11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

PROBLEMS OF INDUSTRIAL AIR EXHAUSTING

KAREL ADÁMEK

VÚTS Liberec, a.s.
Svárovská 619, 46001 Liberec, Czech Republic
e-mail: karel.adamek@vuts.cz

Key Words: *Air Exhaustion, Flow Balance, Numerical Flow Simulation.*

Abstract. This document describes problems of industrial exhausting and their possible remedy.

1 INTRODUCTION

The correct running of industrial equipment operating high-quality, reliable and economic production needs the exhausting of planned (projected) air volumes from several branches into one common exhaust fan. An example of such a system is presented in Fig. 1.

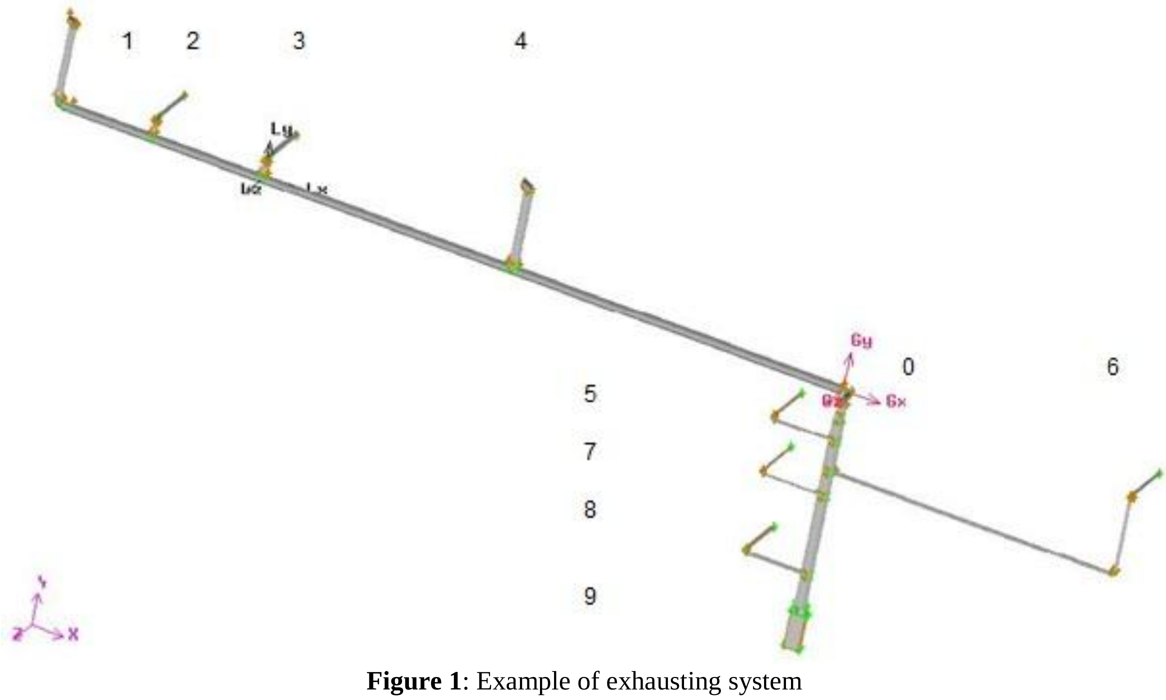

Figure 1: Example of exhausting system

The scheme of the tube system is quite uneven, the lengths, diameters, flow rates in individual branches are very different; the system is situated in a square volume of about 6000 cubic meters. In general, the range of flow ratios in individual branches is of 3:1 approx., only in the branch 4 the flow is higher, with the ratio of 10:1. It is possible that any problems can arise by tuning the flows in such a system.

Therefore, the pressure conditions are very different; too, they could be the reason of actual operating problems. They can be suppressed by regulating flaps, set individually and subjectively. The additional changes of the tubing system can have some influence, too, not listed in the obtained documentation. An additional exhaust in the summer period can have another influence - in the exhausted area, there arises some under pressure and the global pressure conditions in the solved / operated system are changed as well.

2 NUMERICAL FLOW SIMULATIONS

Due to uncertain conditions of pressure, flows etc., the numerical flow simulation in the whole system was realized instead of standard calculation of “linear” pressure losses in individual straight tubes with “punctual” local losses. But such a classical method is not fully exact. For instance, the flow resistance of so-called T-joint depends on the flow rates in all 3 connections [1] and the flow field, disturbed by such a junction, is equalized until after the adequate straight tube length, typically of about ten times of relevant diameter. Therefore, the numerical flow simulation gives a better description of the flow field (pressure losses, velocities, turbulences, etc.) in the whole tubing system.

2.1 General description

The geometry of the system is relative simple - cylindrical tubes with reductions, T-joints, elbows and regulating flaps in each branch.

The used mesh is made from relative rough hexahedral elements; the areas of T-joints are made from tetrahedral ones, see Fig. 2. For general overview, such rough mesh is sufficient, here there are not solved details of flow mixing, wall flows, etc.

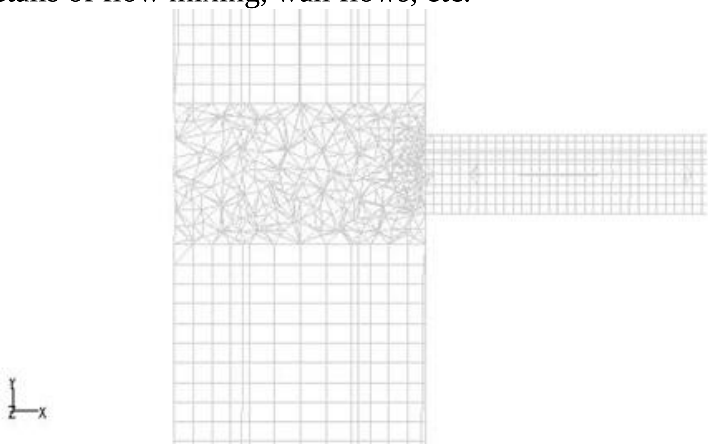

Figure 2: Detail of mesh in branch connection

The model is isothermal, with barometric pressure defined in individual inlets. In the common outlet the suction value is defined to get the prescribed total outflow volume. At the beginning of calculation the “hard” pressure source is used, later it is added the real exhaust fan of defined characteristic $\Delta p = f(V)$.

The used solver is steady, model turbulence k-ε, the used medium is air as ideal incompressible gas.

2.2 First result

The pressure field of the solved system is presented in Fig. 3. For better system description the individual branches are labeled 0 - 9. The expected results are as follows:

The formal branch 0 is a spatial short elbow, only.

The longest branch 1 is well sized; the simulated flow is greater than the project.

The flows in the other distant branches 2, 3 and 4 are markedly smaller than in the project. The question is, if some throttling of the branch 1 could affect the flows increasing in branches 2, 3 and 4.

On the contrary, the flows in the short and near situated branches 5, 7 and 8 are much greater than in the project. Their restrictions/ throttling, used for the flow decreasing, could create some flow instability in the system.

The flow in the long branch 6 is corresponding to the project.

The result explains why the total exhausted flow 9 reaches of about 60% of projected nominal flow, only - it is the negative result of the flow "tuning" by additional restrictions/ throttling of several branches.

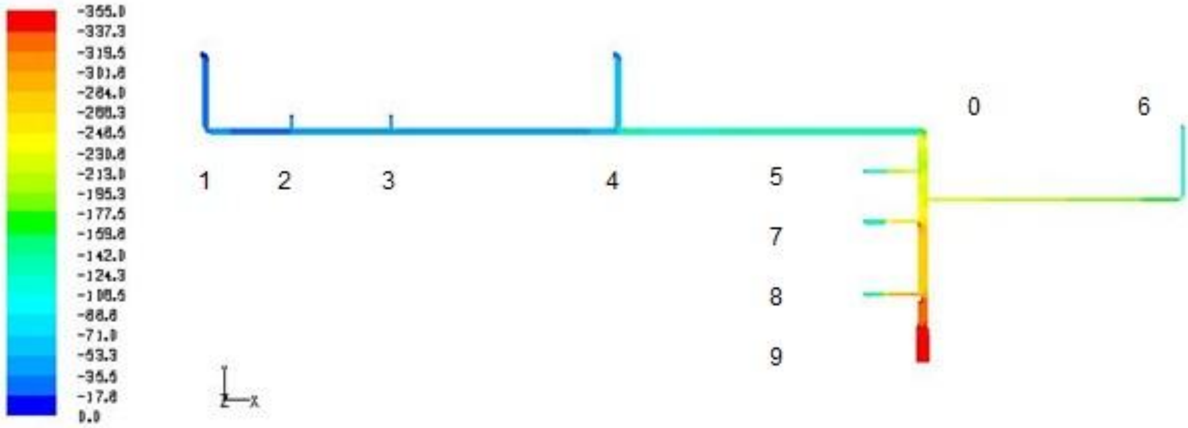

Figure 3: Pressure field in solved tube system with connections marking

As an illustration, in Fig. 4 and Fig. 5, there are the complicated pressure and velocity fields in the area of the connection of one branch. It is visible that values in the observed area are not constant, therefore, the following evaluation of average values of pressure and velocity in individual cross sections of the system contains some deviations. It is clear that so-called local loss, inserted in a specific point of the tube (here at the T-junction) is not fully correct without specifying the flows in individual branches [1].

In general, in the junction point of the main and lateral flow, both velocities are locally decreasing and the pressure is locally increasing (so-called stagnation point). Along the inner side of lateral bend the flow is separating, with backflow in the wake.

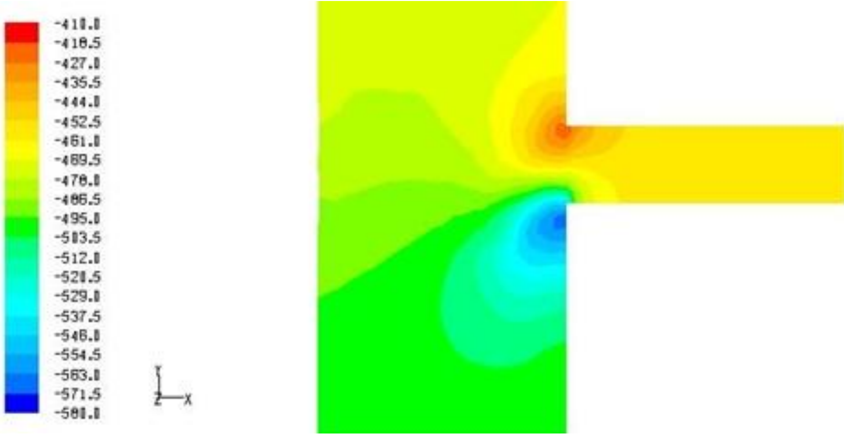

Figure 4: Detail of the pressure field in the connection of the branch 6

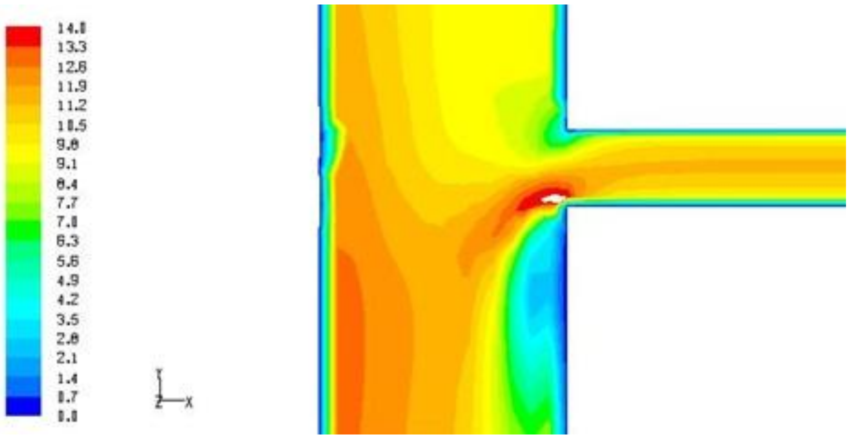

Figure 5: Detail of the velocity field in the connection of the branch 6

3 SYSTEM TUNING

3.1 Theoretical basis

The pressure resistance is theoretically proportional to the second power of velocity, then to the flow, too. The first possibility, how to get the required flow rates, is to change the flow resistances. A simple change is not sufficient because the change in one branch affects a little the flows in adjoining branches, too. The standard procedure of pressure tuning consists of checking of pressure losses, step by step in each connection point from the end of the system to the common outlet. If necessary, the pressure resistances should be modified in each connection point.

The well-known formula for the pressure loss in the straight tube, after the adaptation using the continuity relation, is

$$\Delta p = \lambda \,.\, L/d \,.\, \rho/ \,.\, w^2 = konst \,.\, M^2 \,.\, L/\,d^5, \tag{1}$$

where M (kg/s) planned flows in individual branches,
L (m) planned lengths of branches,

d (m)	planned diameters of branches,
w (m)	flow velocity in branches,
ρ (kg/m^3)	density of transported air, regarded as constant (= 1,2),
λ	friction coefficient of tube.

For smooth tubes is after [2]

$$\lambda = 0{,}0072 + 0{,}61 / Re^{0{,}35} \tag{2}$$

where the Reynolds number is

$$Re = w \,.\, d / \nu \tag{3}$$

where ν = 13,72e-6 (m^2/s) is the air kinematic viscosity.
In the observed system there is the Re = 100000 approx., it means the really turbulent flow, the corresponding values are λ = 0,015...0,02.

If the pressure resistance of two branches in their connection point is not the same, the flows are another than planned and it should be necessary to insert any additional resistance into the branch of smaller resistance, practically to close the flap a little.
This "linear" resistance of straight tubes should be completed by so-called local losses, here for instance by elbows, reductions, T-joints. They can be formulated as so-called equipotent lengths, added to the straight lengths. This is the length of the straight tube of the same resistance as the local loss. After above mentioned simulations (see for instance Fig. 4 or Fig. 5 above) it is clear that the resistance depends on the real configuration, arrangement and flow rates so that the real value of inserted resistance can be different from the value given in any handbook. The value for individual T-joints depends on the flow values and directions in all three inlets/outlets, see for instance [1, 2]. So the real flow rates and influences of adjoining connections should be better to express by numerical flow simulation in the system.

3.2 System adaptation

The used tube diameters are scalable with the quotient of 1,25 :1, then the cross sections are scalable with the quotient of 1,5625 :1. If the flow rate is greater than this value of 1,5625, it is necessary to use a smaller tube diameter. If the flow rate is smaller than 1 :1,5625 = 0,64, it is necessary to use a larger tube diameter. The remaining pressure surplus should be always restricted (tuned) by flap setting. If the flow rate is a little less than 1, only, it is possible that by changes in adjoining branches the flow in the observed branch could be changed, too. Generally said, in all connections it should be any small overflow, to have the real possibility of tuning it by throttling by flap.

Using such various adaptations the total flow can be changed, too, so that at the end it could be adapted by total pressure difference in the system.

3.3 Result

The received main results of the tuning procedure are presented in following graphs. Regarding the large differences of flows (kg/s) in individual connections - after the received data the range until 10:1 - the relative values are presented here as a ratio real / planned. The main steps of the solution are presented in Fig. 6.

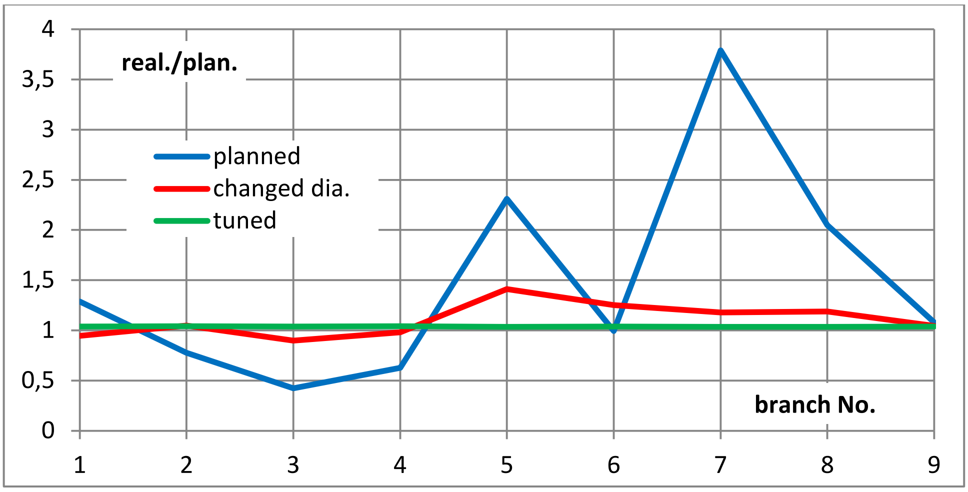

Figure 6: Relative flows in branches 1 to 9 - planned, changed diameters, tuned

Step 1 - flows calculation in planned dimensions, without restrictions, pressure gradient at the outlet of 400 Pa. The system is not well balanced. Flows in some branches (2, 3, 4) are smaller than planned, it should be to use larger diameters. Flows in another branches (5, 7, 8) is greater than planned, it should be to use smaller diameters.

Step 2 – flow calculations in adapted diameters. The system is quite well balanced, but in some branches the flow remains under planned values. This remaining problem was resolved by increasing the fan pressure gradient at 580 Pa.

Step(s) 3 – repeated flow calculations, tuning by branches restrictions, reached resulting differences of 0.1%, only.

Fig. 7 shows the same tuned result of the step 3 in enlarged scale. The average value of tuned branches is of 103.8% of planned flows, standard deviation of 0.2%. To get planned flows (100%), the smaller pressure difference was used, 540 Pa, only, instead of former 580 Pa. It is evident that such interventions make some unbalance again, the average flow of 98.8% with standard deviation of 1.7%. It should be to continue in small additional tuning/ restrictions of flows in some branches, to get a well-balanced system again.

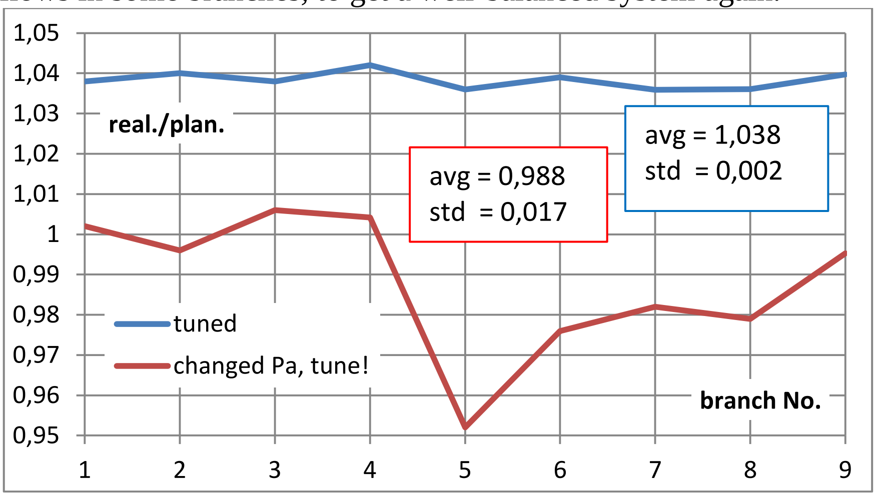

Figure 7: Relative flows in branches 1 to 9 - tuned and unbalanced again by another pressure gradient

It is not known in advance if the flows interferences are given by flows mixing, only, or by some mutual flow influences along tube walls, too. Therefore, it was observed the influence of finer mesh in the whole volume, too, and more, the influence of the wall parameter y+. Some influencing is observable, but it is not very striking, see Fig. 8. Using two kinds of finer meshing, the average flow is changing on 107.4% or 109.9% respectively with standard deviations of 3.6% or 3.8% respectively. From all three figures, Fig. 6 to Fig. 8, it is clear that every intervention into the already tuned system means some unbalancing again and it is necessary to make some additional tuning again.

In general, the tuning of branches 1, 2, 3 and 4 needs the restrictions of some tens of Pa, therefore, the real operational setting will be very sensitive. On the contrary, the branches 5, 7 and 8 need the restrictions of some hundreds of Pa, their operational setting will not be so sensitive. The question is if such additional tuning in an order of some Pa or tenths of Pa could be set in the real operation.

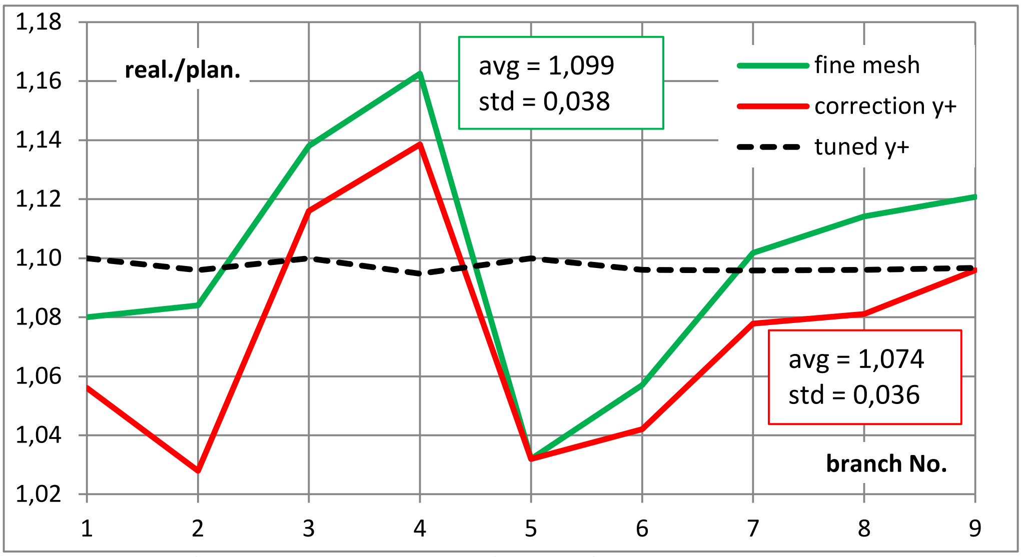

Figure 8: Relative flows in branches 1 to 9 - influence of changed mesh on the tuned system

3.4 Operational point of the system

The results of the previous paragraph use a so-called “hard” source of the flow / pressure. For the real operation it should be to use real fan characteristic $\Delta p = f(V)$ and to state the real operational point of the system. After Fig. 9 it is the crossing point of both characteristics - the increasing resistance of the tube system (typically the curve of the second order after (1)) and the decreasing fan characteristic (typically the curve of the second or the fourth order, after catalogue). Probably, the above described tuning procedure should be repeated for such a real operational point.

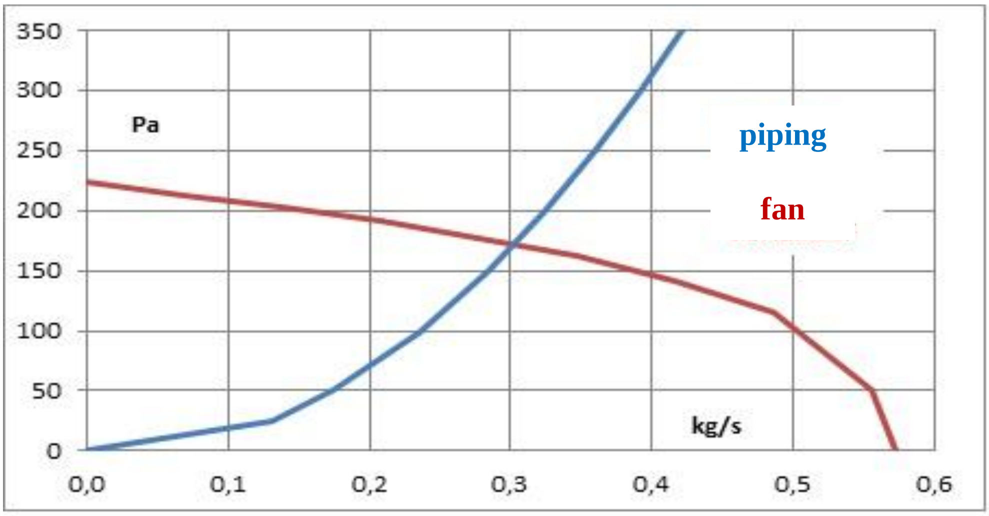

Figure 9: Typical operational point of the system fan + tubing

4 FLAP RESISTANCE

The flap resistance is corresponding to the flap setting toward the flow. The received real exponential correlation between flap resistance and flap position can be defined from separated numerical models of real flaps in real tubes, see the example in Fig. 10.

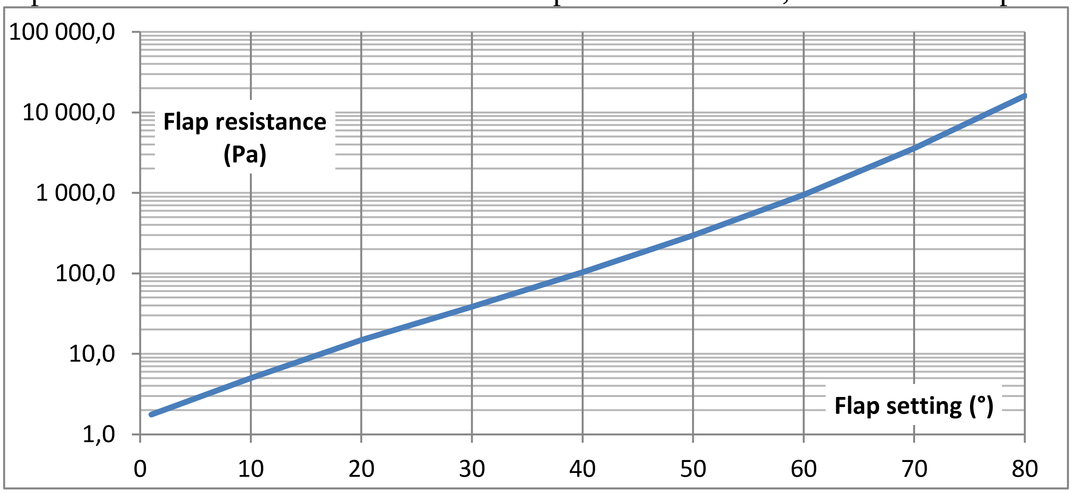

Figure 10: Flap resistance at various setting

The flap resistance is increasing exponentially with a very high coefficient of correlation

$$\Delta p = 1{,}4337.\exp(1{,}1117.V) \qquad R>0{,}999 \tag{4}$$

The separately received results will be adopted into the real system - the entire model of the tube system with flaps would be very complicated and time consuming. For practical use for the solved system it is possible to assign simply the above unknown flap setting to the above calculated flap resistance, necessary for the system balancing.

Corresponding illustrations of the pressure field around the flap in the tube at positions 0°, 30°, 60° are shown in Fig. 11 to Fig. 13. The flap resistance is increasing considerably with the flap setting toward the flow direction.

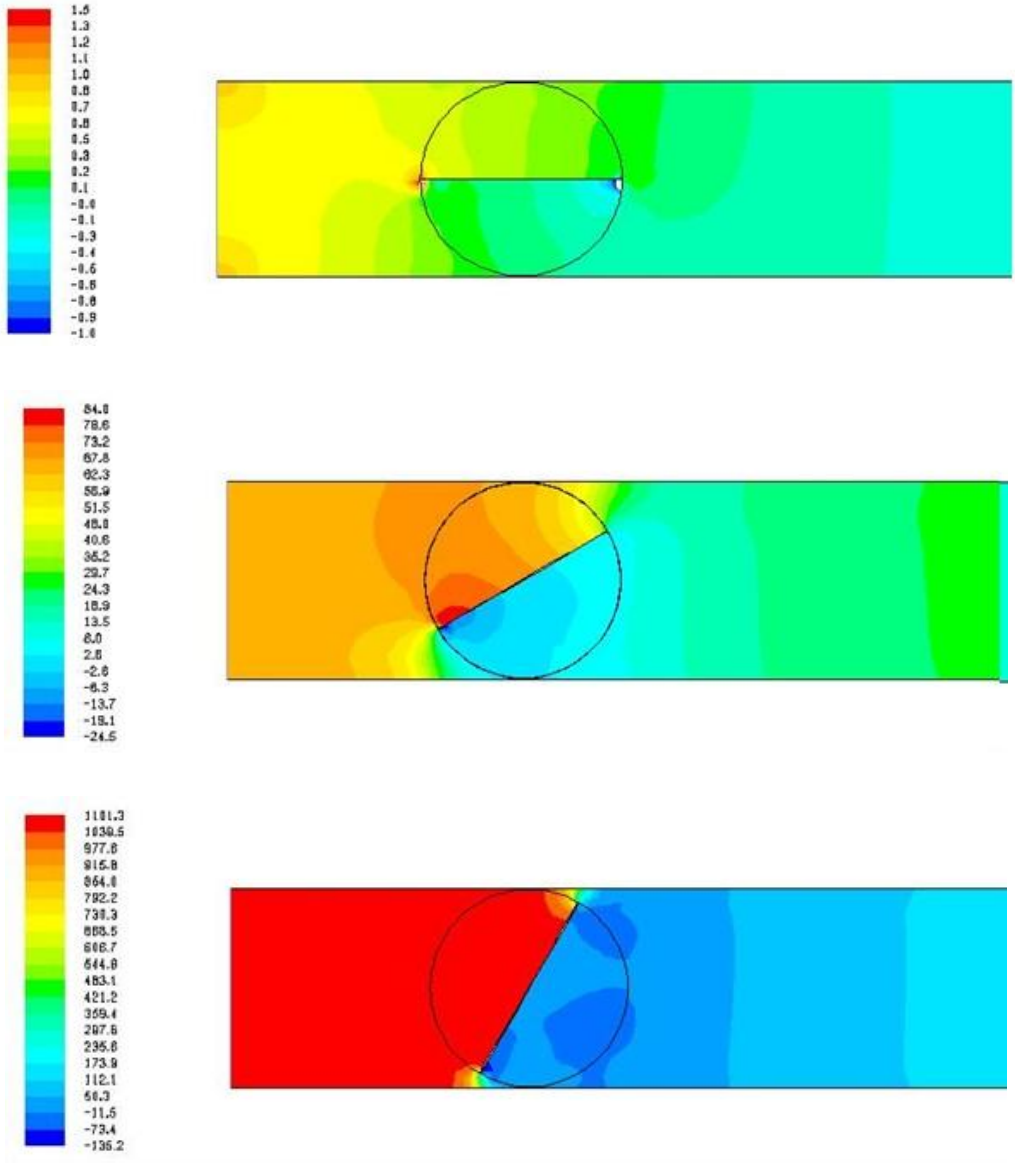

Figure 11 to 13: Pressure fields around the flap in tube at flap setting 0° - 30°- 60°

As an illustration, only, in Fig. 14 there is added the velocity field around the flap at the position of 30°. It is visible the long area of the flow disturbed by the flap. The real flap setting should not to be very high, max. 60° approx., the very intensive throttling of the flowed cross section could cause some operational problems - vibrations, noise, instabilities, etc.

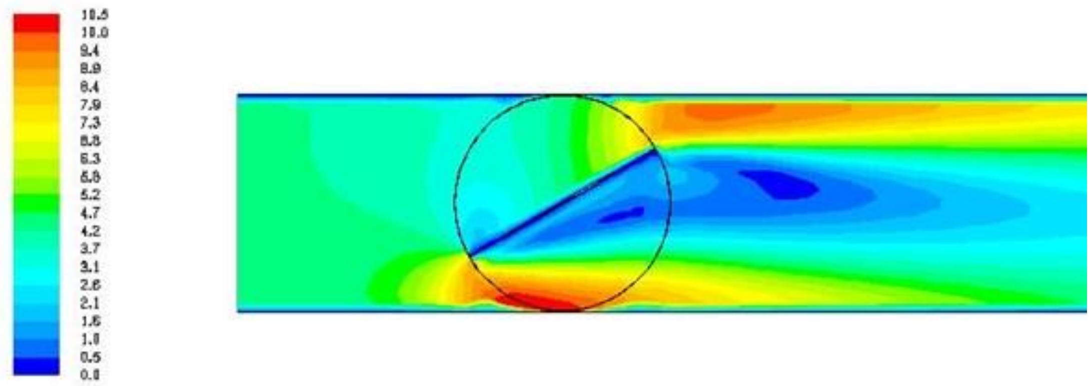

Figure 14: Velocity field around the flap in tube at flap setting of 30°

5 MEASURING

To verify the above mentioned method of the system balancing, using numerical flow simulation, it should be to realize any real operational measuring. Due to a complicated structure, the velocity / pressure profiles are very disturbed, therefore, the locations of the used sensors must be defined very carefully.

5.1 Cross profiles

The cross velocity profiles just after individual connections are quite uneven, see Fig. 15 and Fig. 16.

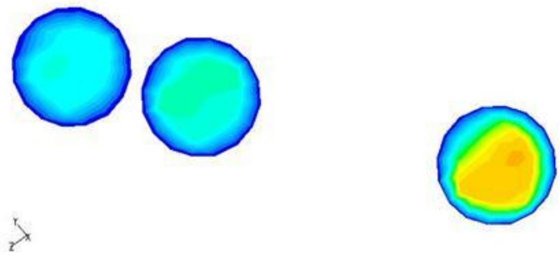

Figure 15: Velocity cross profiles just after connections 2 - 3 - 4

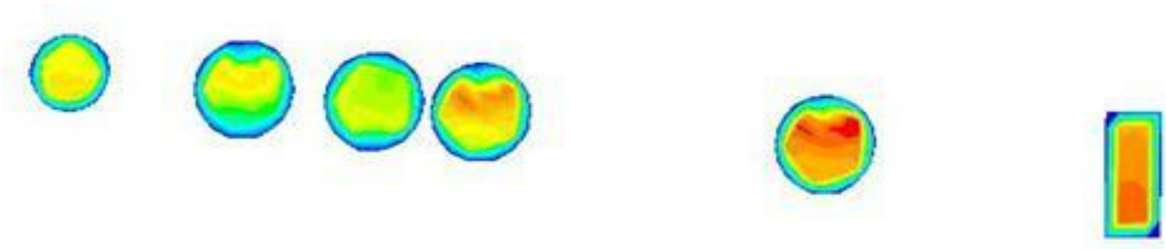

Figure 16: Velocity cross profiles just after connections (0) - 5 - 6 - 7 - 8 - outlet

After [3], etc. for the ideal turbulent flow the ratio of medium to maximum velocity is ws/wm = 0.84. For the observed profiles in Fig. 15 and Fig. 16, the ratio is varying between

0.79 – 0.86, with two extreme values of 0.71 and 0.91. Therefore, it should be expected that the simple one-point velocity measurement will be loaded by some error.

5.2 Axial profiles

Fig. 17 shows the velocity fields in axial cross sections in the vicinity of individual branch connections to the main tube. In general, after each connection there are considerable velocity field deformations together with backflows. It is necessary to take it into account when defining the right sensors locations. For instance, the suitable positions are before the connections 2 - 3 - 4 (horizontal orientation). At connections 5 - 6 - 7 (vertical orientation), situated near one another, it is hardly to find any suitable position, the flows are mutually very affected.

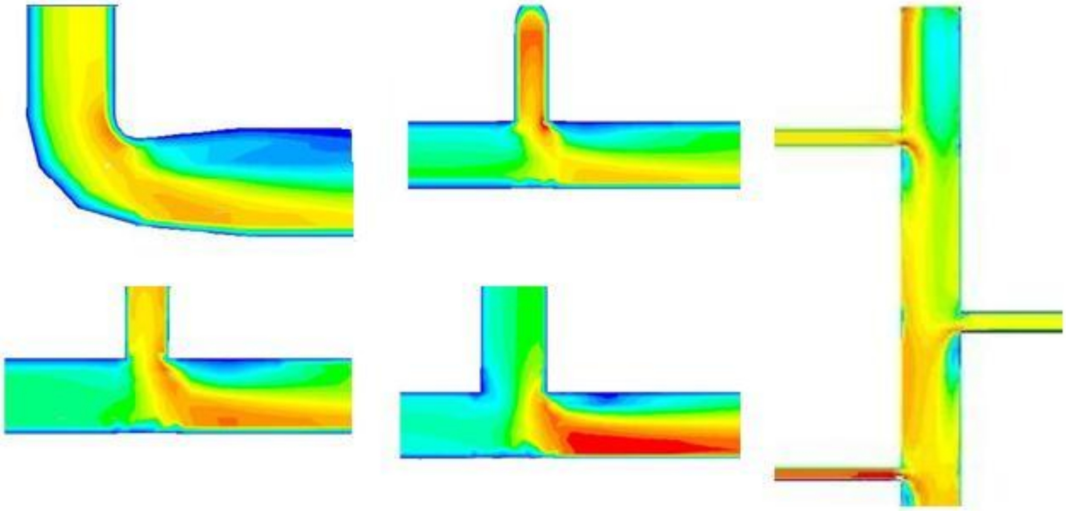

Figure 17: Velocity axial profiles at areas of connections No. 1 - 2 (up), 3 – 4 (down), 5 - 6 - 7 (vertical)

6 CONCLUSIONS

- The classical solution of flows and resistances in the given system is possible. It means to define any losses in straight tubes and to add constant local losses (elbows, reductions, T-joints). But from the simulated images of pressure and velocity fields, it is visible that every such local loss is not constrained locally, but it influences a certain length of the model. Typically, it needs a length of 10 diameters approx. to get a quite even flow profile again. And more, the neighbouring branches are mutually affected, too.
- For this reason, it is more suitable to use the method of numerical flow simulation in the whole tubing system and during several steps of iteration to get an optimal tuning of the whole system, with necessary corrections of some given data. Raw correction by diameters changes, fine correction by flaps setting. On the other side, the real operational setting of individual resistance of some Pa, only, could be problematic.
- From the results of numerical flow simulations through flaps installed in tubes it can be assigned the operational flap setting corresponding to the pressure resistance, necessary for the right tuning of the system.

- Checking of results from numerical simulations by real measuring needs the right placing of measuring sensors. Unsuitable placing means incorrect results.

REFERENCES

[1] Štugler, J., Klas, R., Šperka, O.: *Characteristics of the T-junctions...*. EPJ Web of Conferences, vol. X, (2013), p. 664-675.
[2] Kubíček, L.: *Zdravotní vzduchotechnika*, Práce Praha, (1956).
[3] kol.: *Příručka měřicí techniky pro strojírenství a elektrotechniku*, SNTL Praha, (1965).

ACKNOWLEDGMENT

The work was created under support of VUTS - Centre for Development of Research in Machinery, according to an industrial order.