC)**
Vapor inlet (green)	Cross section trough nozzle, some diameters upstream	Uniform velocity profile, typical turbulence intensity and length scale
Vapor outlet (yellow)	Cross section through the column, some space above the packing bed	Free outlet
Sump (blue)	Liquid surface considered flat	Symmetry, no shear
Walls (white)	Column wall, nozzle wall	Adiabatic for mass and energy, log-law for turbulence

The inlet and column geometies are in many cases symmetrical. This symmetry could be exploited in reducing the size of the numerical model by one half. It has however been observed that flow fields with and without a symmetry condition do not always agree. While the geometry is symmetrical, there is no guarantee that the flow field keeps this symmetry, especially in high Reynolds number flows. Steady asymmetric mean flows or oscillating flows may evolve. This seems to happen mainly when the inlet nozzle diameter is large and the jet momentum of the entering flow is relatively low. Comparisons between CFD simulations with and without symmetry condition applied and LDA (Laser Doppler Anemometry) measurements in an

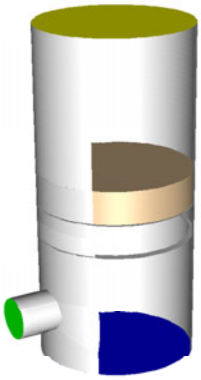


Fig. 1: Computational domain and boundary conditions.

air/water test rig (1m diameter) showed better agreement for the CFD results without the symmetry condition. To avoid oscillations in the results and tedious data collection over a long simulation time, a minor deviation from symmetry is introduced into the CFD model by shifting the nozzle off the symmetry plane by less than one percent of the column diameter.

Typical computational grids used consist of some hundred thousand up to 1.5 million finite volume cells, depending on the resolution required to capture geometrical details of the feed system. Simple geometries can be nicely modeled by structured multiblock grids. Grids are more difficult to generate for complex inlet devices. They can be handled by automatic grid generators using unstructured tetra grids. Commercial CFD software[1] based on the finite volume approach was used for the simulations. Vapor feed systems studied are shown in Fig. 2.

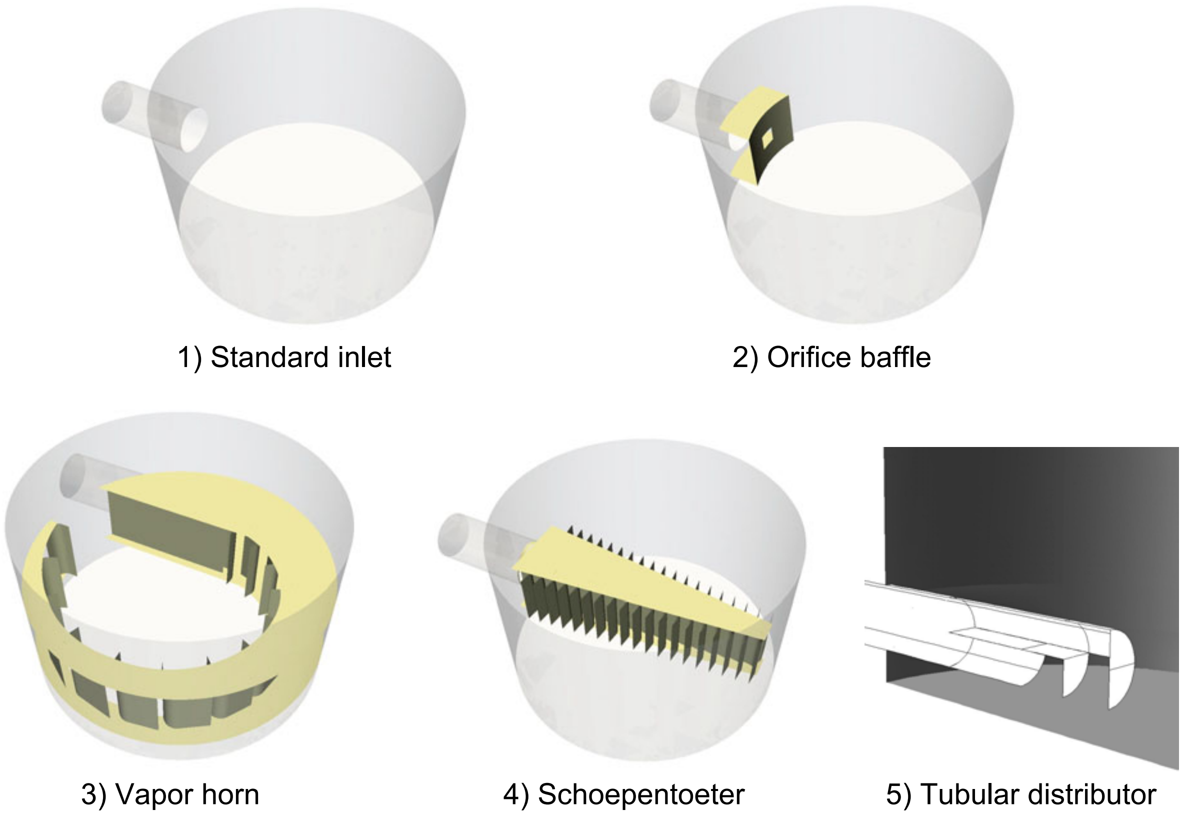


Fig. 2: Vapor feed systems considered. The tubular vapor distributor (5) is cut through the symmetry plane. It divides the vapor stream into two equal parts, which escape towards the sump.

---

[1] CFX-TASCflow from AEA Technology and STAR-CD from Computational Dynamics. The meshes are generated using ICEM Hexa or Tetra.

RESULTS FOR THE STANDARD INLET

The coefficient of variation is used to quantify the uniformity of the vapor velocity profile. It is evaluated on a horizontal plane through the column as follows ($\overline{w}$ is the average speed computed on the same plane):

$$K = \frac{\sigma}{\overline{w}} = \frac{1}{\overline{w}} \sqrt{\frac{\int (w - \overline{w})^2 \, da}{\int da}} \qquad (2)$$

This coefficient does not take into account the spatial scale of variation, which is certainly a disadvantage. The maldistribution index MI [9] seems more appropriate but tends to be sensitive to the resolution of the statistical analysis. The value of K is high at the elevation of the vapor feed. It approaches *zero* as the flow becomes more regular. A uniform vapor flow should be achieved at the packing entry. Therefore the coefficient value K_p below the packing is used to assess different vapor feed systems. The dependency of K_p on the following geometrical and physical parameters should be evaluated for each type of inlet device: Diameter of the column D, distance H between the nozzle and the packing, distance H_s between the nozzle and the sump, packing resistance f_p, the F-factor in the column and in the nozzle (F_n). The latter is related to F by the nozzle diameter d and the column diameter D. Based on dimensional considerations and confirmed by CFD results the system reduces to four parameters shown in Tab. 2.

Tab. 2: Dimensionless parameters.

Definition	Description
$\mathbf{H} = H / D$	Normalized clearance nozzle – packing
$\mathbf{S} = H_s / D$	Normalized clearance nozzle – sump
$\mathbf{F} = F_n / F$	F-factor ratio
$\mathbf{P} = D \, f_p$	Normalized packing resistance

The parameters $\mathbf{F}$ and $\mathbf{P}$ can also be expressed in terms of geometrical data like diameters and specific surface area of the packing. Because it was considered to be a factor of minor importance, the influence of $\mathbf{S}$ has not been studied yet. For similar flow patterns, an empirical correlation can be formulated:

$$K_p = q \; \mathbf{P}^a \, \mathbf{F}^b \, \mathbf{H}^c \qquad (3)$$

Based on an incomplete factorial design, the following coefficients have been determined for the standard inlet as shown in Fig. 2(1): a= -0.5, b=1.3 and c= -1.3. The CFD simulations and model predictions agree within 20% error (black symbols and thin lines in Fig. 3). Two configurations do not match the data well (white symbols) and have not been included into the correlation. They are cases, where the diameter of the vapor inlet nozzle is large with respect to the column diameter leading to an inlet stream with relatively low momentum (**F**<20). The resulting flow patterns differ strongly from cases with larger **F** and the assumption of similar flow fields does not hold any more.

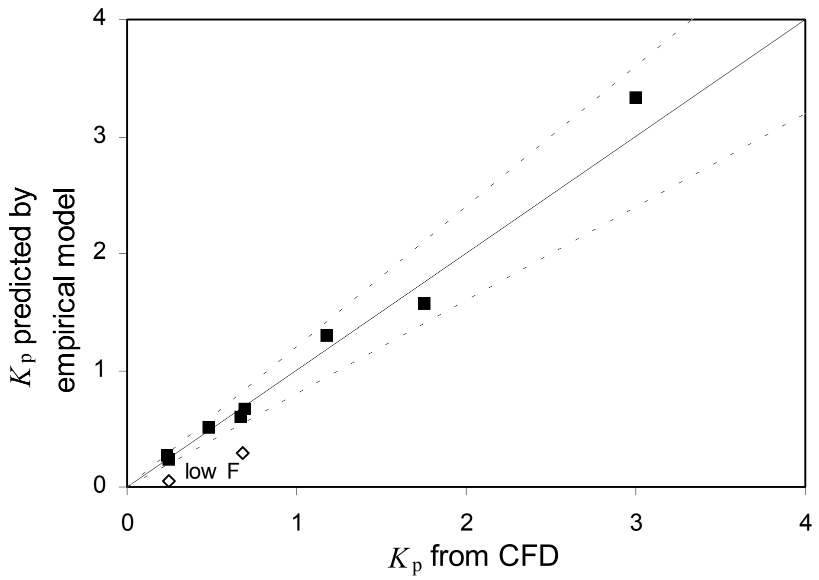


Fig. 3: Parity plot for K_p for a standard inlet (black symbols). The predictions are generated by equation (3).

COMPARISON OF FEED SYSTEMS

The following feed systems have been analyzed by means of CFD (consider Fig. 2 for illustration):

1) Standard inlet.
2) Orifice baffle: A specially designed channel baffle with lateral openings and a central orifice [10].
3) Tangential inlet device, a vapor horn with special guide vanes. This device is usually applied with flashing streams or high velocity vapor feeds carrying a large disperse liquid fraction. The velocity used in this study is rather small.
4) Vapor distributor similar to the Schoepentoeter.
5) Tubular vapor distributor with two openings oriented towards the sump.

For the first two systems a study by parameter variation has been carried out. Only a small number of results are available for systems 3 to 5. This is not sufficient to allow general conclusions regarding their performance. The resulting K_p data are plotted in Fig. 4. Results of CFD simulations (abscissa) and the correlation (3) for the standard inlet applying identical parameters $\mathbf{P}$, $\mathbf{F}$ and $\mathbf{H}$ (y-axis) are combined in this parity plot. Points appearing above the diagonal are indicative of a better performance than a standard inlet. The following conclusions can be drawn:

- With the orifice baffle a reduction in K_p by more than 50% is possible. Under flow conditions that are already favorable for the standard inlet (lower left corner in the diagram) the orifice baffle can however be disadvantageous. The results are valid for an optimized design based on CFD simulations, which are mandatory to obtain best performance of the orifice baffle.
- Preliminary results suggest that the tubular distributor is not better than the standard inlet. This system directs the vapor towards the sump. The configurations considered so far leave a clearance of only 1.5 nozzle diameters between the sump and the nozzle. Larger distances might improve the performance. For

this kind of distributor, the influence of **S** as well as the optimal tube geometry need further investigations.

- The four red symbols in Fig. 4 represent the values of different feed systems all simulated within the same column (see next section). Because the system parameters are equal for all four cases, the model equation (3) predicts the same coefficient of variation, and the points appear on one horizontal line. The vapor horn performs best, followed by the Schoepentoeter and the orifice baffle. For the standard inlet, the current CFD simulation predicts a slightly lower K_p than previous ones. Reasons might be the fact that a tetrahedral instead of a multiblock mesh and a different solver have been used.

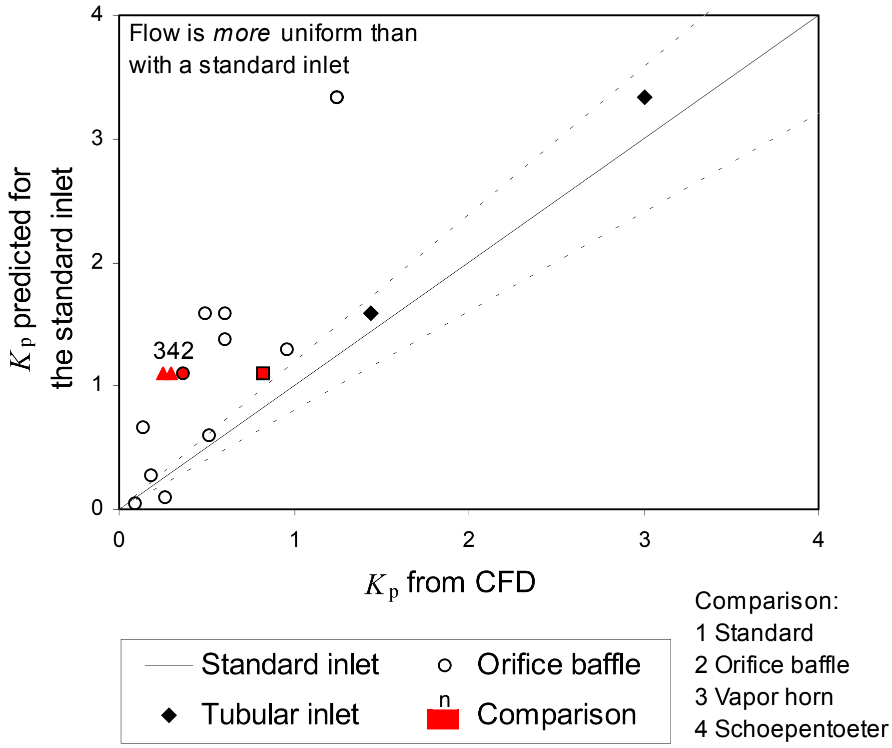


Fig. 4: Parity plot for K_p data of various feed systems. The position in x represents the K_p achievable according to CFD, the vertical distance indicates values expected for a standard inlet with the same parameters according to equation (3). The graph compares the performance of various inlet systems (x-axis) with expected results for a standard inlet (y-axis). A symbol above the diagonal indicates a better performance than a standard inlet, because the corresponding system generates a flow with a lower K_p value.

FLOW DETAILS

The flow field in a D=3.2m column equipped with Sulzer MellapakPlus 252.Y structured packing is simulated using a commercial CFD package. Three different feed systems, the standard inlet (1), the orifice baffle (2) and the vapor horn (3) are considered. The following parameter values apply: **H**=0.38, **S**=0.34, **F**=28, **P**=130. The corresponding coefficients of variation (numbered red symbols in Fig. 4) clearly indicate that the standard inlet performs worst, while the others improve the flow into the packing considerably. Contour plots (Fig. 5) of the vertical velocity component

100mm below the packing confirm these conclusions. The distribution of the standard inlet is still characterized by peaks above 1m/s and even some limited regions with downward motion. On the other hand, no values above 0.6m/s are found in the much more uniform flow which is induced by the vapor horn. The flow field in cases 1 and 2 is not symmetric. As discussed above, symmetry of the geometry does not necessarily lead to symmetrical flow fields. A slight displacement of the inlet nozzle from the central plane within the range of design tolerances was intentionally introduced to trigger potential asymmetrical flow developments.

Streamlines in Fig. 6 illustrate the complexity of the three-dimensional flow that develops in the open space between the sump and the packing. In case 1 (standard inlet) the vapor jet crosses the column and impinges on the wall at the side opposite to the nozzle. From there, the flow spreads along the walls and the part of the packing closest to the point of impingement gets most of the vapor flow. If the orifice baffle is introduced (case 2), the vapor jet is divided in two wall jets and a central jet. If properly designed, the wall jets meet at the opposite side of the column and are directed towards the center of the column where they meet the orifice jet. The flow generated by the vapor horn (case 3) is characterized by a swirl forcing the stream to ascend with a high secondary velocity component. It seems that the swirl and the induced increase in flow path length helps equalize the flow field. An important feature of the swirling flow is the low vertical velocity in the center (see Fig. 5). This agrees with earlier findings [5, 3, 11]; the simulations confirm also that a better vapor distribution can be achieved with the tangential inlet device.

SUMMARY

The coefficient of variation is used to assess the uniformity of the flow induced by a vapor feed into a separation column. So far, we have been able to input the coefficient into an empirical correlation, taking into account the three most important dimensionless parameters for such vapor feed inlets. While only a correlation for the standard inlet was presented, similar correlations can also be derived for other, more complex inlet constructions. By comparing the coefficient of other feed systems with the standard inlet, we may conclude that a very simple device, namely the orifice baffle, is very effective, provided it is properly designed. Given their technical design, it is not surprising that the Schoepentoeter and the vapor horn perform even better. One important feature of these two devices has not been considered in the paper, namely their high capability to disengage dispersed liquid, unequalled by the orifice baffle.

A criterion for how much unevenness of the vapor flow is tolerable for a specific column is still not available today and will have to be addressed in future work.

LITERATURE

1. M. Casey, E. Lang, R. Mack, R. Schlegel, M. Wehrli (1998), Speedup, 12, 43-51.
2. M. Fleischli, R. Schlegel, M. Wehrli (2001), International Symposium on Mixing in Industrial Processes ISMIP4, Toulouse, May 14-16, France.
3. L. A. Muir, C. L. Briens (1986), The Canadian J. Chem. Eng., 64, 1027-1032.
4. F. Moore, F. Rukovena (1987), Chemical Plants & Processing, August, 11-15.
5. L. Fan, G. Chen, S. Constanzo, A. Lee (1997), ICHEME Symposium Series 142, 899-910.
6. X. Yuan, W. Li (1997), ICHEME Symposium Series 142, 931-938.
7. Ph. Süess (1992), ICHEME Symposium Series 128, A369-383.
8. S. Vaidyanathan, M. Wehrli, H. Ulrich, R. Hunkeler (2001), Distillation Symposium, AIChE Spring National Meeting, April 24, Houston, Texas.
9. A. Mohamed Ali, P. J. Jansens, Z. Olujic (2001), Proceedings of the 6th World Congress on Chemical Engineering, Melbourne, 23-28 September, Australia.
10. F. Moser (2002), U. S. Pat., No. 6,341,765 (Jan. 29).
11. H. Z. Kister (1990), Distillation – Operation, McGraw-Hill, pp. 27.

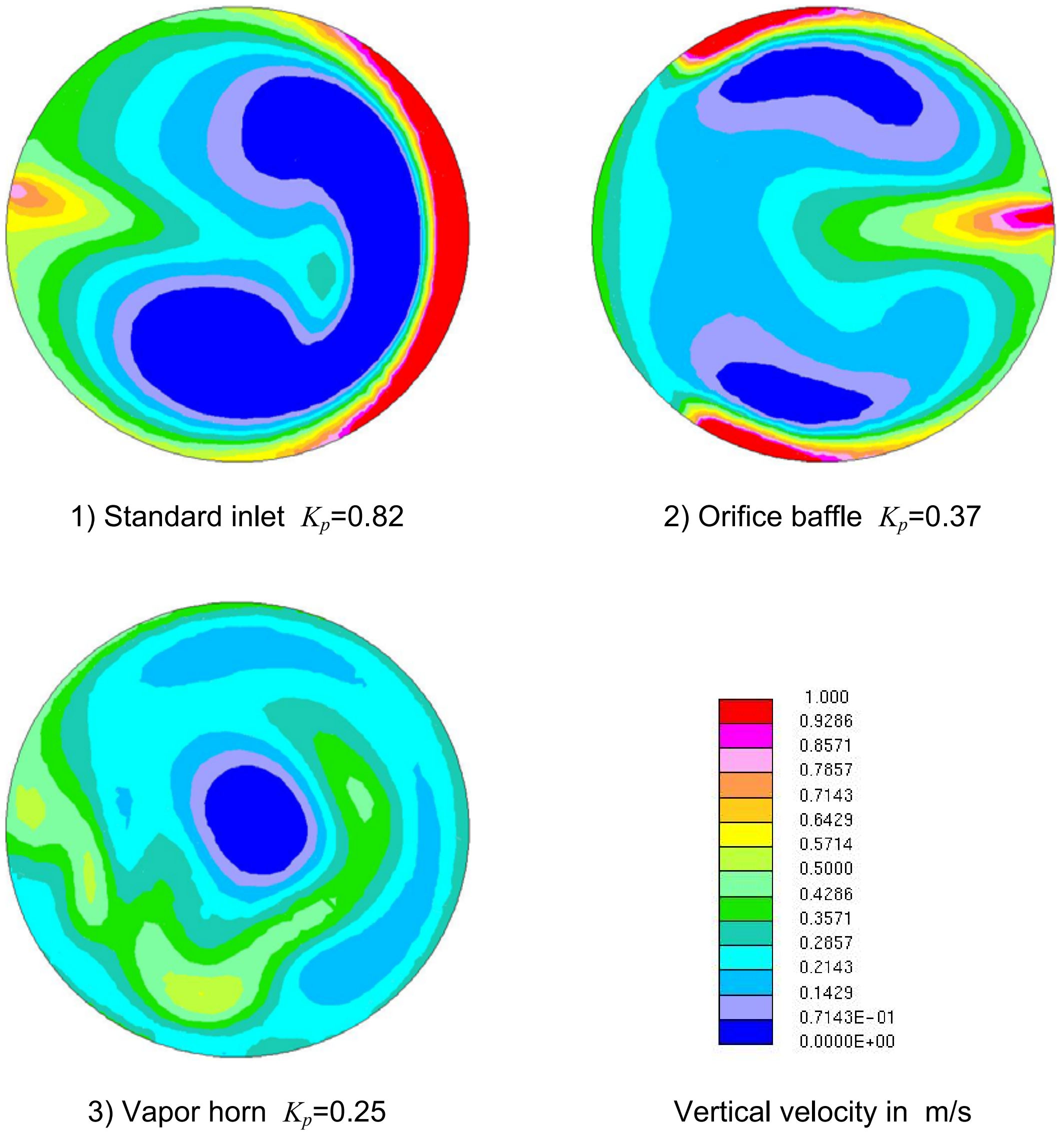
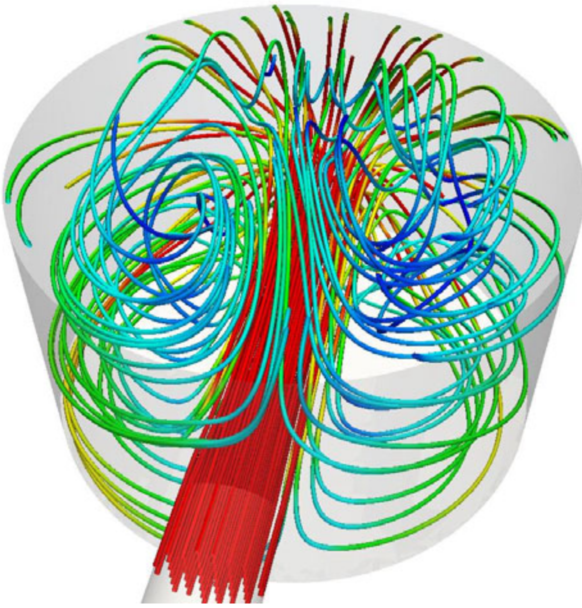
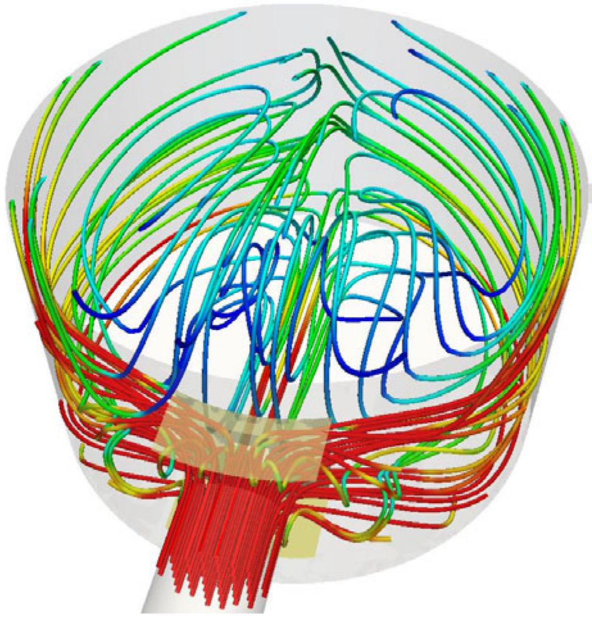


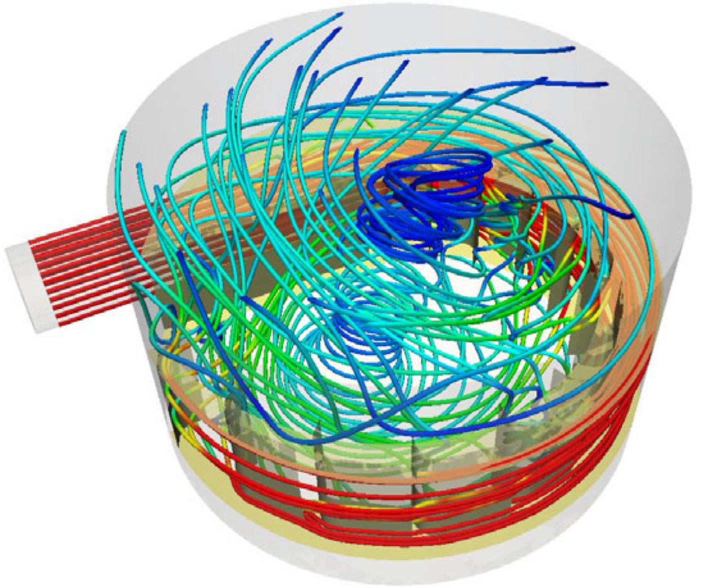
Fig. 5: Distribution of the vertical velocity on horizontal planes, which are placed 100mm below the packing. Four different vapor feed systems. The color scale is clipped at 0m/s and 1m/s. The vapor enters the column on the left hand side.



1) Standard inlet



2) Orifice baffle



3) Vapor horn

Fig. 6: Streamlines colored with the local speed. Color scale: Red is the highest velocity (4m/s and more), blue the lowest (0m/s).