Design of air nozzle for air curtains

Nikola Žižkovsky[1,*], and *Armand* Drábek[1]

[1]VZLÚ - Czech Aerospace Research Centre, Beranových 130, 19905 Praha 9 - Letňany, Czech republic

Abstract. The procedure, which is used in the design of the air nozzle intended to be used on air curtain, is described in this work. The present work is focused on the isolated flow of one air nozzle, while next work will continue with air nozzles constellation. Nozzle reach in the flow field is monitored with and without tangential flow simulating non-isothermal flow. Several basic shapes of nozzles with the same cross-section are evaluated. After searching the design space of the parametric CAD model simulated in isothermal flow – without tangential flow, the best configuration with the longest range were selected. From this smaller set of nozzles was selected one nozzle with the highest directional stability in non-isothermal flow field. The shape of the nozzle with minimal directional deviation in the non-isothermal flow is subsequently optimized by the adjoint method. This method makes it possible to further reduce the pressure loss in the distribution element by deforming the shape according to the local sensitivity of cost function on shape deformation. The final step of design process is to verify effect of modified nozzle on both isothermal and non-isothermal flow filed.

1 Introduction

The development of all products should lead to an improvement in product properties. In the field of transport technology, on which this project is focused, it can be a wide list of features ranging from passenger's comfort to the energy consumption of vehicles. Recently, there is an emphasis on reducing energy consuption, or emissions of harmful substances during operation. In the case of vehicles, there are several areas suitable for flow optimization. While for high speed transport vehicles is optimal to improve external shape in order to minimize drag forces and in the same time comply, for example in rail vehicles, with respect to the slip stream velocities in the wake, for slower transportation vehicles, where the energy consumption for movement is associated generally with overcoming passive and inertial effects, savings in aerodynamics associated with vehicle air conditioning and ventilation are possible. The common denominator in the energy performance of air conditioning and vehicle ventilation is pressure losses and operational conditions. The pressure losses in the air channels depends on its shape, which is different for each vehicle. Even the operating conditions of individual vehicles may vary, but there is a difference between public transport and individual transport. While in individual transport the vehicle and passengers are moved to the same location, in public transport, passenger rotation is more frequent, leading, in particular in climatically more demanding periods, to higher energy demands for heating or cooling. Depending on the nature of the operation conditions, it is possible to reduce the energy loss caused by the opened entrance door using an air curtain. The benefit of the air curtain depends on the operating cycle, with the increasing ratio of travel time to open door time the importance of the air curtain decreases. Air curtains are beneficial for public transportation with frequent stops. In previous projects, see [1], the dependence of heat loss on the air curtain settings was found.

2 Problem

The basic problem of air curtains is their range, which is shortened due to non-isothermal flow, which mixtures and also bending the stream behind nozzle. For air curtains, in respect of proximity to passanger's heads in the boarding place, it is advisable to limit maximum velocity behind the nozzle, the parameter which affect the air stream range mostly. The air curtain range as parameter is difficult to define, because the velocity profile develops with the distance from the nozzle as well as with the distance from the air stream axis, see [2]. In the case of a non-isothermal flow, the air flow is deformed. For this reason, the air flow behind nozzle is evaluated first for isothermal flow filed, where the nozzle and flow parameters are monitored and then the most promissing configurations for futher optimization are selected. Futher simulations runs with non-isothermal flow field, from which the configurations with minimal deviation of maximum velocity at reference distance from nozzle is selected for optimization.

3 Objectives

The aim of this work is to describe the procedure used in the development of shapes of air nozzles for longer reach

*e-mail: zizkovsky@vzlu.cz

https://doi.org/10.1051/epjconf/202226901073

of the air curtain. For this purpose several nozzle configuration with simple parameterization are processed. The basic shapes are shown in figure 1. The basic shape of the considered nozzles is the same and only the dimensions and internal parts differ. The oultet area of all nozzles, resp. the sum of the annulus and circles are identical. The basic configuration T0t has the smallest diameter, the variant T1t has the outer diameter larger and has a solid shaft in the nozzle's axis. The T2t variant is similar to the T1t variant, but the solid shaft is replaced by a tube. The diameter of the nozzle T0 is 9 mm, while the outside diameter of the outlet T1 and T2 is 12 mm.

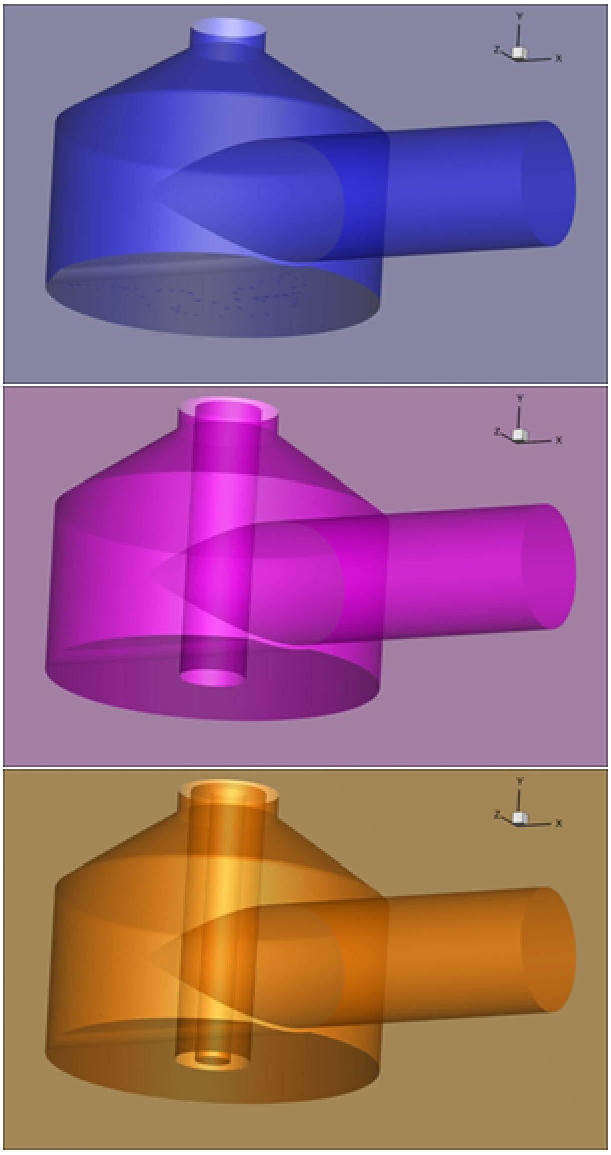

Fig. 1. Geometry from the top: T0t, T1t and T2t for zero angle of tangential input.

4 Methods

4.1 Basic variants

To initially assess the suitability of the air nozzle topology, all three basic variants were simulated in a free isothermal flow field with tangential inlet angles varying from 0 ° (perpendicular to the nozzle's axis) to 90 ° (parallel to the nozzle'saxis). From the analysis of the axial velocities, one to two variants with the highest axial velocity at a distance of 0.4 m from the diffuser were selected from each variant T0, T1 and T2. The automated selection was supplemented by configurations with favorable axis velocities. The simulations was carried on a computational domain with hemispherical shape with radius of 2 m, in which the surface of the hemisphere represented a free stream and the flat plane represented a viscous wall, in which center the nozzle was located. The nozzle is also modeled as a viscous wall. The diffuser has one or two inputs according to the configuration. Selected configurations were subjected to analysis in non-isothermal stream in own solver. The simulations were considered as compressible, viscous and time steady. A coupled solver RANS with W & J EARSM + Hellsten turbulence model was used for k-ω. Wall functions were not used. The computational grid was created in PointWise. The volume is represented by tetrahedrals cells with prismatic cells at the wall surfaces. The computational grid is refined in a distance up to 1 m from the air nozzle. The computational mesh defined in depicted way contains approximately $21 \cdot 10^{6}$ cells, depending on the configuration.

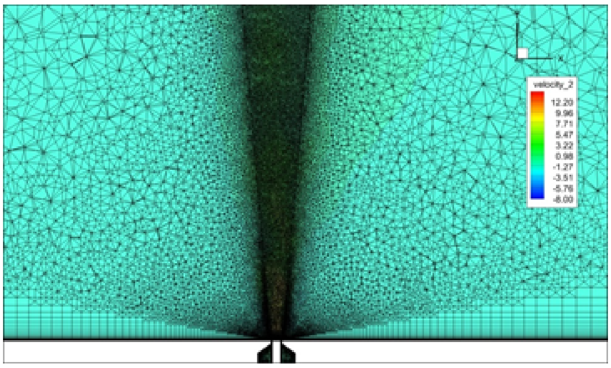

Fig. 2. Computational mesh - T1 variant.

4.2 Non-Isothermal flow

Simulations selected as the most promissing in isothermal flow field are simulated in non-isothermal flow field, which is in this case simulated using a constant velocity approaching in the direction perpendicular to the axis of the diffuser. The actual implementation of static buoyancy forces due to different temperatures in the gravitational field leads to physically accurate results, but at the same time the solution imposes numerical instability. The magnitude of the tangential velocity, $1\ m \cdot s^{-1}$, was chosen based on the speeds read from other simulations of the non-isothermal flow on the target air curtains application.

From these variants, three configuration were selected for further processing, of which only the T2t45 variant will be listed here.

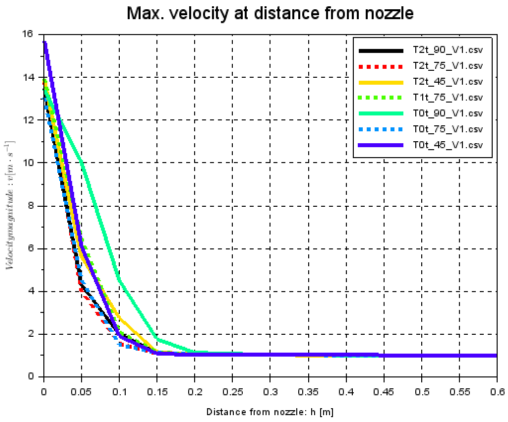

Fig. 3. Distance of maximum velocity behind nozzle.

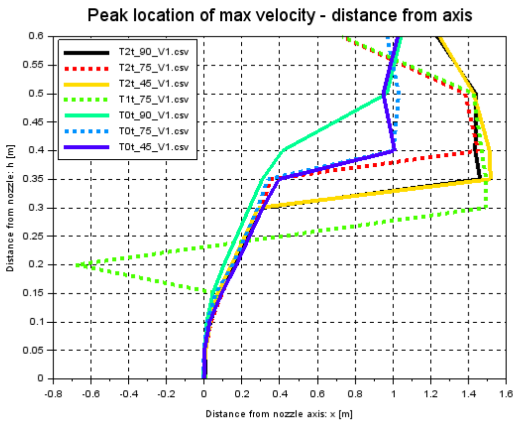

Fig. 4. Maximum velocity peak distance from axis.

4.3 Optimization

The final step of the air nozzle design process is the shape optimization of the considered geometry. Gradient method using Adjoint solver implemented into Star-ccm+ solver is used for optimization. In this case, the optimized geometry is limited only by the interior of the diffuser, see figure 1, without the hemispherical computational domain. The outlet of the diffuser is considered as atmospheric output (*Pressure Outlet*). The boundary condition describing the total state (*Stagnation Inlet*) is used as input, with pressure values corresponding to the total pressure calculated in the previous calculation steps. The computational mesh consists of polyhedral cells in volume, supplemented by prismatic cells on the viscous walls. Generated computational mesh contains abou 921000 cells depending on the configuration. Time steady, compressible, viscous model was used with a turbulence model k-ω SST and a coupled solver. The aim of this optimization process is to minimize the pressure drop across the volume of the diffuser, maximize the uniformity of the outlet speeds and maximize the outlet speed. Based on these requirements, a cost function has been set up, which is maximized by the used solver.

Table 1. Design comparison - monitored parameters.

Parameter	**Input geo.**	**Optimised**
Velocity magnitude $[m \cdot s^{-}1]$	11.6074	12.2405
Axial velocity $[m \cdot s^{-}1]$	9.75486	11.2103
Flow uniformity [1]	0.865	0.929
Pressure drop $[Pa]$	43.67	29.11
Cost function	0.2059	0.3765

$$F = 1 - \frac{\sum_f \mid v_f - \bar{v} \mid \cdot A_f}{2 \cdot \mid \bar{v} \mid \cdot \sum_f A_f} \cdot \frac{\sum_f A_f}{\sum_f v_f \cdot A_f} \cdot \Delta p_T^{-1} \qquad (1)$$

where v is the speed of the output boundary condition (mean value with overline), A_f is the partial area of the nozzle's boundary condition, p_T is the total pressure, respectively difference in inlet and outlet total pressure.

Figures 5 and 6 shows the comparison of input and optimized geometry.

Numerical comparison of input and final geometry parameters is given in the table 1.

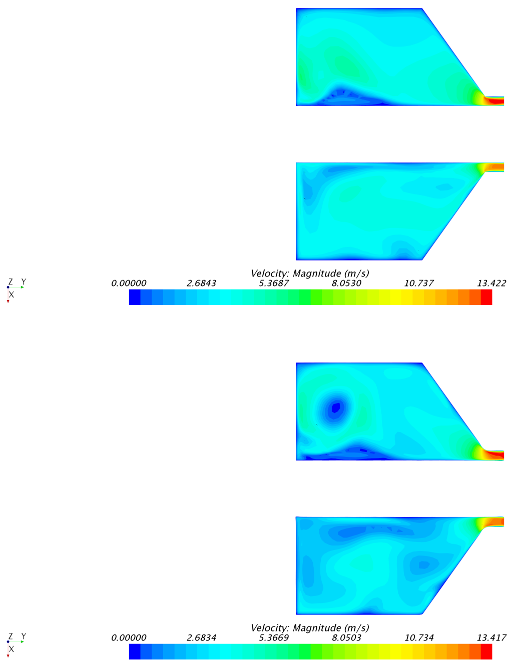

Fig. 5. Nozzle original (upper) and optimized shape - velocity flow field.

5 Conclusion

The text describes the procedure used in the selection of topologies suitable for the application of circular nozzles as air curtains, which are subsequently optimized by the gradient method with the adjoint solver. Unfortunately, the entire design process, selection and optimization is not fully automated. From whole process, only last mentioned part was automated - the optimisation with predefined cost function. The last step is to verify the proposed shape of the diffuser in the non-isothermal flow. The result of this work is a nozzle with a better cost function value of 83 %, a lower pressure drop of 33 % and a better flow field uniformity of 7 %. The target function also resulted in a 15% increase in the output velocities in the axial direction while the velocity magnitude increased only by 5 %. The target function aligns the original rotating flow.

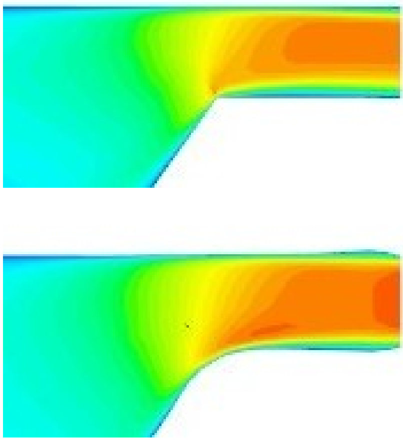

Fig. 6. Nozzle original (upper) and optimized shape - velocity flow field - detail.

In the next part of the project the aim is to create a set of nozzles with lower overall energy consumption, which could replace the slot nozzles used on air curtains.

Acknowledgements. This result was achieved within the institutional support of the Ministry of industry and trade for the development of the research organization.

References

1. N. Zizkovsky, 24th International Conference Current Problems in Rail Vehicles **Volume II**., 365-371 (2019)

2. E. Janotkova, *Technika prostredi - 1. část* (Vysoké Učení Technické v Brně-Fakulta strojního inženýrství-Odbor termomechaniky a techniky prostˇredí EÚ, Brno, 2011)58