Hydraulic losses optimization methods used in air conditioning system valve design for different closure angles

K S Napreenko and A V Lamtyugina
Moscow Aviation Institute (National Research University), 4 Volokolamskoe shosse, Moscow 125993, Russia

konaprko@yandex.ru

Abstract. This article describes methods for determining the hydraulic resistances of A/C system units. The advantages of computational hydrodynamics over the traditional method of calculation using the coefficient of hydraulic resistance are shown. A “step-by-step” example of calculating the hydraulic resistance of an A/C system unit in the ANSYS CFX software package is given (converting 3D geometry into a calculation model; selecting, configuring, and building a mesh model; configuring calculation parameters, including boundary conditions, turbulence models, and calculation stop criteria; calculating and processing results). Special attention is paid to the consideration of the SST turbulence model. The calculation results show the distribution fields of total and static pressure, velocity, and flow lines at discrete rotation angles. Graphically, the dependence of the drop in hydraulic resistance on the increase in the area of the flow section of the damper is shown. Hydraulic losses are also determined when the pressure increases. Conclusions are made about the application of the calculation results in creating a comprehensive model of the system.

1. Introduction
Creating modern aircraft is a complicated, multi-iterative process. Currently, the development of aviation projects is carried out using the modeling method. After a set of models of individual systems and their properties is developed, it is possible to determine the appearance of the designed device [1–3].

Air conditioning system (A/C system) is one of the key units of the aircraft environmental control systems, which main task is to create conditions on Board the aircraft for normal human activity in flight. Any A/C system is a complex distributed multi-channel pneumatic system, the required operation of which is provided by the operation of the flow controller and the temperature of the supplied air. The A/C system consists of air extraction and cooling units, water extractors, filters, distribution equipment (adjustable valves, distributors, taps, etc.), connecting pipelines, etc. [4–6].

Part of the total energy is lost for this network irrevocably that goes to overcome the forces of hydraulic resistance that occurs when real gas moves through pipes and channels. The value equal to this irrevocable loss of total energy in this section is called the hydraulic resistance of the section [7].

For a comprehensive assessment of the system, it is important not only to know the main characteristics of the units but also to be able to calculate the hydraulic losses (the energy required to overcome the forces of hydraulic resistance) in each section.

One of the key units is the adjustable valve, which is designed to regulate the temperature of the air supplied to the cabin by regulating the flow of mixed air in the hotline of the pipeline. In addition to
the main characteristic (flow rate), an important parameter for selecting these aggregates is hydraulic resistance for different closure angles [4].

2. Method for calculating hydraulic losses

The method of estimating section hydraulic resistance by using the coefficient of hydraulic resistance ($\zeta$) is widely known. This coefficient is defined as the ratio of the total pressure loss at the area ($\Delta P$) to the value of the dynamic pressure at the area:

$$\zeta = \frac{\Delta P}{(\rho \cdot \frac{w^2}{2})}$$

or

$$\Delta P = \zeta \cdot (\rho \cdot \frac{w^2}{2}) = \zeta \cdot (\frac{\rho}{2}) \cdot (\frac{Q}{F})^2 = \frac{\zeta \cdot \rho}{2 \cdot (\frac{G}{F})^2}$$

where $\rho$ — gas density in the investigated area, kg/m$^3$; $w$ — velocity of the gas flow, m/s; $Q$ — the volumetric flow rate of gas, m$^3$/s; $F$ — a cross-section of the area, m$^2$; $G$ — mass gas consumption, kg/s.

The values of the coefficient $\zeta$ for different types of hydraulic resistance sections are given in resource books or can be determined as a result of an experiment [7–9].

Along with using calculation methods based on criteria relations for hydrodynamics, it is advisable to develop calculation methods that use modern CFD (Computational fluid dynamics) codes. The completeness of the description of processes with such codes makes it possible to optimize the design and technological solutions, reasonably build physical modeling objects and experimental programs that are necessary for validating the calculated codes [10, 11].

In addition, also, when solving problems of hydraulics by using methods of computational hydrodynamics, it is possible to take into account the geometric features of the design, i.e. the calculation model takes into account the three-dimensional design of the calculation object [12].

Using modern design methods allows you to create a model of the system itself, and simulate its operation, thereby speeding up the process of creating a new aircraft, as well as optimize the cost of design, testing, and certification [13, 14].

3. Source data

The initial data for the calculation is the geometry of the valve and the parameters of the environment. The valve geometry is shown in Figure 1, the diameter of the flow area 80 mm. Environmental parameters: the working environment is air; temperature is 200 °C; pressure is one atm. Calculations of the open valve for the reference pressure of 5.5 atm were also made.

![Figure 1. Three-dimensional model of the valve (pipeline diameter 80 mm). The numbers in figure 1 indicate 1 — the drive, 2 — the transmission mechanism, 3 — the connecting flanges, 4 — the airflow channel, 5 — the rotary part of the valve.](image)

The calculation model is obtained from a three-dimensional model of the valve by “filling” the internal space of the valve elements with the environment and extending the channel to the sides of the air inflow and outflow on 10 diameters. Figures 2 and 3 show the design model of the valve, the angle of rotation of the flap is 30 degrees from the closed position.
Figure 2. Geometry of the flow part of the valve.

Figure 3. Geometry of the valve flow part (longitudinal section of the calculated model).

Figure 4. Computational mesh for the valve model.

The ANSYS CFX software package was used for the calculation. Ansys CFX is a software package that allows us to solve several problems in the field of computational fluid dynamics. The program uses the advanced algebraic multigrid conjugate solver. In the model settings, the solution of multiphase flows, combustion problems, rotary machines, chemically reacting mixtures, radiation, and a combination of conjugate calculations can be enabled [15–17].

4. Description of the selected turbulence model and environment parameters

The Shear Stress Transport (SST) turbulence model is used to model the flow inside the valve.

The SST turbulence model is a model for transferring tangent stresses. It is used when a good solution is required in the layer near the wall, which is important for calculating heat transfer.

The equations for the kinetic energy (k) and the turbulence frequency (ω) have the form:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_{k3}}) \frac{\partial k}{\partial x_j}\right) + P_k - \beta' \rho k \omega + P_{kb}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho U_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_{\omega2}}) \frac{\partial \omega}{\partial x_j}\right) + (1 - F_1) 2\rho \frac{1}{\sigma_{\omega2} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \alpha_2 \frac{\omega}{k} P_k - \beta \rho \omega^2 + P_{\omega b},
\]

where

\[
arg_1 = \min\left(\max\left(\frac{2 \sqrt{\nu}}{\beta' \omega y}, \frac{500 \nu}{5^2 \omega}, \frac{\alpha \omega}{k} \frac{\partial k}{\partial \omega}, \frac{4 \rho k}{CD_{kw} \sigma_{\omega2} y^2}\right)\right),
\]

\[
CD_{kw} = \max(2\rho \frac{1}{\sigma_{\omega2} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10})
\]

where

- \(\gamma\) — distance from the wall, m;
- \(\alpha_2, \beta_2, \sigma_{\omega2}, \sigma_{\omega2} — constants, \alpha_2 = 0,44; \beta_2 = 0,0828; \sigma_{k3} = 1; \sigma_{\omega2} = 1/0,856.\)
The SST model uses the “k-ω” model for the wall region \((F_1=1)\) and the “k-ε” model for the external flow \((F_1=0)\) \([15, 18]\).

When modeling the near-wall flow, the following wall functions are used: standard, scalable, and automatic. For sufficient accuracy in describing the flow near the wall, a boundary layer (15–20 cells, with a growth coefficient of 1.2) was set, the first cell thickness of which corresponds to the value \(y^+=10\) \([19, 20]\).

5. Results of calculation

The hydraulic resistance of the valve at a temperature of 200°C is summarized in the table 1 and graph (Figure 5).

| Flap rotation angle, degrees | ΔP, Pa |
|-----------------------------|--------|
| 25                          | 420000 |
| 30                          | 250000 |
| 35                          | 165000 |
| 40                          | 100500 |
| 45                          | 59811  |
| 50                          | 33306  |
| 55                          | 18749  |
| 60                          | 11033  |
| 65                          | 6692   |
| 70                          | 4286   |
| 75                          | 2857   |
| 80                          | 2100   |
| 85                          | 1633   |
| 90                          | 1392   |

Figures 5, 6 show the results of calculations for angles of flap rotation 90° and figures 7, 8 for 75°. The drawings also contain information about the distribution patterns of excess static and total pressure, velocity, velocity vectors, and flow lines.

(a)  
(b)  

**Figure 5.** Pressure distribution, angle of flap rotation 90° (a) — excessive static pressure; (b) — excessive total pressure.
Figure 6. Distribution of velocity, angle of flap rotation 90°. (a) — velocity, (b) — velocity vector, (c) — flow lines.

Figure 7. Pressure distribution, angle of flap rotation 75°. (a) — excessive static pressure, (b) — excessive total pressure.
Figure 8. Distribution of velocity, angle of flap rotation 75°. (a) — velocity, (b) — velocity vector, (c) — flow lines.

6. Conclusion
The method for calculating the hydraulic resistance of the valve described in this paper can also be applied to other air conditioning units. After performing all calculations and validating them, the found values of hydraulic resistance should be included in the general model of the A/C system, thus, creating a complete and reliable picture of the pressure distribution in the system.

References
[1] Smagin D I, Satin A A, Saveliev R S, Starostin K I, Suzdaltseva D S and Serebryansky S A 2018 The concept of the creating an integrated mathematical model of the aircraft including the main onboard systems J. Quality and life 4 (20) 397–403
[2] Aircraft engineering ed M A Pogosyan. 5th ed., revised and enlarged edition (Moscow Innovative mechanical engineering) p 864
[3] Pogosyan M A, Savelievsikh E P, Shagiliev R M and Kozelkov A S 2013 Problems of atomic science and technology Ser. Mathematical modeling of physical processes 2
[4] Voronin G I 1973 Air conditioning systems on the aircraft: textbook (Moscow: Mechanical Engineering)
[5] Kravchenko A G, Shilova A K, Tamba-Tamba V P and Ozersky AI 2017 Review of the main units of the air conditioning system of the aircraft J. Technics. Technology. Engineering 3 24–7
[6] Akopov M G, Bekasov V I and Dolgushev V G and others 2005 Aircraft equipment systems: Textbook for C40 students of the higher technical educational institutions ed A M Matveenko and V I Bekasova 3rd ed. (Moscow: Mechanical Engineering)
[7] Idelchik I E 1992 Handbook of the hydraulic resistance ed M O Steinberg. 3rd ed., revised and enlarged edition. (Moscow: Mechanical Engineering) p 672
[8] Altushul A D, Kalitsun V I and Mayranovsky F G etc. 1976 Examples of the calculations for hydraulics: Textbook (Moscow: Stroyizdat) p 256
[9] Kireev V, Nizamova A and Urmancheev S The hydraulic resistance of thermoviscous liquid flow in a plane channel with a variable cross-section *J. of Ph.: Conf. Series* **1158** 3

[10] Borzenko E I, Rylkova K E and Schrager G R 2019 *Bulliten of Tomsk State University Mathematics* Numerical study of the characteristics of the flow of a non-Newtonian fluid in demand with a sudden contraction **58** 56–70

[11] Afremov D A, Afonin S Yu, Zakharov A G, Nedayvozov A V and Smirnov V P 2015 Verification of three-dimensional CFD codes for calculating the forced flow in full assemblies of fast reactors *NIKIE Annual Report – 2015* Ed E O Adamova. (Moscow) p 154

[12] Anikeev A A, Molchanov A M and Yanyshev D S 2010 *Fundamentals of computational heat transfer and hydrodynamics. Textbook* (Moscow)

[13] Federal Aviation Regulations. Part 25 Airworthiness Standards: Transport Category Airplanes

[14] Federal Aviation Regulations. Part 29 Airworthiness Standards: Transport Category Rotorcraft

[15] Ansys help, ansyshelp.ansys.com

[16] Methods for the Design of Modern On-Board Systems of Advanced Aircraft. 2019 IEEE 10th Int. Conf. on Mech. and Aerospace Eng. (ICMAE) 2019 DOI: 10.1109/ICMAE.2019.8880986

[17] Molchanov A M 2019 *Numerical method for solving the Navier-Stokes equations* OSF Preprints, Web

[18] Badennikov A V, Piralishvily S A and Guryanov A I 2018 Results of Numerical Modeling of Combustion Processes in a Vortex Chamber *MATEC Web of Conf.* **209** 00023

[19] Benner P, Gugercin S and Willcox K 2015 *A survey of projection-based model reduction methods for parametric dynamical systems* SIAM Rev. **57**

[20] Vreman A W, Sandham N D and Luo K H 1996 Compressible mixing layer growth rate and turbulence characteristics *J. of Fluid Mech.* **320** 235–58