SIMULATION AND OPTIMISATION OF A WIPING KNIFE ON THE LAB SCALE

Jikang Wang – Brandenburgische Technische Universität, Cottbus, Germany
Tobias Klinkberg – Salzgitter Mannesmann Forschung, Salzgitter, Germany
Matti Teschner – Salzgitter Mannesmann Forschung, Salzgitter, Germany
Dominik Weirich – KHS, Bad Kreuznach, Germany
Thomas Koll – Salzgitter Mannesmann Forschung, Salzgitter, Germany

ABSTRACT

In the present study Computational Fluid Dynamics (CFD) were used to design a new air-knife for the hot-dip galvanizing simulator of Salzgitter Mannesmann Forschung in order to improve the flow field inside the knife and at the nozzle exit. The overall objectives were to improve the surface quality of coated samples and enhance coating control capabilities compared to the existing air-knife by optimising the gas flow inside the knifes. The existing configuration yields coating thicknesses from 7 µm on, although coating homogeneity could be optimised. The flow field inside the nozzle was modelled by transient RANS simulation.
An ideal air-knife should create a uniform gas flow with low turbulence at the nozzle exit. However, this is difficult to achieve, being affected by various factors. It could be shown that the existing air-knife bears some significant fluidic disadvantages at the gas inlet and distribution inside the nozzle, causing an inhomogeneous, unsteady flow.
A new air-knife design with optimised inlet and an innovative shape for rectified fluid deflection and distribution was designed. The additional application of a turbulence manipulator and an improved feeding geometry of the exit channel finally lead to a much more uniform gas flow. The turbulence of the flow is reduced significantly by the new design, compared to the existing nozzle

KEYWORDS

Gas jet wiping, air-knife design, Computational Fluid Dynamics, hot-dip galvanizing simulator, fluidic optimisation of gas flow.

INTRODUCTION

In the continuous hot-dip galvanizing process, a moving, heat treated steel strip is continuously passed through a molten zinc bath and dragging up molten zinc adhered on the strip surface. Because the thickness of the zinc film is usually much thicker and much more inhomogeneous than requested, a pair of two high speed opposing plane gas jets are located above the bath to adjust the film thickness. The impinging gas jets remove any excess molten zinc from the strip surface using the combined forces of pressure gradient, wall shear stress and gravity, which cause the excess zinc to flow back into the bath. The remaining coating thickness is reduced to an order of few micrometers by this process, referred to as gas jet wiping, schematically shown in Fig. 1. Usually nitrogen or compressed air is utilized for the gas jet, the nozzle of which is commonly called air-knife. Today the gas jet wiping method, which has replaced the squeeze roll system about 50 years ago, is widely used in commercial plants because of easy process control and good productivity. [1,2,3,4]
Galvanizing simulators are used for years to research and simulate the galvanizing process of steel sheet on a lab scale. Just like on industrial lines air-knives are applied to wipe down the unnecessary molten zinc after the steel sample is dipped in the liquid zinc. Fig. 2 shows the galvanizing simulator of Salzgitter Mannesmann Forschung (SZMF).

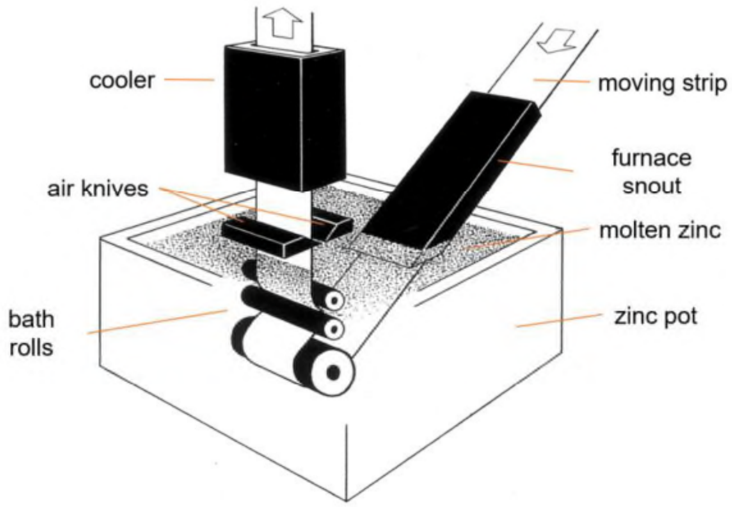

Fig. 1: Schematic view of galvanizing process.

Fig. 2: Galvanizing simulator of SZMF.

Many studies have been done investigating the influence of process parameters e.g. nozzle slot opening, standoff distance, nozzle pressure, strip velocity and liquid properties of the molten zinc on the wiping effect. Various steady state models are proposed for predicting coating thickness based on Navier-Stokes equations considering inter alia wall pressure profile, shear stress effect, and surface tension of the zinc film. Computational fluid dynamics (CFD) simulations are used by now as a standard. [1,2,5,6,7,8]

Another research focus is the dynamic flow behaviour of the gas jet impinging on the liquid zinc film, which is highly complex. The gas jet is strongly affected by the nozzle design, hence it should be constructed prudentially. Unsteady CFD simulations show that the gas flow leaving the nozzle is not stable, inducing oscillating forces on the film and possible coupling between fluctuating jet and liquid flow. Nonuniformities of the coating, such as waviness and/or check marks, are caused by this phenomenon. Minimizing variations of gas jet should therefore lead to a more homogeneous zinc coating. [4,9,10,11]

The existing air-knife configuration of the galvanizing simulator is used for years, designed solidly, enabling to generate samples with coating thicknesses from 7 µm on, but coating appearance and homogeneity could be optimised, see Fig. 3. The zinc coating shows a horizontal and vertical oriented wavy pattern with inhomogeneous coating thickness of about ± 20 %, see Fig. 4. Several tests identified pressure fluctuations in the chamber of the simulator due to a poor adjusted exhaust valve to be responsible for the horizontal one and could be eliminated by optimising the course of the background pressure in the chamber. Because no hint was found that the vertical structure depends on a similar process or equipment parameter, an inhomogeneous, unsteady gas jet induced by the air-knife design was assumed as the root-cause. The flow field inside the nozzle was therefore modelled by transient RANS simulation (Reynolds-averaged Navier-Stokes) to identify fluidic optimisation potential and design a new air-knife.

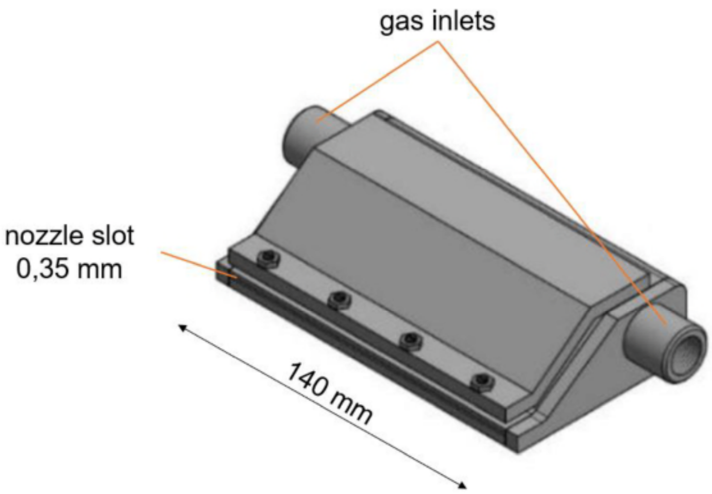

Fig. 3: Schematic view of existing air-knife.

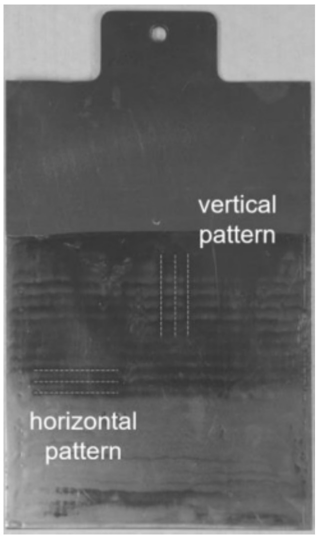

Fig. 4: Galvanized sample.

APPLICATION OF FLUIDIC PRINCIPLES

It is known that nozzle design strongly affects the flow behavior of the gas jet, for example velocity profile, turbulence and flow distribution across air-knife width. The gas flow around the nozzle, especially vorticity and turbulence between strip and outer nozzle shape and their interaction with the gas jet, is also influenced. Therefore designing an air-knife with a gas flow as steady as possible at the nozzle exit should consider at least following issues:

- The gas supply of the nozzle must ensure a uniform filling and gas distribution across air-knife width. Due to space restrictions the gas inlet is typically positioned on the side, which provokes on the one hand that the gas flow must be deflected by 90 degrees and on the other hand that volume flow decreases across the width. Fluidic measures are necessary to achieve homogeneous gas distribution and minimize cross flow and vortices. A tapered gas distribution chamber for example compensates the decreasing volume flow and leads to constant feeding flow rate across nozzle width. Manipulators like perforated plates could homogenize current by creating flow resistance on the other hand.
- The inside contour of the nozzle should be ideally designed in such manner, that uniformity of the deflected gas flow is maintained or at least improved, and turbulence is not generated. Therefore, components like screws or braces inside the nozzle should be reduced or avoided as far as possible, because they significantly disturb the gas flow. Moreover, should the inside contour have no or only few edges or small radii to prevent stalling, leading to turbulence and backflow.
- Reducing the cross-section in direction to the exit channel accelerates the gas flow almost to transonic speed. An as constant as possible increase of flow velocity without peaks in acceleration is considered as advantageous to prevent turbulence. This is achieved with a nonlinear convergent contour, ideally following a reciprocal function. The subsequent outlet channel affects the direction and the velocity profile of the gas further on. With increasing channel length velocity profile becomes approximately parabolic, but aerodynamic drag increases, impairing air-knife efficiency.
- A tapered outer nozzle shape shows advantages over obtuse form, the latter giving a more confined space between strip and outer nozzle contour leading to turbulence and higher vorticity in this area. Pointed nozzles with less steep angles reduce interactions between exit gas flow and the liquid film because they leave more space for the gas.

NUMERICAL MODELING

The aim of the numerical investigations is to improve the understanding of the flow conditions forming in the nozzle and to identify fluidic optimisation potential. Therefore various nozzle models were simulated by Ansys AIM. The following basic settings were selected for the CFD simulations.

Geometry and Meshing

The starting point of the CFD simulations used for the optimization is the inner volume of the existing nozzle shown in Fig. 3. Fig. 5 shows the meshing of the nozzle body and the areas close to the wall. Characteristic for the existing nozzle are the gas inlets on both sides and the geometrically simple shape of the nozzle body.

To optimise computing time, memory space and accuracy, a structured mesh with hexahedron-elements is used for most nozzle variants, especially in the nozzle exit area. Only the final nozzle variant with a more complex plenum shape use an unstructured mesh in the plenum area. Furthermore, 5 boundary layers are provided in the areas close to the wall. This results in numbers of elements between 550.000 and 750.000 for the different nozzles.

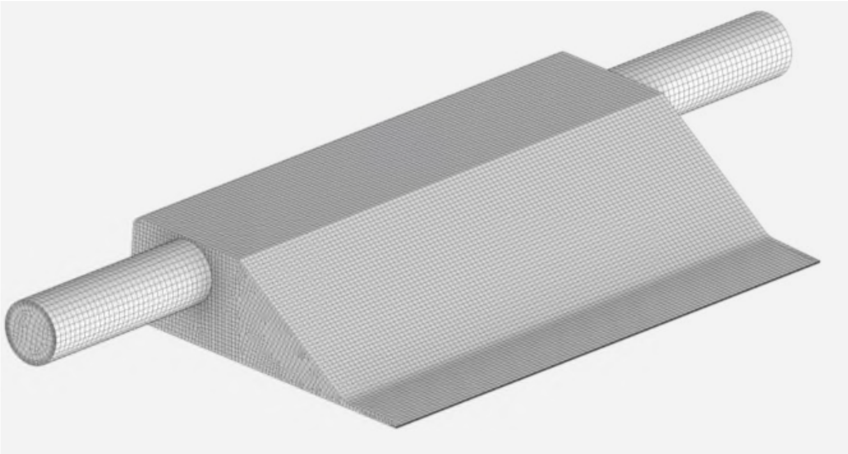

Fig. 5: Meshing of the existing nozzle.

Initial and boundary conditions

Due to the high velocities of 200 - 250 m/s at the nozzle exit and the compressibility of the gas that must be considered, a set value for the mass flow is used as an initial condition for the transient simulation. Thus, a mass flow is determined iteratively by stationary calculations, which corresponds to the required velocity at the air-knife using nitrogen at atmospheric pressure and a temperature of 295 Kelvin. To prevent dynamic influx effects, the mass inflow is exponentially increasing, such that the final flow value is reached after a defined period after which the mass flow remains constant. The boundary condition at the outlet is atmospheric pressure

Turbulence model

For the simulations the SST model (Shear Stress Model) is used, which combines the benefits of the k-ε model for areas away from walls and the k-ω model, yielding good results especially in the direct vicinity of walls during laminar-turbulent change.

The turbulence intensity calculated by solving the RANS equations (Reynolds-averaged Navier-Stokes) represents, besides the velocity distributions, an essential evaluation criterion for the quality of a flow and is used together with the velocity distributions for the comparison of the different development stages.

Evaluation method

To evaluate the flow condition inside the nozzle, in-house tools of the simulation software like streamline or velocity vector visualisation are used. Additionally, time dependent flow velocity in all three dimensions at the nozzle exit is analysed. As a criterion for turbulence and flow conditions the turbulence intensity Tu is used, see Eq. (1) and Eq. (2). This value is dimensionless and relates velocity fluctuations U‘ to average velocity U, small values indicating a less turbulent flow condition.

$$Tu = \frac{U'}{U} = \frac{\sqrt{\frac{1}{3}\left(u'^2_x + u'^2_y + u'^2_z\right)}}{\sqrt{u_x^2 + u_y^2 + u_z^2}} \qquad \text{Eq. (1)}$$

$$u'^2_{x,y,z} = \frac{1}{1-n}\sum_{i=1}^{n}\left(u_i - \bar{u}_i\right)^2 \qquad \text{Eq. (2)}$$

RESULTS AND DISCUSSIONS

The simulation result for the existing air-knife shows a highly turbulent flow field inside the nozzle and confirms the presumed fluidic disadvantages of this design. Due to gas inlets on both sides, the incoming flows collide, creating a chaotic flow condition with intense turbulence, Fig. 6. At the nozzle exit the flow is not uniform, but varies widely in any spatial direction, shown in Fig. 7. Because of the lateral gas feeding and missing measures to deflect the flow and to reduce eddies, the fluctuations in nozzle width direction (x-coordinate) are particularly strong and even change flow orientation. That is why turbulence intensity is biggest at the nozzle centre.

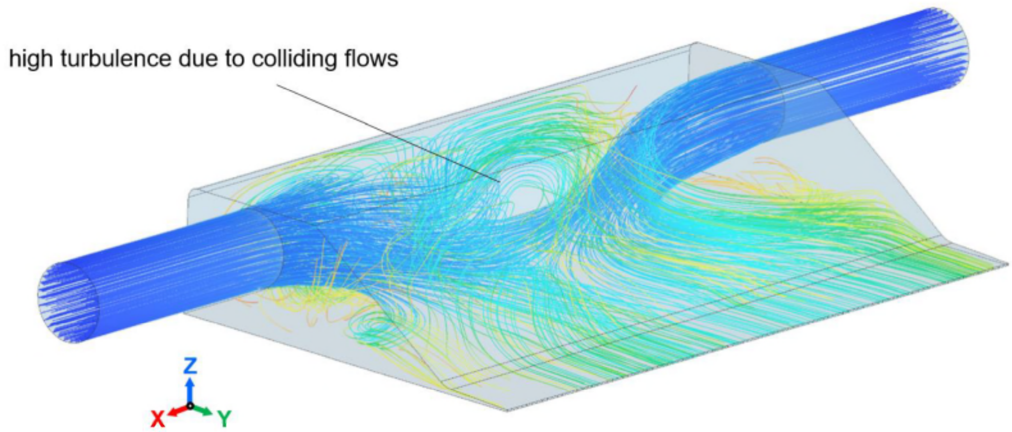

Fig. 6: Streamline visualisation of gas flow inside existing air-knife
(representative snapshot, colour gradient represents different runtime of flow).

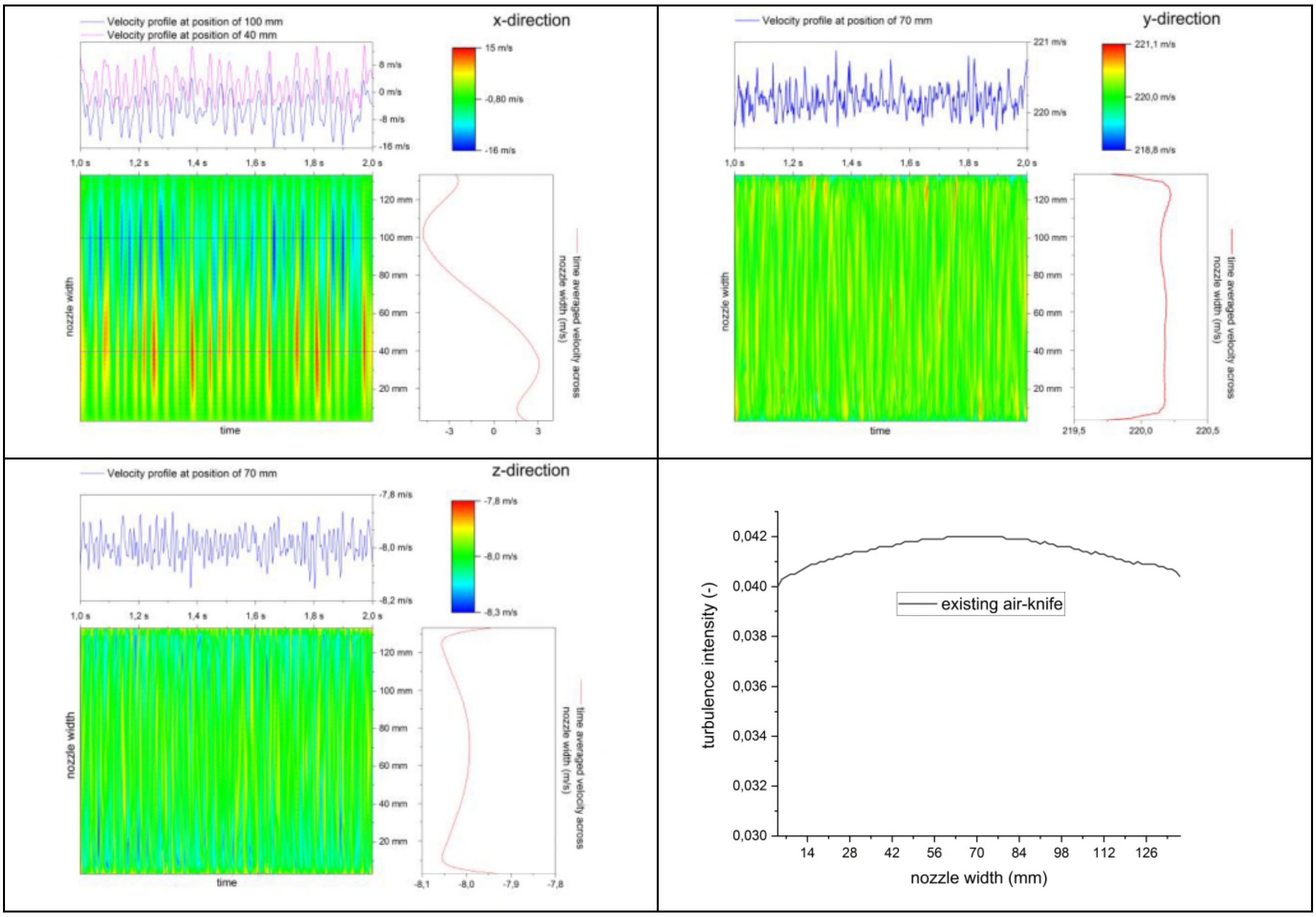

Fig. 7: Time dependent flow velocity (x-, y- and z-direction), turbulence intensity at the nozzle exit.

Based on earlier simulation results, a first optimised air-knife was designed, which is shown in Fig. 8 [12]. It has the following fluidic characteristics:

- Gas feeding of the air-knife is done now from one side, whereby the cross-section of the pipe exceeds the summed inlet cross-section of the existing nozzle more than twice. By this the flow velocity in the feeding line and at the gas inlet is decreased causing less turbulence and pressure drop.
- The cylindric feeding chamber has a narrow, elongated outlet slot creating flow resistance leading to better homogenization across nozzle width and deflection of the flow towards the nozzle exit. The cross-section ratio between in- and outlet of this "chamber" is approximately 2:1.
- After deflection of the stream, the cross-section of the channel increases to reduce flow velocity and pressure losses. A flow manipulator is additionally installed in the widened channel to decrease possible turbulences and cross flow.
- Afterwards the channel narrows to accelerate the gas flow towards the nozzle exit. To avoid acceleration peaks and reach constant speeding up of the gas flow, the cross-section decreases non-linearly after a hyperbolic function instead of a linear manner as the existing air-knife.

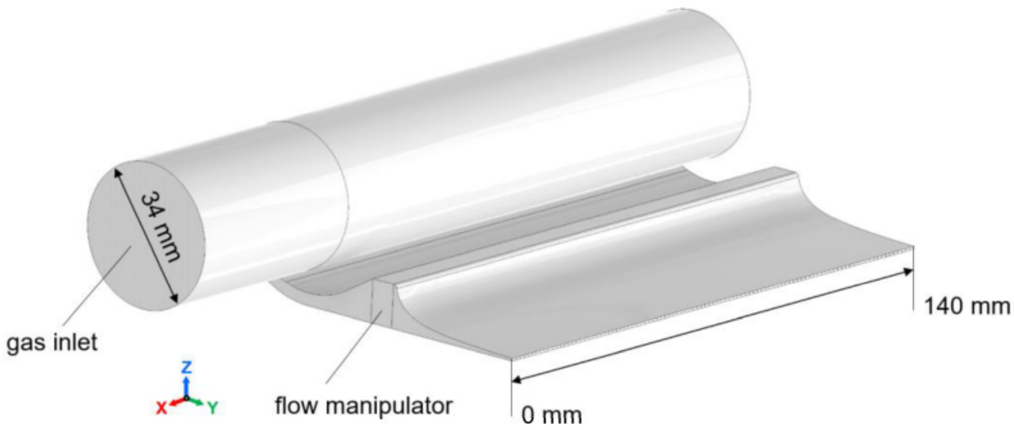

Fig. 8 Flow volume of optimised air-knife design.

Simulation results of the first optimisation design show clearly a much more stable gas flow inside the nozzle with less turbulence in comparison to the existing configuration, see Fig. 9 and Fig. 10. The flow at the nozzle exit is in general more constant, but imperfections can be observed on both external sides. On the feeding side the gas flow is deflected around a too sharp edge creating local turbulence and increased cross flow, see Fig. 11. On the other side flow fluctuations occur because of vortices at the end of the feeding chamber induced by vertical chamber boundary, although filling of the nozzle is better than for the existing air-knife.

Another fluidic irregularity can be seen at the inflow area of the manipulator. Because the channel widens quickly, stalling arises on the upper side, even creating a vortex and backflow, see Fig. 12. To avoid this effect, the gradient of the cross section in flow direction should be significantly decreased, by which either the duct length increases or the inlet or outlet channel height in this area need to be modified.

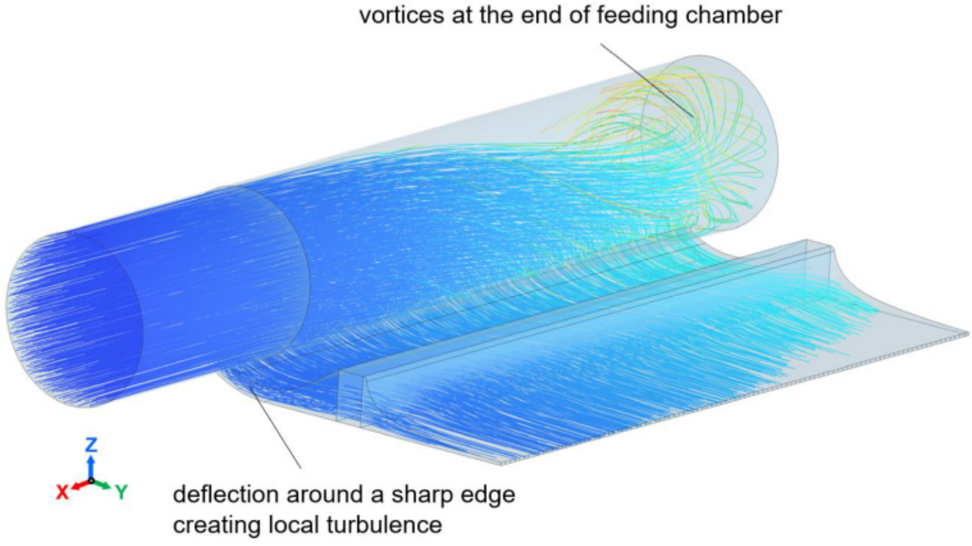

Fig. 9: Streamline visualisation of gas flow inside optimised air-knife
(representative snapshot, colour gradient represents different runtime of flow).

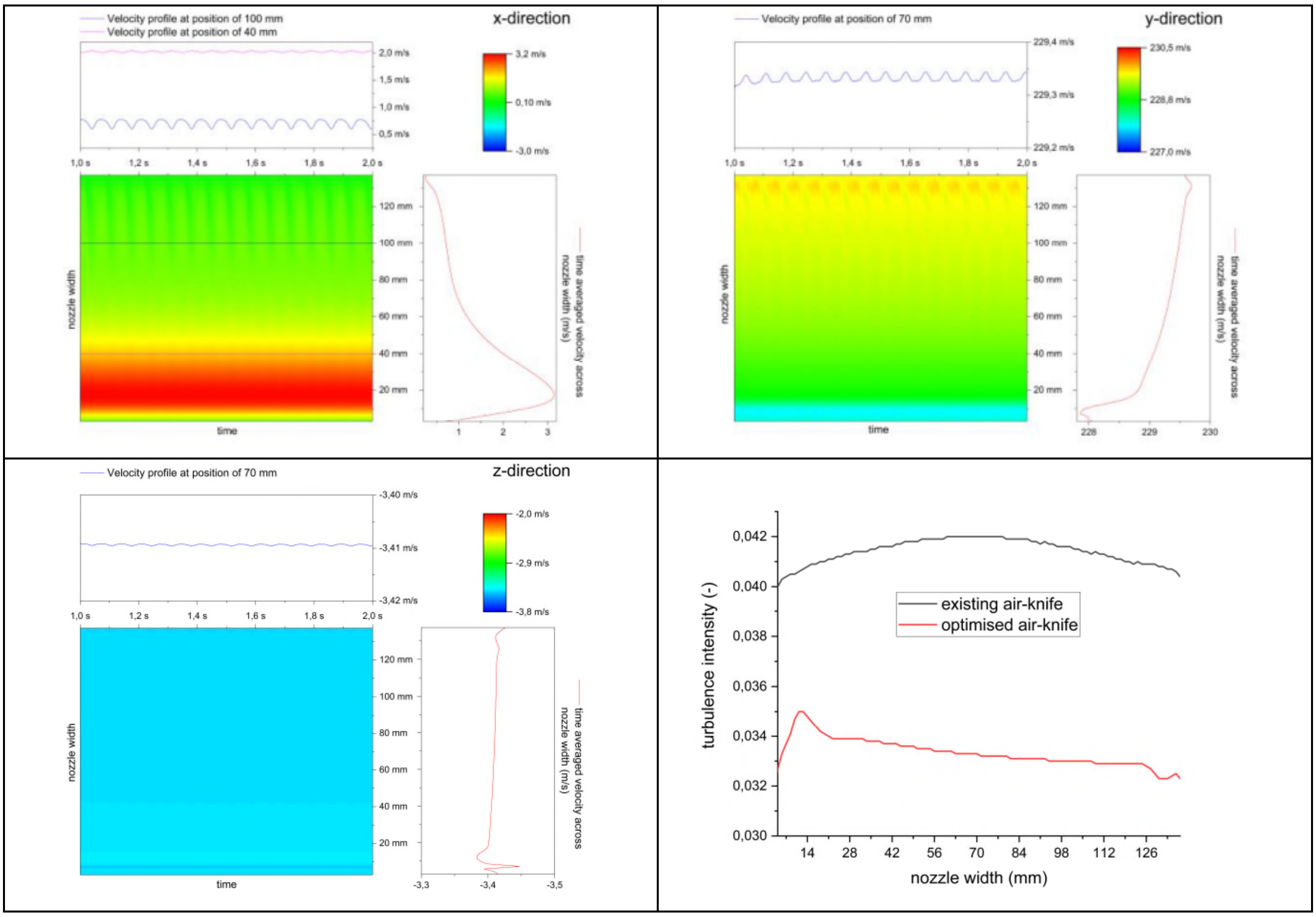

Fig. 10: Time dependent flow velocity (x-, y- and z-direction) and turbulence intensity at the nozzle exit.

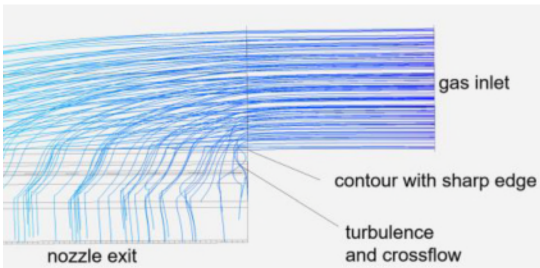

Fig. 11: Streamline visualisation of deflected gas flow (view from behind/below).

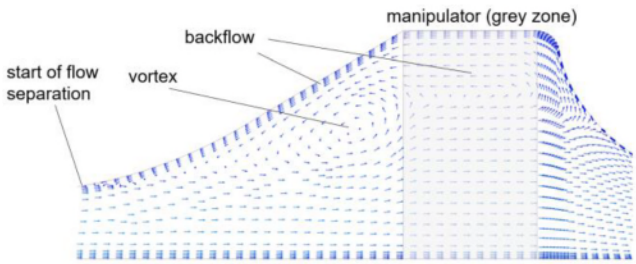

Fig. 12: Gas flow vector field visualisation at the exit of the feeding chamber and in the area of the flow manipulator (sectional view from the side).

Considering the findings of these simulations a final air-knife was designed, shown in Fig. 13. The fluidic features of this design are listed below.

- Gas inlet is again done only from one side into the feeding chamber. To compensate the decreasing volume flow and reach a uniform gas distribution across nozzle width the chamber is tapered in such a manner that the radius decreases as per root function, i.e. the cross-section area decreases linear. Because of this the chamber gets its organic looking shape.
- The sharp edge direct at the inlet, which was existent at the first optimised design and created turbulence is now rounded to get a smoother change in direction of the gas flow.
- As with the first optimized design a flow manipulator is installed to reduce turbulence and straighten the flow after deflection. The exit channel also narrows non-linear after a hyperbolic function to accelerate the gas flow steadily as possible.

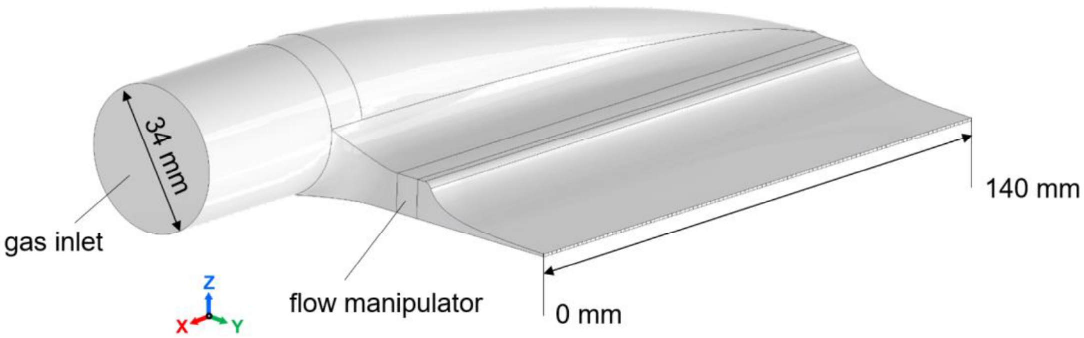

Fig. 13: Flow volume of final air-knife design.

With the final design, the resulting gas flow inside the nozzle can be further improved. No large vortices can be observed and the flow is very steady, see Fig. 14. At nozzle exit the flow velocity in main direction (y-coordinate) is very homogeneous over time and across nozzle width. Cross flows and their fluctuations are reduced once again (x- & z-coordinate). The uniform flow condition can also be seen in the turbulence intensity, which is lowest and most regular in comparison to the existing and first optimised design. For the new design a turbulence reduction of almost 25 % is reached, see Fig. 15.

In summary the simulation results indicate that fluidic measures like tapered feeding chamber, rounded edges, flow manipulator and non-linear exit channel are effective to optimise gas flow inside the nozzle.

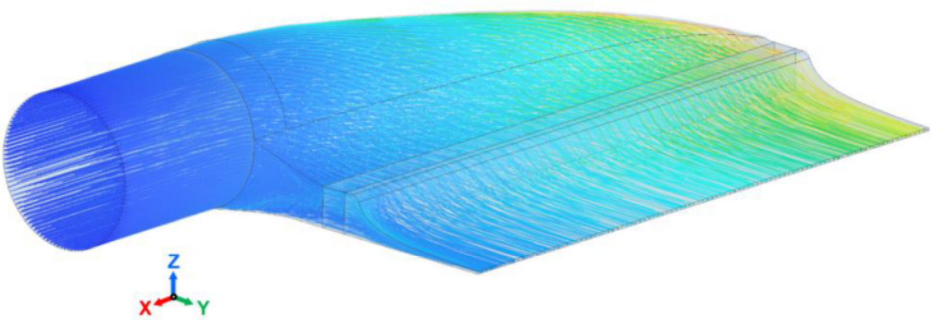

Fig. 14: Streamline visualisation of gas flow inside final air-knife
(representative snapshot, colour gradient represents different runtime of flow).

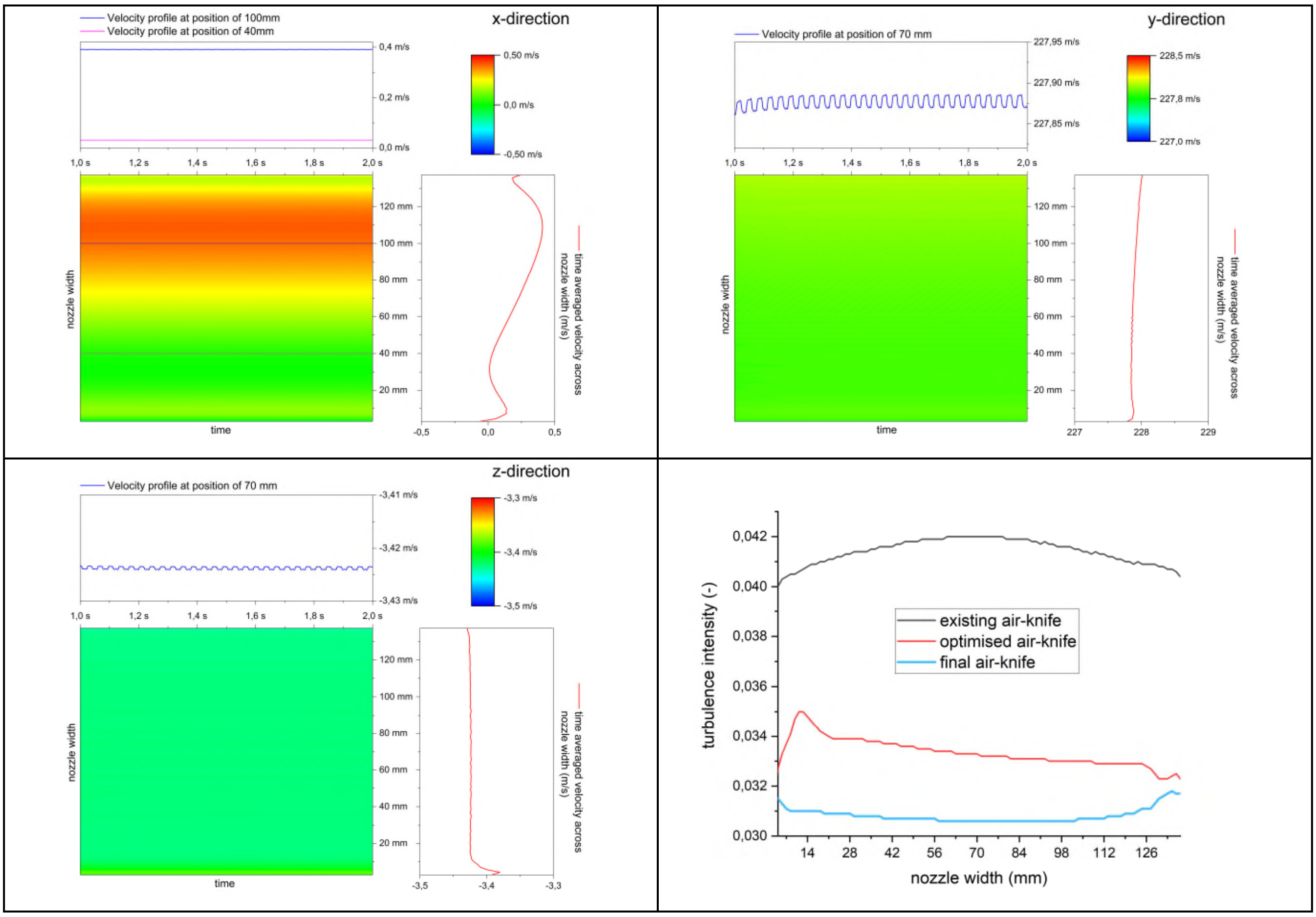

Fig. 15: Time dependent flow velocity (x-, y- and z-direction) and turbulence intensity at the nozzle exit.

In Fig. 16 the distribution of average flow velocity at the nozzle exit and turbulence intensity of the three considered air-knife designs are compared. You can see clearly the imperfection of the existing design especially regarding cross flow and turbulence. The final new design shows much better performance having in all directions a uniform flow velocity and lowest value for turbulence intensity.

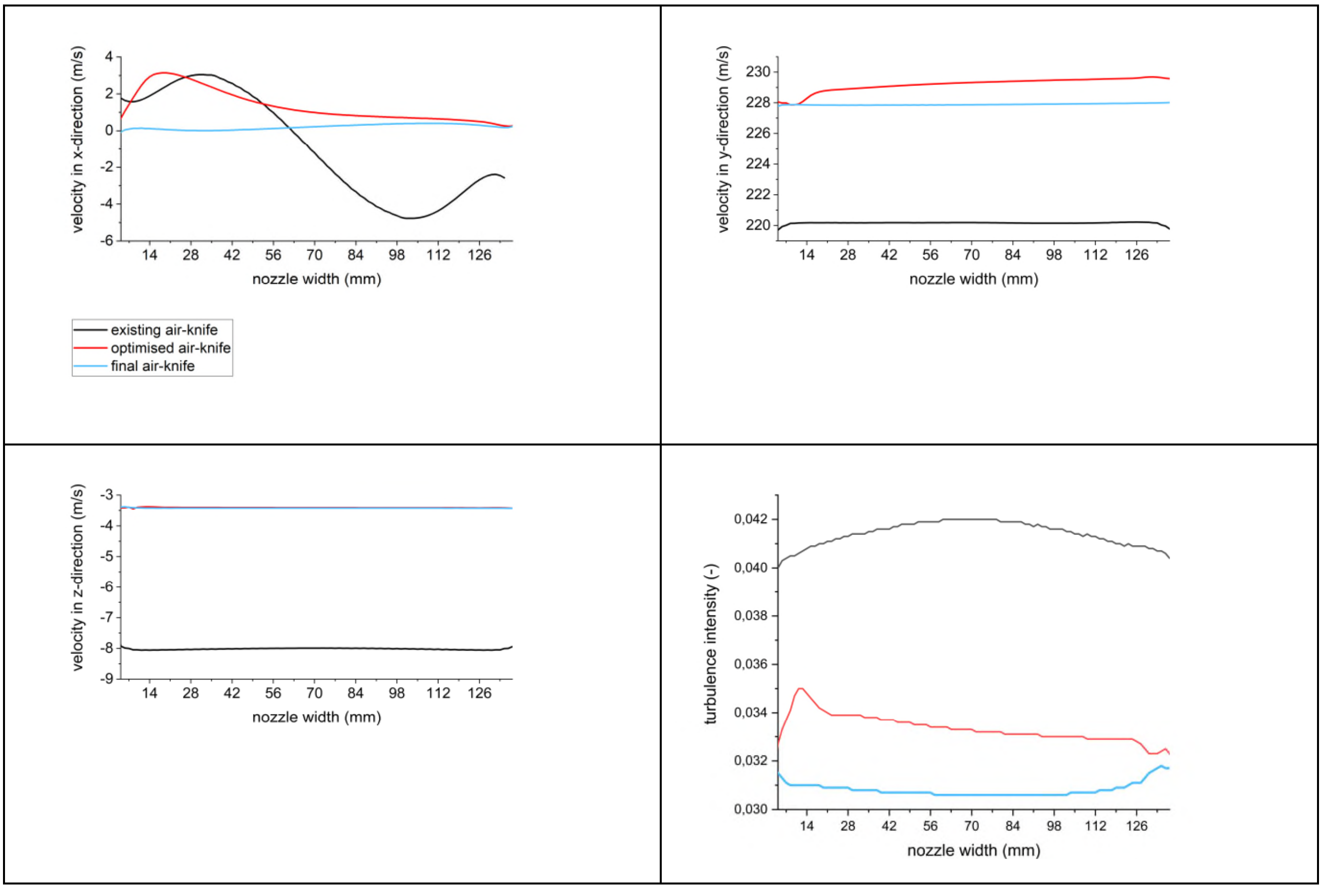

Fig. 16: Comparison of the velocity distributions and the turbulence intensity of the existing, the optimized and the final air-knife.

5. CONCLUSIONS

The inside flow field of the existing air-knife of the galvanizing simulator was modelled by transient RANS simulation, thus fluidic optimisation potential could be identified. The existing design shows a highly turbulent flow field inside the nozzle causing an inhomogeneous, fluctuating gas flow at the nozzle exit. The design lacks fluidic measures for soft flow deflection, turbulence reduction and smooth flow acceleration towards nozzle exit.

Based on the findings of simulation results some new models for an optimised air-knife were developed. It could be shown, that reducing the lateral gas inlet from both sides to only one side in combination with a feeding chamber is useful to reach controlled deflection. A more uniform gas distribution across air-knife width is reached with a narrow exit of the feeding chamber generating flow resistance. Unfortunately, this measure complicates the design of the following inner nozzle contour, because cross section changes need long overall length to avoid flow separation. A better solution was found for the final air-knife design, which uses a tapered feeding chamber to compensate the decreasing volume flow and does not require a narrow transit slot. As expected, the simulation proved that sharp edges, e.g. at the inlet, should be rounded sufficiently to minimize vortices. Furthermore, the final design comprises a flow manipulator to straighten the flow after deflection, and a non-linear narrowing exit channel to accelerate the gas flow close to sonic speed as steadily as possible.

The new air-knife design shows almost 25 % less turbulence and a much more even flow velocity at the nozzle exit in comparison to the existing configuration. The velocity variations could be reduced by several orders of magnitude, notable in direction of air-knife width.

Next step is to manufacture the new air-knife and test it at the galvanizing simulator, which is planned for 2021.

REFERENCES

1) J.A. THORNTON, M.F. GRAFF, Metallurgical Transactions B 7B (1976), p.607.
2) C.H. ELLEN, C.V. TU, Journal of Fluids Engineering 106 (1984), p.399.
3) E.O. TUCK, Physics of Fluids 26 (1983), p 2352.
4) M. DUBOIS, Proc. 8th Int. Conf. on Zinc and Zinc Alloy Coated Steel Sheet, Genova (2011)
5) E.O. TUCK, J.-M. VANDEN BROECK, AIChE Journal 30, (1984) p.808.
6) H. YONEDA, L.E. SCRIVEN, Proc. 7th Symp. AICHe Spring National Meeting, Atlanta (1994)
7) D. LACANETTE, A. GOSSET, S. VINCENT, J.-M. BUCHLIN, E. ARQUIS, Physics of Fluids 18 (2006), p. 042103
8) K. MYRILLAS ET AL., Chem. Eng. and Process. 68, (2013), p.26
9) H. G. YOON, G. J. AHN, S. J. KIM; M.-K. CHUNG, ISIJ International Vol. 49 No. 11, (2009), p.1755
10) C. PFEILER, M. MATALN, A. KHARICHA, C.K. RIENER, G. ANGELI, Proc. 10th Int. Conf. on Zinc and Zinc Alloy Coated Steel Sheet, Toronto (2015)
11) C. PFEILER, W. ESSL, G. REISS, C.K. RIENER, G. ANGELI A. KHARICHA, steel research int. 88 No. 9 (2017)
12) N. NORDHOFF, Bachelor Thesis, HAWK Göttingen (2017)