RIGA TECHNICAL UNIVERSITY

Natalia Sidenko

Optimization of Parametres of the Aerodynamic Stand Meant For Free Flight of the Human In Air Jet

PROMOTION WORK

Riga 2012

RIGA TECHNICAL UNIVERSITY
Faculty of Transport and Mechanical Engineering
Aeronautics institute

Natalia SIDENKO

PhD study programm Transport „ Aviation transport" doctoral candidate

OPTIMIZATION OF PARAMETRES OF THE AERODYNAMIC STAND MEANT FOR FREE FLIGHT OF THE HUMAN IN AIR JET

Promotion work

Scientific Supervisor
Dr. habil. sc. ing., profesors
V. USHAKOV

Riga 2012

Šis darbs izstrādāts ar Eiropas Sociālā fonda atbalstu projektā «Atbalsts RTU doktora studiju īstenošanai».

This work has been supported by the European Social Fund within the project «Support for the implementation of doctoral studies at Riga Technical University».

Эта работа выполнена при содействии Европейского социального фонда в рамках проекта «Поддержка развития докторантуры РТУ».

ANNOTATION

In the introduction are considered general lines for the development of constructions of stands for sports flights in vertical air jet. Special attention is paid to the modern constructions of aerodynamic stands, which vary in design, size, range of speeds, and ways of creating a stream. Reviewed the principle of operation of the simplest constructions, intended to create a vertical air stream of large diameter. Highlighted main advantages and disadvantages of modern constructions. Performed the analysis of modern engineering methods of calculation of the elements of stands structures. Formulated the relevance of research, aims and objectives of the dissertation.

In the first chapter is formulated basic method of computer simulation and calculation of parameters of large diameter vertical air jet generators (or open wind tunnels generating the air jets). The method includes main steps for this class and contains the main characteristic details inherent in this task. The method enables parametric computer calculations which pretty accurately model researched physical phenomena and allow defining their interesting the practice characteristics. Calculation of developed aerodynamic stands must be done by means of the known engineering CAD/CAE programs that ensure compliance with all the necessary work to be done in a reasonable time period.

In the second chapter developed method of computer simulation and optimization of geometric and aerodynamic characteristics of single elements of aerodynamic stands.

The method allows using 3D parametric computer simulation packages SolidWorks and CFDesign to create different 3D modifications of geometric models and perform their virtual blowing reaching required geometric and aerodynamic parameters of developed stands. Method allows relatively quickly to obtain results, satisfactorily corresponding with the data of natural experiment or known theoretical calculations confirming, the adequacy of the method.

In the third chapter developed the computer simulation and optimization method of geometric and aerodynamic parameters of different modifications of aerodinamics stends (single screw and multi-screw). On the basis of the developed method was created a universal air jet generator designed to create conditions for the free flight of human in the vertical air stream. Universal generator can be used for both mobile and stationary aerodynamic stands. Computer simulation of complex volumetric constructions of wind tunnel with symmetrically placed fans allowed resolution of a number of tasks such as connecting the sections of air channels, taking into account to the necessary narrowing and widening angles, smooth transitions and turns of channels, pairing a several channels, taking into account the restrictions of curvature and rotation of the corners.

Using developed methods have been developed computer models on which base were designed and installed single screw and multi-screw aerodynamic stands.

CONTENTS

INTRODUCTION

1. Overall evolution of constructions of simulators for sport flying in a vertical air jet.

Currently flying in vertical air flow of aerodynamic stand is an integral part of the training process of most skydiving teams.

On ground located aerodynamic stand for human flight in a vertical air jet simulates the feelings what is experiencing the skydiver in free flight. If the time of free flight of the skydiver is about one minute, meanwhile the free flight in the on ground located stand, without taking into account economic considerations, is limited only by the human physical abilities.

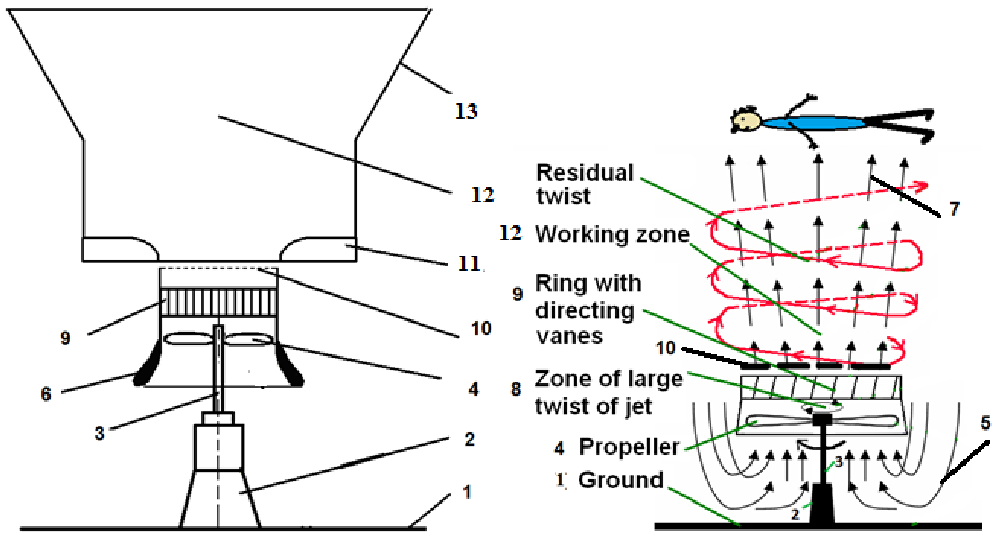
Stands of this type are used as:

- ground stands for simulation of long skydiving (Accelerated Free Fall) skydivers training program);
- training of paratroopers and units of special forces;
- testing and experimental blowing of equipments and devices;
- new type of sport "Bodyflyght";
- sport and entertainment of the public.

The principle of operation of one of the most common aerodynamic stands with open aerodynamic circuit and an open working section (area) is shown in Fig. 1.

Aerodynamic stand is installed on the ground 1 (in some cases on the mobile automotive platforms). The air flow is generated by a large diameter (~ 3 ÷ 4.5m) fan or propeller (4), which is mounted on a vertical shaft (3) of the stationary engine (2). Propeller 4 generates in front of itself zone of low pressure and forms an air flow (Fig. 1.b. Scheme of the jet), which inside the input device-6 moves from the bottom up and reaches the propeller 4 (Fig. 1. a, b). Under the action of the propeller is formed flow 7, which moves up with a great vertical velocity and a certain angular velocity 8 (flow swirling, large twist of jet) relative to the vertical axis of the propeller. For generation in the working section (area) air jet without swirling and with a sufficiently uniform profile of vertical velocity, and to reduce the "failure" of velocity close to the jet axis, behind the rotating screw is coaxially installed ring with directing vanes (rectifying device) 9.

Construction of rectifying device largely is determined by the geometry of the propeller and its angular velocity. At some distance from rectifying and smoothing device is installed guard net (safety grid) 10, designed to prevent the fall of the athlete on the propeller or fan. Along the perimeter of the grid usually are positioned airbeds (airbags) 11. For security purposes, the lateral surface of the aerodynamic stand behind the open working section (area) 12 is limited be guard net (safety grid) 13. It should be noted that the presence or movement of the athlete in the working section of the jet has an effect on its characteristics and, consequently, the conditions of free flight.



a) b)

Fig. 1. Scheme of the open circuit aerodynamic stand with an open working section (area, zone): a) Schematic design; b) Scheme of generation of a vertical air jet.

Approximate value of the flow velocity ***V*** (m/s) (1) in the working area, which is required to hold the athlete in a vertical air jet is determined by the known expression of the drag force ***Fx*** under condition of its equality to weight of the athlete ***X= mg*** (m - mass of a body, g - acceleration of gravity). Flow pattern of force Fig.2.

$$V = \sqrt{\frac{2 \cdot m \cdot g}{\rho \cdot C_x \cdot S}} \quad , \tag{1}$$

Where ***S*** - cross-sectional area of the athlets body (m^2), $\boldsymbol{C_x}$ - coefficient of drag force ***Fx*** (N), $\boldsymbol{\rho}$ - density of air under normal conditions (kg/m^3).

Considering that ***Cx*** = 0.04 - 0.08, ***m*** = mass of an athlete is 80 kg, the acceleration of gravity g = 9.81 m^2/s, cross-sectional area of the grouped athletes body ***S*** ~ 0.8 m^2, the value of the required velocity ***V*** will be ~ 60 m/s.

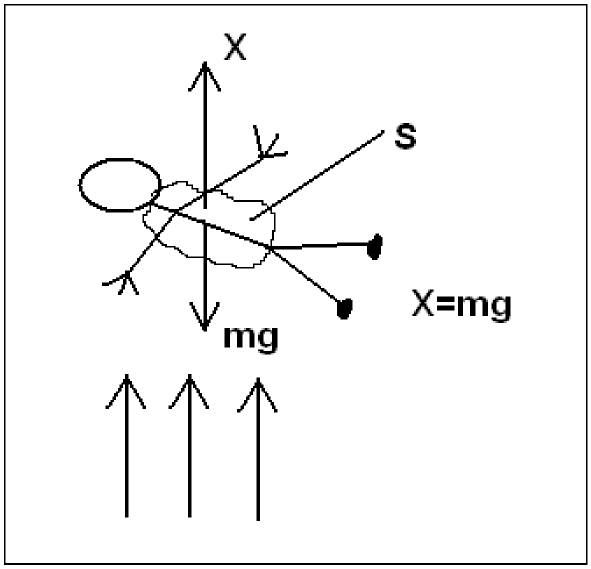


Fig. 2. Flow pattern of force

In the world there already is a large number of aerodynamic stands for free flight of human, which vary in design, dimensions, range of velocities, and ways of generation of a jet.

Known aerodynamic stands can be with a closed aerodynamic circuit and closed working section, with open aerodynamic circuit and closed (i.e., limited by side walls) or an open working section . Stands with an open aerodynamic circuit and open working section should create a long range vertical jet (altitude of the flight can reach 20-25 meters from the exit section of the tube). In the cross-sections of working section of a jet flow must be sufficiently uniform and without swirling. Average vertical velocity in a working section is approximately 60 -70 m/s at standard atmospheric conditions.

Stationary aerodynamic installations with closed circuit and closed working section.

Construction of these aerodynamic stands is close to the so-called vertical wind tunnels [1-4] and they represent long-term capital structures having all the characteristic elements of the closed-type wind tunnel with a vertical working section.

There are known stationary closed type installations like "Vertical Sky Venture" (USA, Florida) with a vertical flow, which is generated in a closed (i.e., limited by side walls), vertical working section of a vertical closed wind tunnel [5-6].

Usually such installations are made as one, two or more closed (reverse) channels (Fig. 3, 4.). To reduce the level of flow turbulence, fans are placed in the upper part of the structure (Fig. 3.), and to reduce hydraulic losses in the area of turning of channels are installed guide blades.

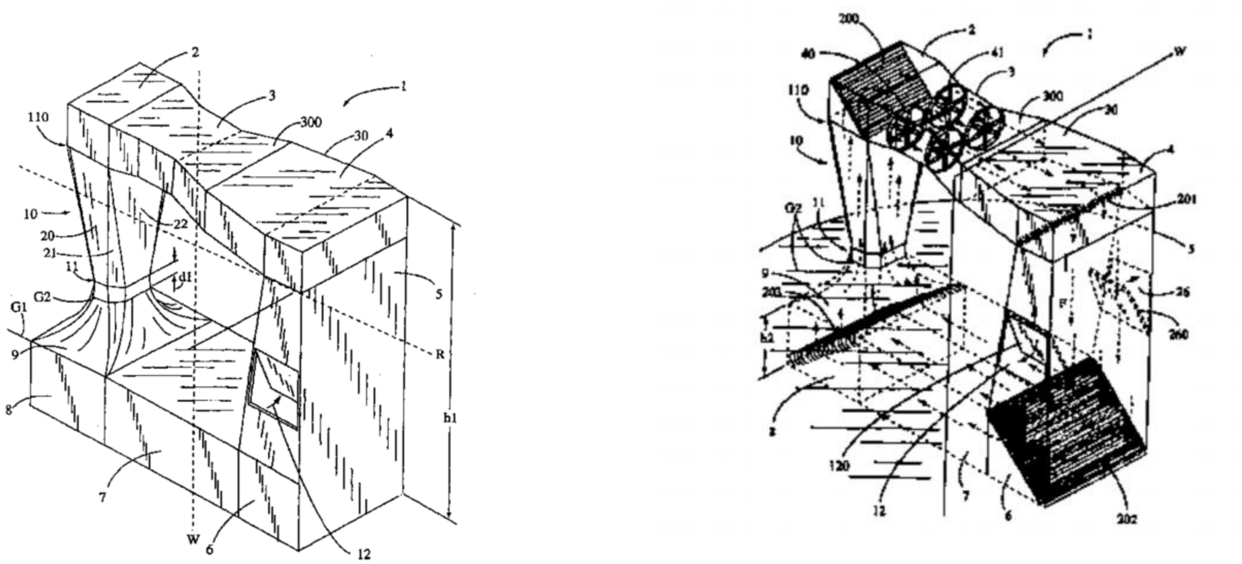


Fig 3. Closed single-channel vertical wind tunnel SKY VENTURE.

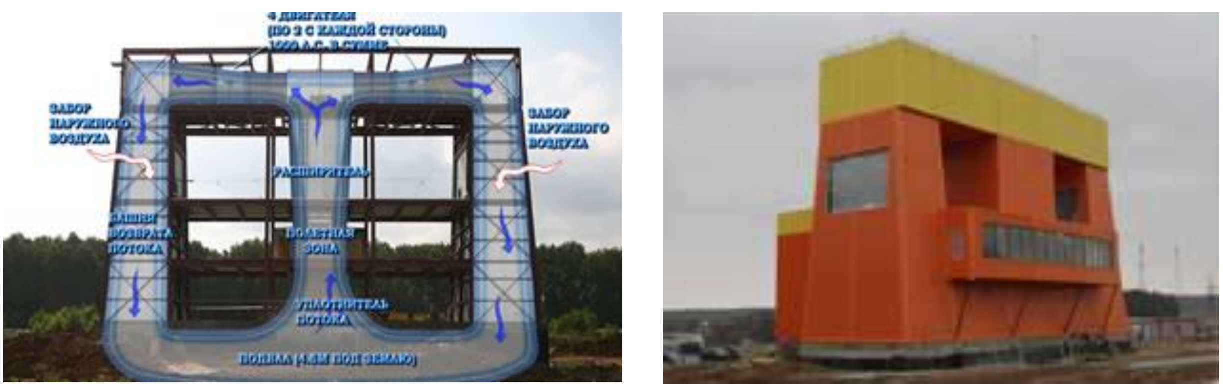


Fig.4. The closed type vertical wind tunnel for paratroopers training in the "Sky Venture"

Main advantages of closed type vertical wind tunnels:

- In a closed wind tunnel flow circulates in a closed circuit, that allows partial use of mechanical energy what flow has already acquired;
- With the same dimensions of working area and with the same engine power in the working section with a closed flow velocity is higher than that in tube with non-closed flow

Disadvantages:

- In case of horizontal location of the fan or aerodynamic screw axis, for generation of vertical jet it necessary to turn the flow be find the fan at 90˚ by means of special channels or guide blades, what causes the asymmetry of the jet -velocity profile and the additional energy losses;
- Space for flights is limited by solid side walls of working section of the tunnel;
- The inability to demonstrate sport shows with group flights for a large number of spectators, such as stadiums, fairs, etc.;
- High cost of creation of such installations and the lack of possibility of their rapid assembly and disassembly, when it is needed to change the location of the installation.

Stationary installations with open circuit and closed working section.

Stationary closed type installations (Fig. 5, 6) are large temporary structures, which have all characteristic elements of wind tunnels with open circuit and closed work section. Fan, which is creating the flow, can be placed as in front of working section (Fig. 5) and as well behind it (Fig. 6) [7].

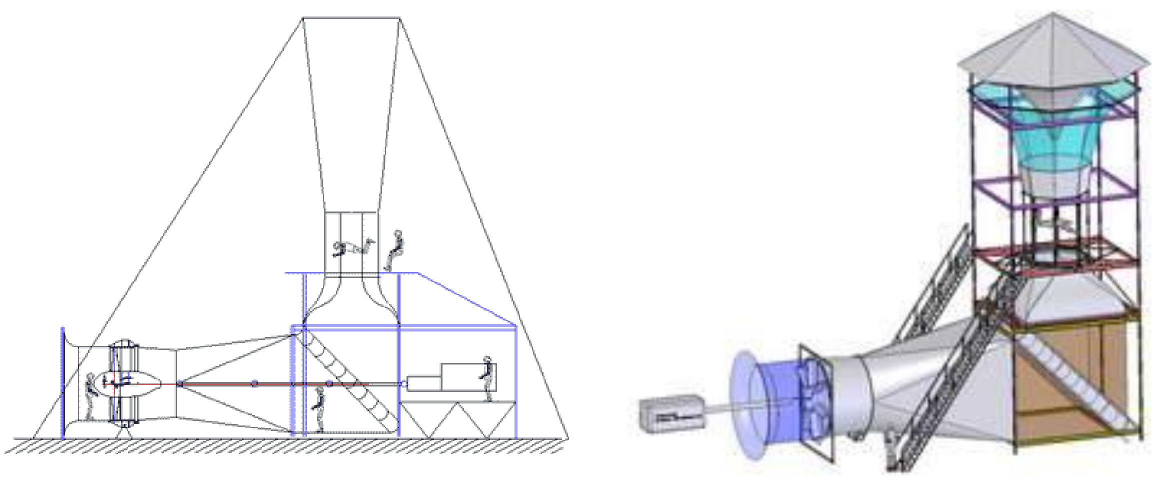
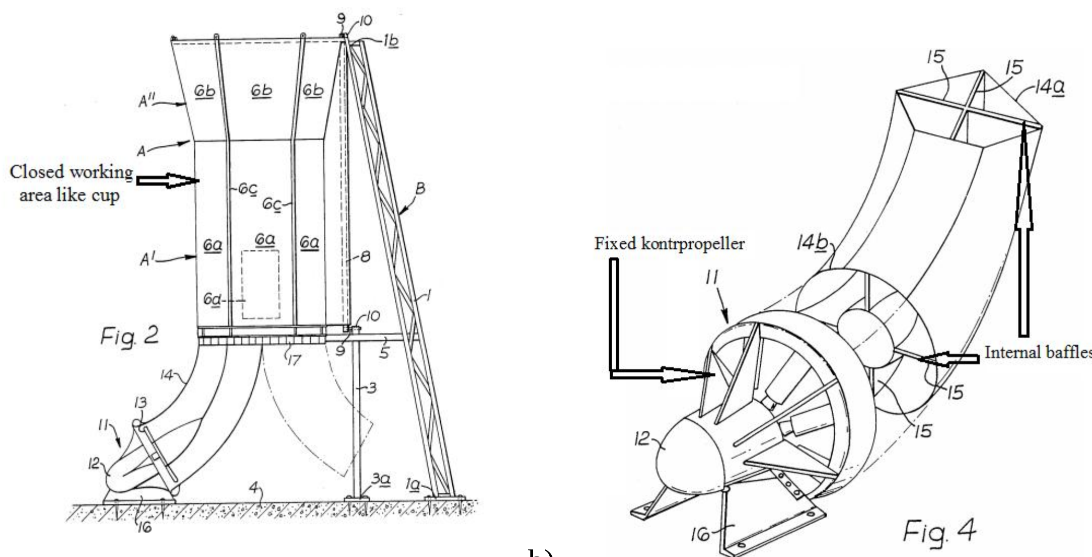


Fig. 5. Model of a vertical sport aerodynamic stand with open circuit and closed working section AST-1, Russia, Samara.

Fig. 6. The aerodynamic stand with open circuit and closed working section Sky Venture "(USA, Florida).

There are also known constructions with the fans systems with a closed working section like cup [8] Fig. 7.a. The main feature of this type of constructions is that all the fans are mounted in the cylindrical side wall of the inlet chamber of jet. In front the fan is installed a fixed kontrpropeller Fig.7 b , which must swirl the flow in front of screw to the opposite direction so, that flow behind the screw had only axial velocity. For smooth flow entry into the inlet chamber of jet, there is installed the fairing. Chamber inlet of the jet is divided by four separation walls, which play a role of guide vanes (baffles). The similar scheme is implementing in the closed wind tunnels with an open working section [4].



a) b)

Fig. 7. a) The aerodynamic stand with a system of fans and with a closed working section in the form of a cup; b). Is a perspective view of a motor/fan unit forming part of the simulator.

Main advantages of installations with open circuit and closed working section:

- Relatively low costs comparing with the closed type tunnels;
- Possibility of assembly and disassembly of installation if necessary, change the of its location;
- For stands with the system of fans, which are installed in the profiled inlet devices, hydraulic losses in the gas-dynamic channel are lower, and the volumetric flow rate and flow velocity behind the screw is higher.

Disadvantages of considered installations:

- In non-closed tunnels is ongoing continuous change of gas flow, which should accelerate till design speed. Therefore energy consumption in such tunnel is higher than in a closed installation, where it is only necessary to maintain the movement of the circulating air;
- Lack of space for flying and the inability to demonstrate sport show with the group flights;
- For the version of installation with the system of fans in the absence of a specially profiled inlet part of supply channel, may cause appearance of flow separation near its inlet edges and a significant increase of energy losses in the system.

Stationary and mobile installations with not closed circuit and open work-section.

There are known stationary open type aerodynamic stands with the free vertical air jet flowing into the atmosphere whose work section of the jet, in general, is not limited by side walls and the upper air permeable grid (see, e.g., Fig. 8. and mobile Fig .9.)

Aerodynamic configuration of such facilities is open type. These aerodynamic stands are mainly intended for flight enthusiasts and sports shows, as they allow carry out flights in the working section of the free vertical air jet at height approximately (1 - 20) meters from the bottom of the air permeable protective grid. They can be quickly disassembled, transported and installed at the new location.

The characteristic structural elements of a system of generation of vertical air jet in open type aerodynamic stands are:

- Aerodynamic screw or fan connected to the engine;
- Channel for supplying of air to the screw or fan (in some prototypes it is absent);
- Off-take canal inside which, in general, are installed the guide vanes and rectifying apparatus for elimination of flow swirling behind the screw (fan) and for formation of a

uniform vertical air flow at the outlet from the bottom of air permeable protective grid, from which starts working section of free vertical air jet.

Fig. 8. Stationary aerodynamic stand with not closed circuit and open work section.

a) b)

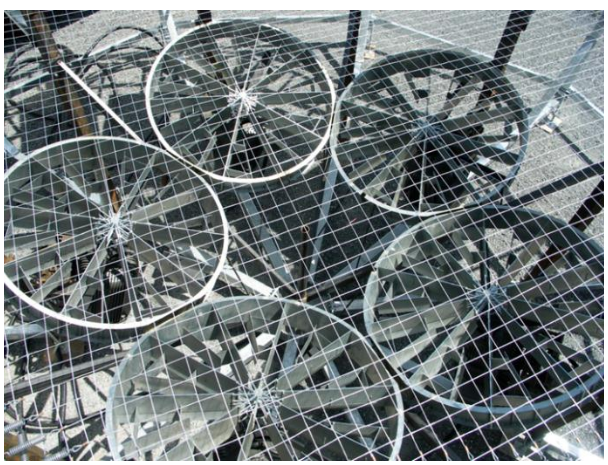
Fig. 9. a) Mobile installation with not closed circuit and closed working section;
b) Mobile units with not closed circuit and open work section.

Main advantages of stationary and mobile installations with not closed circuit and open work section are relatively small dimensions, possibility of fast assembly and disassembly, as well as their low cost.

Main disadvantages of a system for generation of free vertical air jet with an open work section are following:

- High energy consumption and a low efficiency factor due to release of the flow in motionless environment, inappropriate design and increased operating costs due to high flow energy losses as in separate elements, and in the system as a whole by combining elements into a single complex without consideration of their mutual aerodynamic influence on each other;
- Absence in a number of constructions of specially profiled inlet part of supply channel, which causes appearance of flow separation near its inlet edges and significantly increase of energy losses in the system;
- Relatively low quality of air flow in the jet because of the practical impossibility of using of known methods of improvement for stationary closed type installations (for example, through the creation of a shaped nozzle with honeycomb behind the screw or high compression of flow for smoothing of pulsations and velocity profile in the jet);
- With open work section it is easier to train a group of people, but even with located in a circle air mattresses, this zone is with increased possibility of accidents.

There are known stationary installations which are using the system of fans installed with their longitudinal axes along the radius of the circle at equal distance from the centre, with not closed circuit and open work section Fig. 10.



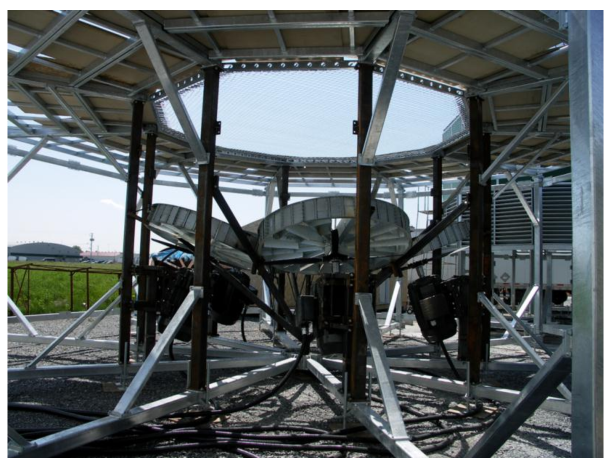


Fig. 10. Stationary installation with the system fans and not closed circuit and open work section.

In the Fig. 10. Is shown the construction design, consisting of five symmetrically installed propellers in front the short cylindrical channels with rectifying vanes whose axes are directed at an angle of 80º to the vertical axis of the jet. Fans create five free colliding and should create jets the required large diameter vertical jet.

The main advantage of this type of installation is an opportunity of application of small-diameter propellers, which reduces the total power losses and also reduces the aerodynamic noise. The failure of one of the fans does not create a critical situation in process of exploitation.

Main disadvantages of the construction - big pressure losses due to colliding of jets and lack of guiding gas-dynamic channel forming a vertical air jet behind the aerodynamic screw:

From the above it follows that each of the considered types of aerodynamic stands for human flight in a vertical air jet has its own characteristic features, advantages and disadvantages.

It is known that for the training of professionals (military special purpose units, troops, athletes - parachutists, etc.), are more preferable stationary aerodynamic installation with closed circuit and closed working section, which have a high level of flow uniformity and low turbulence, e.g. reproduce well the natural flow.

Aerodynamic stands with open circuit, basically, are intended for entertainment. Their advantages are lower capital costs and possible mobility. At the same time, at comparable size of working section with closed circuit installations, they require more power of power installation and working creating higher aerodynamic noise. It follows that these installations without special sound insulation cannot be mounted in a small city park, but only at a considerable distance from the residential area.

2. Engineering calculation methods of elements constructional for stands for human flight in a vertical air flow

As noted in first paragraph of this chapter, simple aerodynamic stand consists of following main elements: inlet device, aerodynamic screw, rectifying apparatus. Each of these elements can be calculated using the known engineering calculation methods. However, in these calculations most cases are used empirical data without taking in to account mutual influence of aerodynamic elements against each other. Below are analysed several well-known and widely used engineering calculation methods that can be used for engineering of elements for stands for human flight in a vertical air flow.

Calculation of aerodynamic characteristics of screw.

Modern aerodynamic theories of screw can be classified into five groups [9-16].

- Theory, which makes it possible to establish a relation between traction force and power on the one hand, and velocities in the jet - on the other. However, these theories can not relate geometrical dimensions of blades with velocities in the jet and according to them it is impossible to design a screw.

- Isolated blade element theories. These theories also can not be used for engineering of screws.
- Theory, relating the first two, for example, the theory of Sabine - Yuryev, which establish relation between flow and shape and size of blades.
- Vortex theories of screw. For example, the vortex theory of N. Zhukovsky, which defines the relationship between circulation and velocities in the jet caused by the screw, circulation and design parameters of blade and between the circulation, power and traction of screw. Zhukovsky theory is widespread and allows to construct screws and make calculations of their characteristics with sufficient accuracy for practical purpose.

From the said follows that first two groups of theoretical calculations of screw can serve only for approximate estimations and they have a practical application for some particular tasks. The next two groups are used for engineering of new screws and a more deeper investigation of its work.

To estimate the parameters of the air jet behind screw (for example, axial and circumferential velocity behind screw and in the plane of rotation), can be used theory of ideal propeller and ideal screw. For this must be known screw diameter ***D***(m), engine power ***N*** (HP)and momentary rotation speed of screw $\boldsymbol{n_s}$ (RPS). According to known circumferential and axial velocity can determine the inclination angle of flow behind screw to its axis, which is required for the designing of rectifying devices.

Fig.11. illustrates the theoretical form of jet, formed behind a ideal propeller.

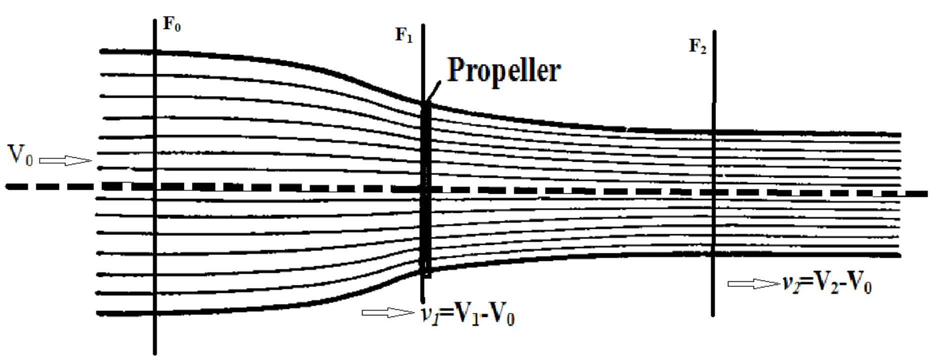


Fig.11. Elementary circular jet of ideal propeller.

Power $\boldsymbol{T}$ and traction force $\boldsymbol{P_0}$ of ideal propeller working on place are determined by following equations :

$$T = \frac{F \cdot \rho}{4} v_2^3 = \frac{\pi \cdot D^2}{16} \rho \cdot v_2^3 \qquad (2)$$

Were, area of a jet $\boldsymbol{F=\pi D^2/4}$(m^2) , $\boldsymbol{D}$- propeller diameter (m), $\boldsymbol{\rho}$- density of air (kg/m^3), $\boldsymbol{v_2}$ –increment velocity (m/s), $\boldsymbol{v_2=V_2-V_0}$, $\boldsymbol{V_0}$- velocities before propeller (for propeller working on place V_0=0), $\boldsymbol{V_2}$- velocities in the jet well over propeller. So, v_2= V_2 and v_1= V_1

$$P_0 = \frac{\pi \cdot D^2}{8} \rho \cdot v_2^2 \qquad (3)$$

Particularly velocity v_2 behind propeller for can be determined by following equation (4) [9], where power engine $\boldsymbol{N}$ is expressed in HP.

$$v_2 = 14{,}5\sqrt[3]{\frac{N}{D^2}} \qquad (4)$$

Accordingly longitudinal velocity $\boldsymbol{v_1}$, induced in plane of screw, is $\boldsymbol{v_2 = 2v_1}$.

Velocity behind screw $\boldsymbol{V_2}$ at its operation on place can be determined from the chart shown on Fig. 12.

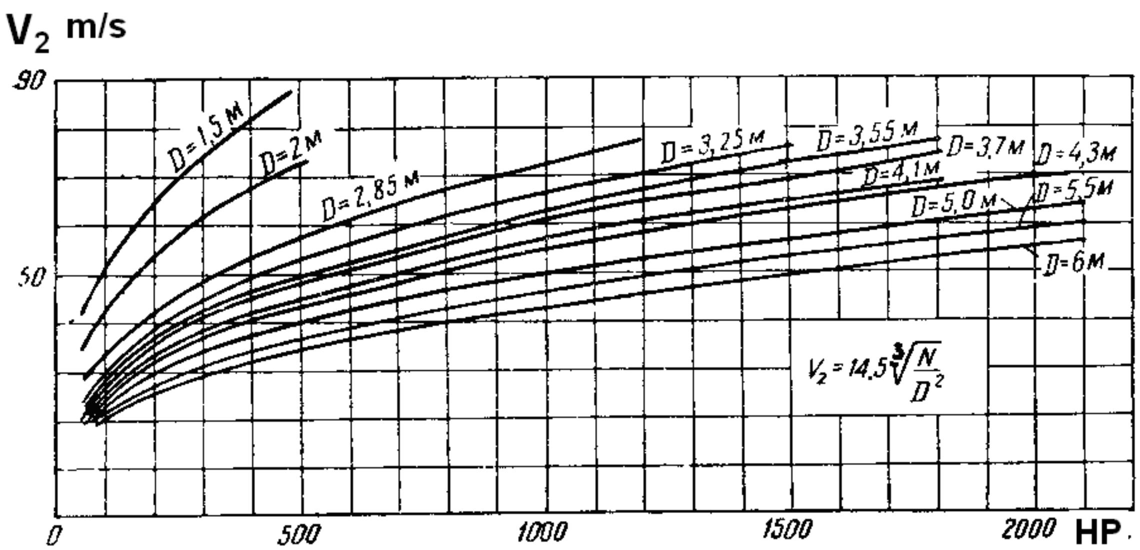


Fig. 12. Chart to determine velocity behind screw at its operation on place.

As opposed to theories of ideal propeller in to theories ideal airscrew has induced angular velocity ω Fig.13.

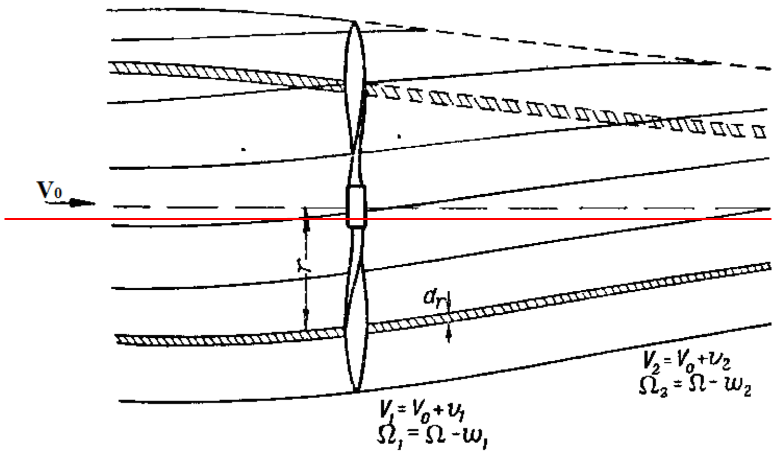


Fig.13. Elementary circular jet of ideal screw.

Induced angular velocity behind screw can be calculated by formula (5) [9] .

$$\omega_1 = \frac{\Omega}{2}\left(1 - \sqrt{1 - \frac{8 \cdot B \cdot V_0^2}{D^2 \cdot \Omega^2}}\right) \tag{5}$$

Where Ω-angular velocity of screw rotation $\Omega = 2\pi n_s$., B - is the load factor, which depends on compression of jet. When working on place factor B is largest. Were B=0, Po=0 – don’t compression of jet

Respectively

$$\omega_2 = 2\ \omega_1 \tag{6}$$

Jet swirling angle ε_2 can be expressed:

$$tg\varepsilon_2 = \frac{\omega_2 \cdot r}{V_2} \tag{7}$$

Vortex theory of Zhukovsky allows determination of relation between induced velocity and value of circulation *Γ* (8), (9).

$$v_1 = \sqrt{\frac{k \cdot \Gamma \cdot \Omega}{4\pi}} = \frac{1}{2}\sqrt{\frac{k \cdot \Gamma \cdot \Omega}{\pi}} \tag{8}$$

$$v_2 = 2v_1 = 2\sqrt{\frac{k \cdot \Gamma \cdot \Omega}{4\pi}} = \sqrt{\frac{k \cdot \Gamma \cdot \Omega}{\pi}} \tag{9}$$

where *k* - number of blades, *Γ* – circulation (m^2/s).

Note that for screws of Zhukovsky these formulas give sufficiently accurate results, but for others - more approximate.

For aerodynamic stand which creates a vertical air jet is usually defined required value of velocity v_2 in the work area (velocity at the outlet section of nozzle without consideration of influance of ring and rectifying apparatus.) Formula (10) allows one to determine value of circulation at a known rotation speed of rotation of shaft screw and velocity v_2

$$\Gamma = \frac{2 \cdot \pi \cdot r^2 \cdot \omega_2}{k} \tag{10}$$

Final formulas for screw of Zhukovsky for traction and power in operation of screw on place (V_0=0) at known circulation are as follows.

$$\overline{P} = \frac{P}{2\pi \cdot \rho \cdot R^4 \cdot \omega^2} = \frac{2}{\pi^3} \cdot \overline{\alpha} = \overline{\Gamma}(0{,}9375 - 2{,}77\overline{\Gamma} - 1{,}5\mu \cdot \overline{V}_1); \tag{11}$$

$$\overline{T} = \frac{75N}{2\pi \cdot \rho \cdot R^5 \cdot \omega^3} = \frac{2}{\pi^4}\overline{\beta} = \overline{\Gamma}(0{,}9375 \cdot \overline{V}_1 + 0{,}656\mu - 1{,}5\mu \cdot \overline{\Gamma}) \tag{12}$$

Here circulation $\overline{\Gamma}$ (13) for screws of Zhukovsky thrust $\overline{\alpha}$ (14) and power $\overline{\beta}$ (15) coefficients are determined by formulas:

$$\overline{\Gamma} = \frac{\Gamma \cdot k}{4\pi \cdot R^2 \cdot \omega}; \tag{13}$$

$$\omega = \frac{2\pi \cdot n}{60} \text{ RPM};$$

$$\overline{\alpha} = \frac{P}{\rho \cdot n_c^2 \cdot D^4} \tag{14}$$

$$\overline{\beta} = \frac{T}{\rho \cdot n_c^3 \cdot D^5} \tag{15}$$

n (RPM) - number of shaft revolutions per minute; μ – value reciprocal to aerodynamic quality of profile $\frac{1}{\mu} = \frac{C_y}{C_x}$, it takes into account, mechanical losses due to friction on the screw; $\mu \approx 0.03$; $\overline{\alpha}$ and $\overline{\beta}$.

Geometric arrangement of screw is performed according to formulas (16), (17), (18) , determining angle of indraft of jet β_{Π} ,width of blade $\bar{в} = \frac{в}{R}$ and resulting axial and circumferential induced velocities $\overline{\omega}_1$

$$tg\beta_n = \frac{\overline{V}_1}{\bar{r} - \frac{\overline{\Gamma}}{r}}; \tag{16}$$

$$\bar{в} = \frac{8\pi}{K} \cdot \frac{\overline{\Gamma}}{C_y \cdot \overline{W}_1}; \tag{17}$$

$$\overline{\omega}_1 = \sqrt{V_1^2 + \left(\bar{r} - \frac{\overline{\Gamma}}{\bar{r}}\right)^2} \ . \tag{18}$$

Value of lift coefficient Cy is determined by known Cy(λ) (see, for example, [17]).

Evaluation of screw quality

Value, evaluating quality of screw, is its efficiency factor. Theoretically at operation of screw on place efficiency factor is zero, so it can not be taken as a measure of operation quality. In practice the quality of screw on place can be evaluated using ratio P_0/T_0,where.

$$\begin{cases} P_0 = \alpha_0 \cdot \rho \cdot n_s^2 \cdot D^4 \\ T_0 = \beta_0 \cdot \rho \cdot n_s^2 \cdot D^3 \end{cases} \tag{19}$$

In this case coefficients α_0 and β_0 for a given type of screw will depend only on angle of screw installation φ. A typical diagram of this dependence is shown in Fig. 14. Taking into account equations (19), expression for quality of screw becomes.

$$K = \frac{P_0}{T_0} = \frac{\alpha_0}{\beta_0} \cdot \frac{1}{n_s \cdot D} \tag{20}$$

Which implies that quality of screw depends on the value α_0/β_0 or the value n_sD, which is proportional to circumferential speed of blades. Indeed, angular velocity of blade rotation is

$2\pi n_s$, and circumferential is $2\pi n_s r$ or for the ends of blades $\pi n_s D$. Circumferential velocity U_0 of blade end will be.

$$U_0 = \pi \cdot n_s \cdot D = 0.0523 \cdot n \cdot D \tag{21}$$

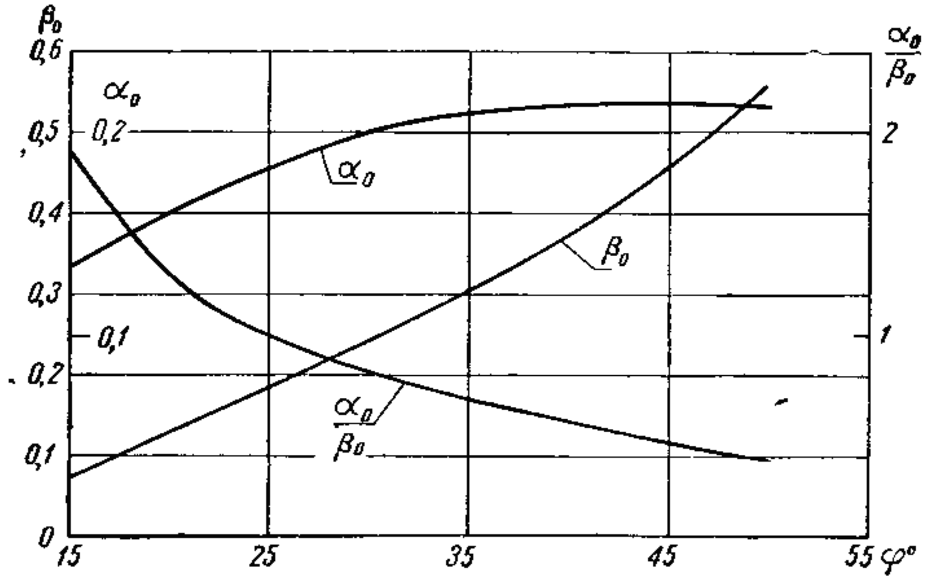


Fig. 14. The coefficients of thrust and power of screw at operation in place.

In Fig. 14 is shown curve $\alpha_0/\beta_0=f(\varphi)$. As you can see, increasing angle φ, α_0/β_0 decreases and, hence, at the same speed $U_0 = \pi n_s D$ = const according to formula (20) quality of screw R_0/T_0 increases with decreasing φ.

But more than that quality will increase, if circumferential velocity of the blade $n_s D$ will be reduced, i.e., reduced diameter or speed of screw, or both. However, decreasing the number of revolutions and diameter also power will be reduced. For in this paper investigated stands interesting was a question, how to increase quality at the same power T_0. To do this, the second formula (19) is determined $n_s D$ and put into the formula (20).

$$K = \frac{P_0}{T_0} = \frac{\alpha_0 \cdot D^{\frac{2}{3}}}{\beta_0^{\frac{2}{3}}} \cdot \sqrt[3]{\frac{\rho}{T_0}} \tag{22}$$

Consequently, quality K depends on $\alpha_0/\beta_0^{2/3}$ and $D^{2/3}$. In Fig. 15. is shown a curve $\alpha_0/\beta_0^{2/3} = f(\varphi)$, which shows that decreasing the angle of blade installation φ, $\alpha_0/\beta_0^{2/3}$ increases. Consequently, according to formula (22) quality of screw will be higher, if angle of setting φ will be smaller and diameter *D* will be larger.

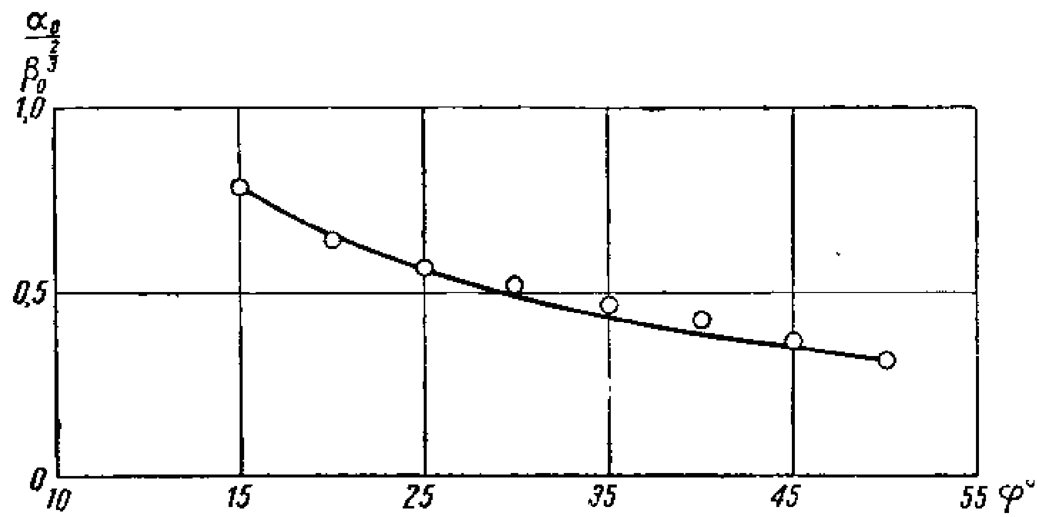


Fig. 15. Dependence of function $\alpha_0/\beta_0^{2/3}$ that characterizes operation of screw in place on angle of setting φ.

Selection of blade shape

In terms of the gaining in efficiency factor blade shape is not very important Fig. 16. Narrow blade ends for calculated advance ratio give maximum efficiency by 2-3% higher than the wide ends. [18-20].

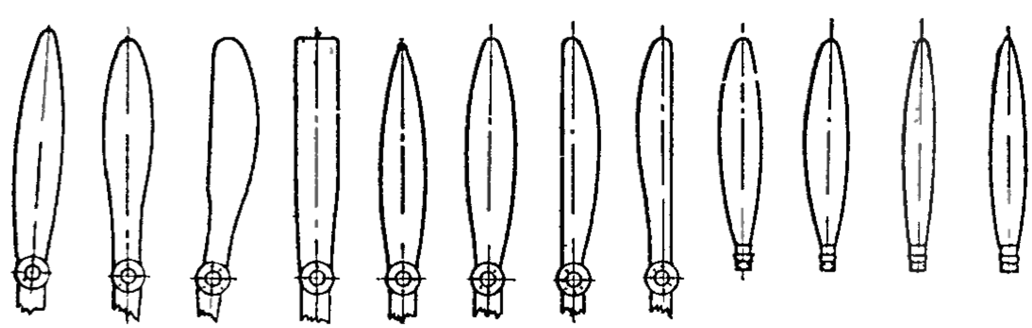


Fig. 16. Blade shape.

In the report NASA No 643 [21] are shown test results of three-bladed screws with the same diameter and different position of the maximum blade width along the radius of the screw. Tests have shown that from the viewpoint of maximum efficiency factor the best parameters has a screw, whose section of maximum width is located close to the hub (at 0,35-0,40 R), which has narrow blade ends. Normal screw has slightly smaller efficiency factor.

Quantitatively difference between efficiency factors of screws is very low and does not exceed 3%. But from the viewpoint of traction on place, blades with the narrow ends are disadvantageous (losses could reach be up to 15 - 20%). In this case it is best to use usual screws with location of a maximum width at 0,5 - 0,55 R and the wide end of the blade.

Airscrew diameter

To increase thrust and power of screw most often increase the diameter D and number of revolutions n_c. However, increasing of screw diameter increases speed of end of the screw blades to a value close to the speed of sound, what results worsening the aerodynamic characteristics of the screw [22].

Therefore, when choosing the screw diameter and speed necessary to estimate the circumferential speed of blade tips U_0 (21). Using the Mach number $M = \frac{U_0}{a}$, where *a*- is the speed of sound, we obtain

$$M_{blad} = \frac{\pi \cdot n_c \cdot D}{a}. \tag{22}$$

If $M_{blad}>M_{krit}$ where M_{krit}-critical Mach number at which on the screw arise zones at supersonic speeds, appears wave drag and significantly decreases efficiency factor. In addition, significantly increases level of aerodynamic noise.

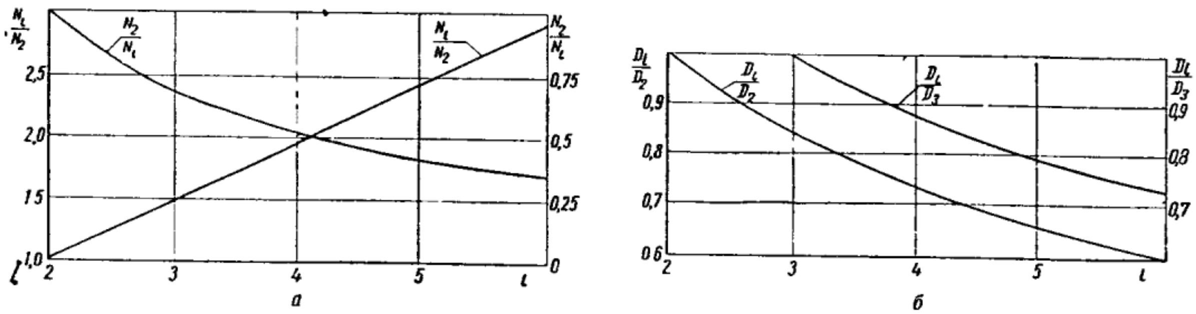
Practical limit of allowable Mach number on noise levels is $M_{blad}\leq 0,80$ (at relative thickness at blade tip $\bar{c}_{blad} \geq 10\%$). $M_{blad}\leq 0,85\div 0,9$ (at $C_{blad}\approx 6\%$). Normal value of circumferential speed at screw operation in place is $U_0=(150\div 200)$m/s.

Noise can be reduced lowering U_{blad}, that is, using slowly rotating large diameter screw.

In addition, increasing the screw diameter not always is possible due to design reasons. Increase of blade causes a significant increase in weight. Main disadvantage of screw with a large angle of setting at operation in place is that it doesnot gives opportunity to reach maximum speed and maximum engine power. Often the most appropriate way is to increase the number of blades. With increasing of speed n_c appear high cirumferencial speed, which leads to restrictions similar to those described above.

Having a graphs of two blade screws you can go to multiblade screws using approximate transition coefficients. In Fig. 17a is given the ratio of the power, absorbed by screws with a different number of blades, to the power of two blade screws. As we can see, at transition from two-blade screw to i-blade screw, absorbed power increases approximately i/2

For selection of multiblade screw can be used normal digram of characteristics of two-blade type, increasing the value of β in i/2 times. Ratio of diameters at constant power and resultant velocity of blade tips is shown in Fig. 17b. As you can see, three-bladed screw may have a diameter equal to 0.84, and a four- bladed - 0, 73 of diameter of two-blade screw.



a) b)

Fig. 17. Coefficients for transition from two-blade screw to a multiblade screw.

a) power ratio absorbed by blades, b) diameter ratio at constant power and the resulting velocity of the blade tips

However, number of blades k increases mutual influence of blades and increase their drag. Number of blades also affects maximum relative efficiency factor, which is reached at the blade installation angle $\varphi_{0.75}^{0} \approx 10^{0}$ for two-and three-bladed isolated screws (Fig. 18.).

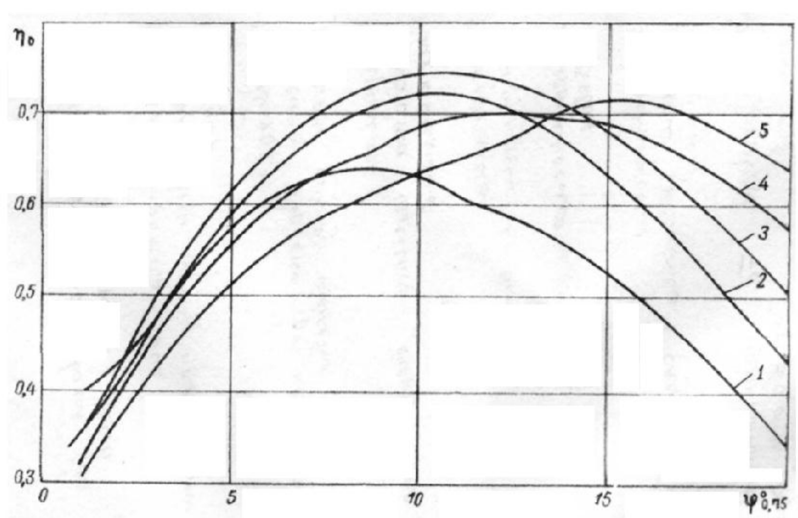


Fig. 18. Summaryzing diagramm of srews of CMB series: 1 - two-blade, 2 - three-blade; 3 - four -blade 4 - six-blade 5 - eight-blade

As the graph shows, excessive increase of number of blades κ reduces efficiency factor and shifts its maximum upwards to installation angle $\varphi_{0.75}$.

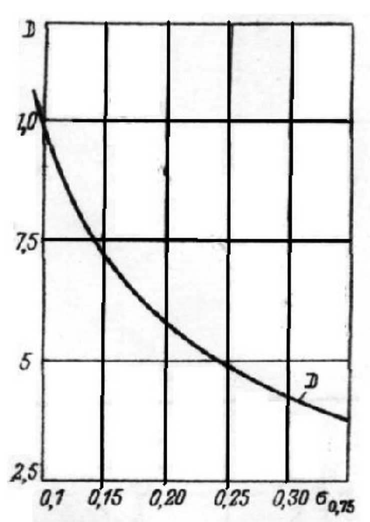
Note that in case where it is necessary to increase the coefficient solidity of a screw, often from the viewpoint of weight savings would be more beneficial increase the number of blades, rather than increase their width. For example, if it is desirable to increase the area of two-blade screw at 50%, then it may done increasing its width by 50%. To maintain the strength and eliminate vibrations, approximately for the same ratio should be increased also the thickness. As a result weight will increase 2.25 times. If instead to that make the screw with an extra blade, the weight of it will increase by about one and half times.

At a specified power consumption of screw, its diameter will depend on the coefficient solidity of a screw $\sigma_{0.75}$.

$$\sigma_{0.75} = \frac{k}{D} b_{0,75}, \qquad (23)$$

Where k - number of blades, $b_{0,75}$ - chord of blade section on radius $\bar{r} = 0.75$.

Typical graph of dependence of screw diameter on coefficient solidity of a screw is shown in Fig. 19.a

a) 

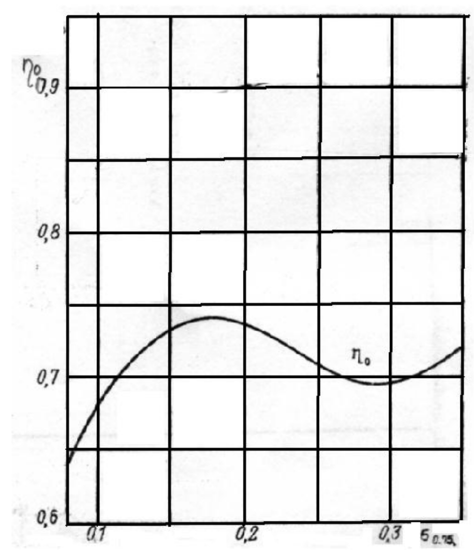
b) 

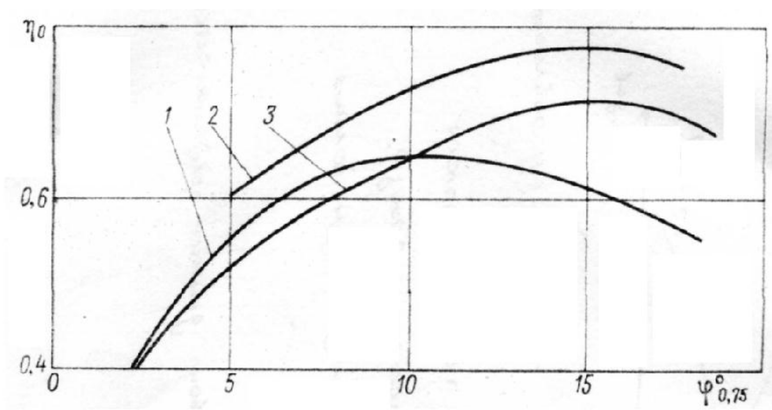
Fig. 19. a) of screw diameter depending on coefficient solidity of a screw $\sigma_{0.75}$

b) Change of efficiency factor η_0 on the hovering regime depending on coefficient solidity of a screw $\sigma_{0.75}$

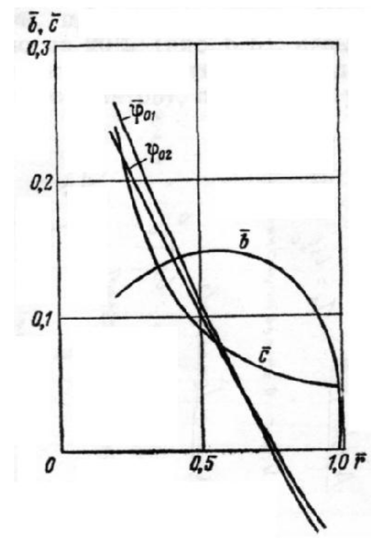
It is obvious that inrease of $\sigma_{0.75}$ causes approximately inverse proportional decrease of D. Note that all graphs 18 - 21 are given for serie of high speed metalc screws (CMB).

Coefficient solidity of a screw $\sigma_{0.75}$ as well influence relative efficiency factor η_0 of screw. Character of dependence $\eta_0(\sigma_{0.75})$ is shown on Fig. 19.b. from the graph foolows that highest efficiency factor η_0 is reached if $\sigma_{0.75}$≈0,176.

For the same number of blades for a single, coaxial and dual screws greatest relative efficiency factor has coaxial screw with two oppositely rotating screws (Fig. 20.a.)



a)



b)

Fig. 20. a) Change of efficiency factor η_0 for single (1), double (2) and coaxial (3) screws depending on $\varphi_{0.75}$. 1 - single eight – blaided screw 2 - coaxial four – blaided screw 3 - double four – blaided screw; b) Geometric characteristics screws (CMB).

Maximum efficiency factor is achieved when the blade installation angle $\varphi_{0.75}$ isapproximately 5º larger than for single screw.

From above presented graph it follows that at working in place of an isolated screw ($V_\infty = 0$), from the viewpoint of efficiency factor it is impractical to use large angle of setting $\varphi_{0.75}$, and it would be preferably limit with value $\varphi_{0.75}^0 \approx 10^0$.

At the same time from position of getting highest screew thrust (respectively maximum velocity in the jet behind screw) it is desirable to increase the installation angle till $\varphi_{0.75}$~ 25 ÷ 35º. This conclusion is supported by experimental data (Fig.21.) for screw thrust coefficient and geometric characteristics, shown on Figure 20b.

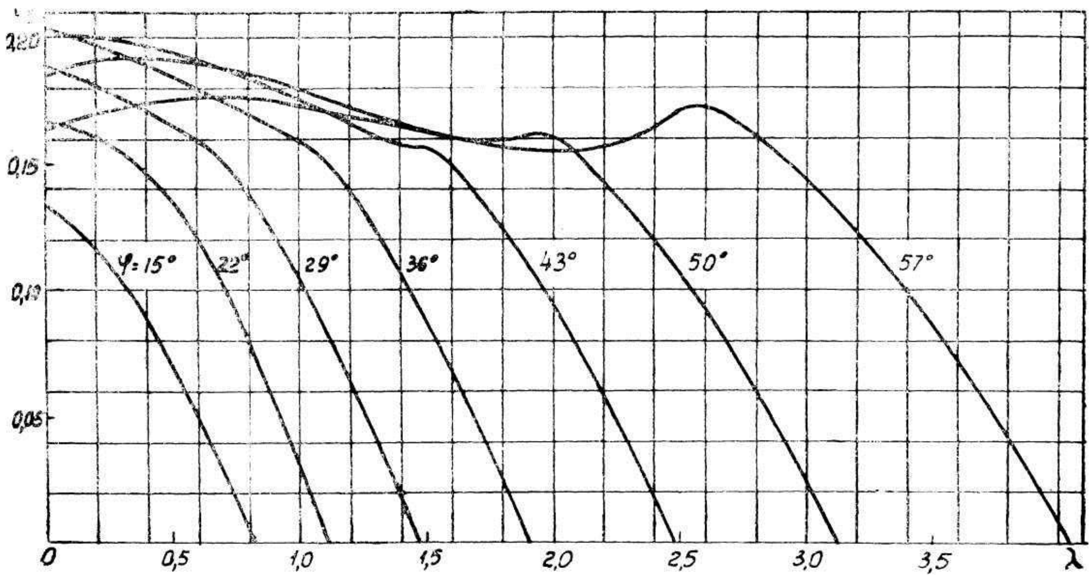


Fig. 21. Dependence of thrust coefficient $\overline{\alpha}$ from advance ratio $\overline{\lambda}$ at different blade installation angles of 8CMB-I2 series screw

Thus, selection of optimal angle of blade should be realized experimentally, based on particular operational conditions of screw installation.

The influence of ring on operation of aerodynamic screw in the system "shrouded propeller" (screw in the ring).

By flow streamlining of isolated ring by axial flow in the narrowest part of the ring will be achieved the highest velocity, and of course the highest depression. The thicker the ring, the higher effect on value of velocity and pressure inside the ring. If in the narrowest section of the ring would install the screw and put it in rotation supplying power to the screw axis, than to the depression created by shaped ring is added depression created by screw. In the screw rotation plane creates pressure drop. Additional depression in front of screw increases air mass flow rate through the screw rotation plane in the ring compared with the isolated screw. Ring can either increase traction force of "screw in the ring," or decrease [22,26]. Fig. 22 shows a comparison of jets of screws with and without rings. Ppropeller slipstream encased in a ring is not compressible and has double of cross-sectional area compared with jet of free screw. The consequence is an increase in efficiency factor of screw.

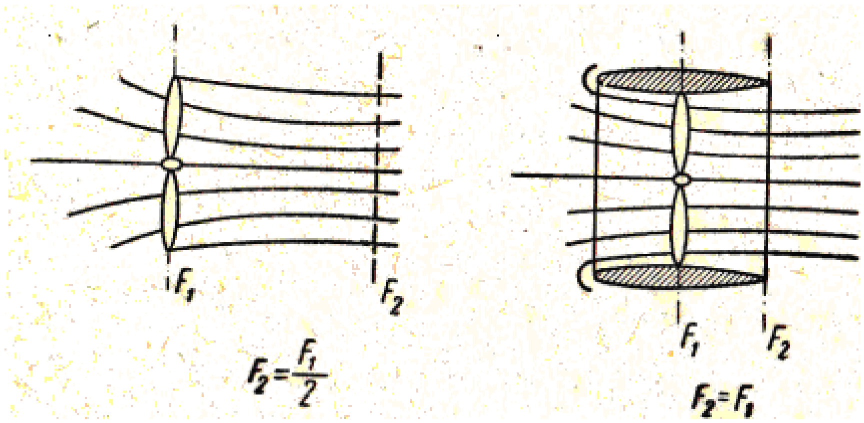


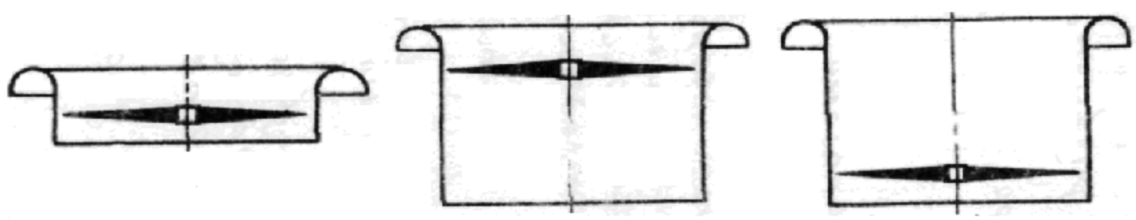
Fig. 22. Comparison of the jets for screws with and without ring

Ratio of efficiency factor of the ideal propeller and ideal propeller enclosed in ring is:

$$\frac{\eta_t}{\eta} = \frac{2 \cdot V_1}{V_1 + V_0} > 1, \qquad (24)$$

Since V_0 always is less than V_1, and in the case of aerodynamic stand $V_0 = 0$. Thus, efficiency factor of propeller in the ring is higher than of free propeller. As can be seen from the formula, the difference is greater, the greater the difference between the V_1 and V_0. Consequently, when $V_0 = 0$ the gain from the use of the ring is twice higher.

For stands, creating a vertical air jet interesting are systems "screw in the channel" with a one-sided profiled rings, which are shown in fig. 23 a, b, c: screw in the short channel (fig.23 a), screw in a long channel with shallow location of screw plane (fig.23 b), screw in a long channel with a deep location of rotation plane (fig. 23 c).



a) b) c)

Fig. 23. Possible screw locations in the channel: a) Screw in the short channel; b) Shallow placement of screw; c) Depth placement of screw.

Since the investigated ground aerodynamic stands have restrictions on height, the most interesting is the screw in the short channel. In works [22-25] are presented numerous semi-empirical data about effect of channel geometry, screw parameters and especially its blades installation angle on the flow pattern at the entrance into the channel and aerodynamic characteristics of system "screw in the ring" applicable to aircraft of veticalo takeoff and hovercraft.

Note that in the system "screw in the ring" the important role plays the clearance between the blade tips and the ring, inside which is placed the screw. Reducing clearance increases the traction. Graph of dependance of coefficient K_1 characterizes the decrease of truction coefficient $\bar{\alpha}$ on relative size of clearance δ (in percent) is shown in Fig 24. [10]. The figure shows that the clearance should be minimal

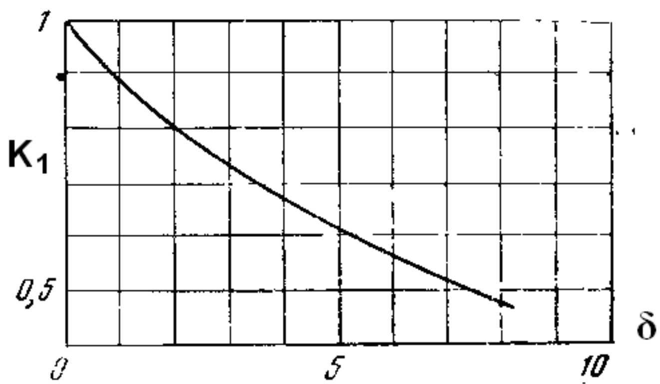


Fig 24. Graph of dependance of coefficient K_1 characterizes the decrease of truction coefficient $\bar{\alpha}$ on relative size of clearance δ% (in percent).

Relative size of clearance is determined by the formula:

$$\delta = \frac{D_r - D_s}{D_s} \qquad (25)$$

Where D_r – ring diameter , D_s - screw ring diameter.

Methods of elimination of swirling behind screw

Rotation of jet behind screw is not required for obtaining of axial air velocityor geting traction. The reason of jet rotation is that the screw passes to air its own torque. To resolve this problem, to the jet must be applied torque with an oposite sign. In some cases jet behind the screw is heavily swirled, and its rectification greatly increases efficiency factor of installation. Therefore bring great benefit gives contrapropeller or rectifying vanes (rectifying apparatus) which increases traction. contrapropeller or rectifying apparatus can be installed either in front the screw, or behind it.

Theoretically sections of contrapropeller should have a shape similar to the shape of corresponding sections of screw to swirl the flow behind (or before) itself like a screw, but in opposite direction. As a result, flow behind (before) contrapropeller will have a swirling (fig.25 a velocity of rotation behind a screw (fig. 25 b before) contrapropeller $u = 0$). Installation angle of sections of contrapropeller can be approximately calculated by the Yuryev method [10] depending on corresponding angle of setting of sections and operation regime of airscrew, as well as direction of its rotation (clockwise or counter-clockwise).

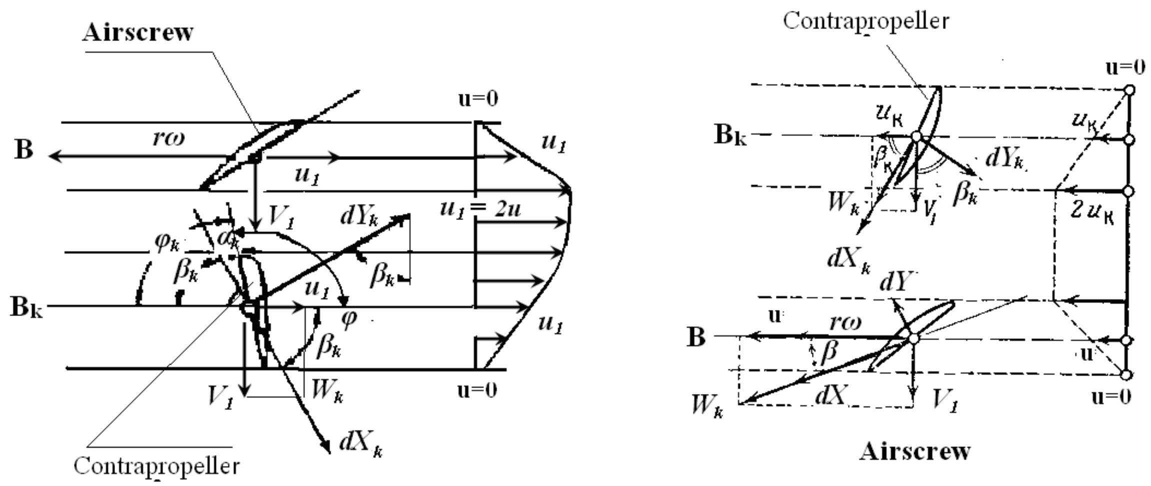


Fig. 25. Airscrew and contrapropeller.

It should be noted that in practice, calculated by this method rectifying apparatus do not provide a complete rectification, but the error usually is not too high. Therefore, in case of need rectifying apparatus shoud be tuned in a natural experiment.

Note that the jet behind screw is compressed. Therefore contrapropeller which is placed in front of screw has a larger diameter than the screw, but placed behind the screw has a smaller diameter than diameter of the screw.

Considered calculation method of rectifying apparatus gives a good result for rectifying apparatus and fans or screws placed in the ring (pipe), as there is no compression of the jet.

Input device

The inlet device of system "screw in a ring" (shrouded propeller) has axisymmetric shape and are characterized by three parameters: length and diameter of inlet hole and the minimum cross-section. There are a variety of inlet devices (see, e.g. [22]). Hydraulic losses at the inlet of device depend on form of inlet device and can be considered using corresponding resistance coefficient . To reduce losses, except smoothly streamlined inlet, velocity in the inlet must be as low as possible. Therefore the inlet area should be the greatest.

It is difficult to calculate accurately pressure losses in the inlet device, especially considering mutual influence of flows "screw in a ring" (shrouded propeller) - an input device. For the approximate engineering calculation of pressure drop can be used published material about inlet sections of channels and pipes [27].

For an approximate estimation of losses in idealized inlet device with a smooth unseparated flow entrance in the ring, can be used the method described in [23]. There also are given the results of comparison with experimental data.

These results can be used for engineering of aerodynamic stand.

Decrease of curvature radius of collector increases the local resistance coefficient, and respectively also resistance of the flow part. If $r/D \to 0$, that is with the sharp edge of the ring, the value $\xi = 1$.

For r/D > 0,3 local resistance coefficient remains almost constant $\xi \approx 0.25$ Fig.26.

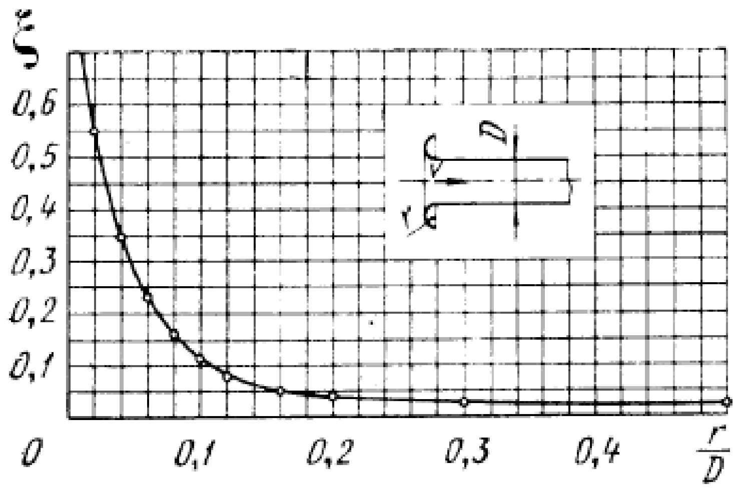


Fig. 26. Dependence of curvature radius of collector from local resistance coefficient

2.Engineering calculation method of circuit of wind tunnels

For calculation of gas-dynamic channels of aerodynamic stands investigated in the thesis, can be used well-known engineering method of calculation of wind tunnels, which is a consistent selection of the parameters of the individual parts considering hydraulic losses [22, 27].

In this case, the problem of engineering of circuit of an aerodynamic stand consists from following steps:

- Definition of shape and size of the individual elements;
- Determination of the resistance coefficients of network (total pressure loss) of these elements;
- Determination of requirments for fan pressure and required engine power;

In general, except engineering of circuit is required:

- To select or design the fan, which is most suitable for a given flow rate, diameter and required pressure;
- Selection and adjustment of engine operation joined with the fan.

Dissertation addresses the first three stages related to engineering of circuit of aerodynamic stand.

Evaluation of hydraulic resistance of air-gas channel of aerodynamic stand

Aerodynamic resistance of flow part of the stand (inlet collector of ring (input device), screw, part of ring behind screw with surfaces of vanes of rectifying apparatus, aerodynamic losses of the jet exit from fan ring, mesh - trampoline, etc.), following the known calculation methods of wind tunnels [22,27] can be presented as the sum of hydraulic resistances:

$$H = H_e + H_d + H_{ns}, \tag{26}$$

where H_{ns} - resistance of suction network channels (from the beginning of inlet collector till screw); H_d - resistance of discharge network channels (from the screw to the exit from the ring, including additional resistance of elements of directing vanes); H_e-losses of dynamic pressure of air jet at the exit from ring (sudden expansion).

Using the known coefficients of local losses ξ_i and friction losses of air on walls of considered channel λ_i, expression (26) can be written as

$$H = \xi_e \frac{\rho \cdot V_e}{2} + \Sigma \xi_i \frac{\rho \cdot V_i^2}{2} + \Sigma \lambda_i \frac{l_i}{d_i} \cdot \frac{\rho \cdot V_i^2}{2} \tag{27}$$

Where:

ξ_e - coefficient of dynamic losses at the exit from ring with an average velocity V_e;

ξ_i - local resistance coefficient of considered section of flow part, where average velocity is V_i;

λ_i - friction coefficient of air against walls of considered section of channel;

l_i , d_i - length and equivalent diameter of mentioned section;.

Coefficients ξ_e, ξ_i and λ_i, , in general, depend on Reynolds number, profile of velocity field, wall roughness and flow turbulence. They are determined theoretically or using reference data [27].

The second and third members of expression (27) can be written as:

$$H_k = \Sigma(\xi_i + \xi_{\lambda i})\frac{\rho V_i^2}{2} = \xi_k \frac{\rho \cdot V_e}{2}; \tag{28}$$

$$\xi_{\lambda i} = \lambda_i \frac{l_i}{d_i} = const, \tag{29}$$

$$\xi_k = \Sigma(\xi_i + \xi_{\lambda i})\left(\frac{F_e}{F_i}\right)^2; \tag{30}$$

Where $\xi_{\lambda i}$ - reduced friction coefficient, F_e, F_i - cross sectional area of ring and outlet section of the considered channel.

As a result the total losses considering jet exit from the ring and taking in to account expression (28) are defined as follows:

$$H = (\xi_e + \xi_\kappa)\frac{\rho \cdot V_e^2}{2} = (\xi_e + \xi_\kappa)\frac{\rho}{2}\left(\frac{Q}{F_e}\right)^2 \tag{31}$$

Equation (31) describes the total energy loss in the hydraulic network of fan inlet manifold (screw), hydraulic and takes into account the losses in the jet out of the ring

Using equation (31) can be estimated overall aerodynamic losses in aerodynamic stand, as well as contribute into the losses of some of its elements, such as profiled ring and rectifying apparatus.

Main advantages of considered engineering calculation method:

- Shape and size of individual elements of vertical pipe in a first approximation can be selected based on historical data about wind tunnels;
- For some characteristic elements of aerodynamic stand, such as an inlet device, honeycomb, nozzle, safety grid, working part, diffuser, engine, exhaust diffuser, etc., values of resistance coefficients ξi are given or in the tables, or as a simple analytical dependencies [27, 28].

Disadvantages of considered engineering calculation method:

- Results of calculations of hydraulic resistance are approximate, as they don’t take into account the mutual influence of aerodynamic elements of stand; obtained results usually must be clarified during finishing process and natural experiment;

- In many cases at calculation of non-standard aerodynamic stands, there are not available required necessary experimental reference data, and as well reasonable recommendations for selection of individual elements of construction;
- The need for fine-tuning in the natural tests results in significant material expenses.

3. Numerical calculation methods of elements of aerodynamic stand

Computer methods of calculation, use professional programs and the specialised techniques developed for the certain class of problems, are widely used in a cosmonautics, aviation, propulsion engineering, calculation of winds tunnels and so on.

Currently quality improvment of aerohydrodynamic installations is a very actual task, whose solution is cused by high competition among their producers on the world market, as well a tendency of requirements increase for energy saving of installations during operation.

The practical significance is determined by ability to use results (models, algorithms and their software implementation, results of calculations) to resolve a number of applied tasks of numerical flow simulation in a aerohydrodynamic installations. Including for calculation of integral parameters of operation modes: power, efficiency factor; dynamic characteristics: noise, transient loads on elements of installation and local characteristics of flow: vortexes, cavitation zones.

Significant increase of productivity of computing systems causes utilization of increasingly complex models and formulation of problems in order to adequately describe the flows, considering their specific features previously unavailable for simulation.

For example, programs of calculation of aerodynamic characteristics of the bodies installed in working sections of winds tunnels, are assign for replacement of aerodynamic experiment, computer calculations. That is partial replacement of physical experiment with the numerical. Such programs [29,30] include on application of specifying coefficient at replacement of digital experiments with the physical.

In the work [31] is reviewed and proposed mathematical model that is implemented in the technological cycle of execution of testing in ETW (European Wind Tunnel). It is used in two main directions. With its help are investigated flow characteristics in the path of wind tunnel and the influence of main elements of construction on this flow. Besides is realized the necessary correction of the experimental data in order to take into account the influence of walls and supporting devices on the results of experiment.

Numerical calculation of aerodynamic characteristics of developed airscrews, allows improving appreciably-volume of experiment and quantity of experimental models.

Currently for the calculation of air screws with specified geometry of blade in practice are widely used two well proven numerical methods:

- Nonlinear vortex theory of screw with a finite number of blades [32-34];
- Nonlinear nonstationary theory of screw, based on method of discrete vortexes [35,36].

Application of these methods allows to use the advantages and benefits of each of them in addressing of specific applied task. For example, the vortex theory of screw allows you to quickly perform parametric studies, and the method of discrete vortexes - to determine the flow field in its vicinity, to clarify distributed and total characteristics, to investigate interference effects.

Numerical method developed based on the work of G..Maykopar and A.Lepelkin on the vortex theory of screw [32-34]

In the creation of calculation scheme of screws is used the approach described in the work [32]. Load bearing surface of the blade is replaced by adjoint radial vortex with variable along the length circulation. Descended from the blade vortex sheet is simulated by system of vortexes.

In the works [37,38] are considered main regularities of development of the dynamic breakdown from the profile of the blade and methods of its calculation. Based on these methods in the work [39] is proposed an approach that allows within the vortex theory to calculate aerodynamic parameters of screw at presence of dynamic breakdown from those sections of blades, where local angles of attack of sections exceed the critical ones.

Nonlinear vortex theory of screw with a finite number of blades

In the work [39] a nonlinear nonstationary theory of screw [35,36] is applied to research problems of aerodynamics of aircraft air screw and its interference with other parts of the aircraft (ring, horizontal and vertical empennage, etc.), where the use of the vortex theory of screw based on the methods of plane sections is ineffective.

To solve this problem is used in the nonlinear theory of screw [35,36] generally accepted hypothesis and assumptions:

- It is considered the operation of screw with arbitrary number of blades with the specified geometry (shape in plan, twist, profile curvature, etc.) in an ideal incompressible fluid;
- On blades of the screw is fulfiled the boundary condition of impermeability;
- On the vortex sheet behind screws are fulfiled conditions of pressure continuity and the normal velocity component;

- All edges of the blades, from which descends the vortex sheet , perfom the hypotheses Chaplygin-Zhukovsky;
- At the infinity from the screw and its trace of perturbations caused by it dempend.

For numerical solution of the problem bearing surfaces of screw blades and the vortex trail behind them is simulated by closed vortex frames. This allowed the use the modular principle of construction of computational vortex scheme, which has proved as very effective for solving of interference problems.

Found from the solution of the system of linear algebraic equations for determination the tensions of joined vortex frames allow to calculate the velocity field in the vicinity of screw and build the position of vortex sheet. Thus, in the calculation process we can obtain full information about the vortex structure of screw and perturbed velocity field. The problem is solved by the setting of discrete time steps.

Based on this theory in the work [39] was developed the calculation program, which allows to calculate not only the aerodynamic characteristics of an isolated screw, but also to study its interaction with other parts of airframe of aircraft.

Presented in the work [39] numerical methods for calculation of aerodynamic characteristics of isolated air screw and developed on their basis work programs are used in JSC "Scientific and Industrial Services" in the development of a light aircraft "Phoenix 2" and "Remez-3".

Known numerical methods of calculation of sustainer screws and the air-gas channel of wind tunnels by means of programm ANSYS

In the work [40] is presented the method of numerical determination of screw thrust characteristics of sustainer screws of the hovercraft taking into account its aerodynamic interaction with the composition elements of the hovercraft. In the work for investigation of aerodynamic interaction of sustainer screws of the hovercraft with the composition elements of the hovercraft was used direct method of mathematical simulation for description of turbulent flow of a viscous fluid. Implementation of the method have been performed using modern package of computational fluid dynamics software Ansys CFX on a multiprocessor computational cluster.

Have been considered six bladed air screw with diameter 2,5 m, which rotates with angular velocity ω = 900 - 1000 RPM. The screw is located inside of profiled ring. Hub of the screw is fixed by group of pylons. Ring with the pylons is streamlined by air with a specified flow rate at infinity in front of screw equal speed to of hovercraft (speed range from 0 to 90 km/h). Air is considered as an incompressible fluid. Flow regime – turbulent.

Air movement have been described by the system [41] of continuity and momentum equations, and the equations of k - ε-model, where the unknowns are the functions Vx, Vy, Vz, p, k and ε.

But in of the problem under consideration was that it was considered the principle of inverse motion [42]: the blades are stationary; the air is swirled around the screw with an angular velocity ω.

The equations of fluid motion were solved numerically by the finite volume method. For the construction of computational mesh was used ICEM CFD package. A feature of this package is its modular structure. Area is divided into blocks, in each of which is constructed a family of coordinate lines. Generated mesh is called as structured.

According to the results of numerical investigation have been changed installation angles of the blades, designed rectifying device that allows reduction of amplitude of fluctuating loads on the screw by 25%.

The computational optimization of flow part of wind tunnel of MGSU, considered in [43]. The main purpose of the numerical simulation was to optimize the design of small size wind tunnel of MGSU with closed circuit of circulation in order to minimize irregularities in the horizontal and vertical distribution of velocities in the working area of the tunnel. Used CFD Ansys Fluent.

At designing of aerodynamic stand BT-1 fig. 27 in Moscow Aviation Institute, were applied numerical calculation methods for elements of flow part of aerodynamic stand, considered in the work [44] in order to perform verification calculations of flow in diffusers of different forms.

Calculation was based on the numerical solution of the averaged by Reynolds Navier-Stokes equations with turbulence model of Spallart-Allmares [45]. Flow velocities in the circuit of system did not exceed the Mach number, equal to 0,3, was used the model of a viscous incompressible and heat nonconducting environment. Difference scheme approximating the system of differential equations was based on the control volume approach and the Semi-implicit algorithm semi-implicit algorithm for linking of the pressure equations [46].

The computational model did not contain items such as honeycomb, safety grid and covering of the working zone of flow by parachutists. The power installation was modelled only by case of cowl, which did not allow taking into account the effect of flow swirling behind the fan.

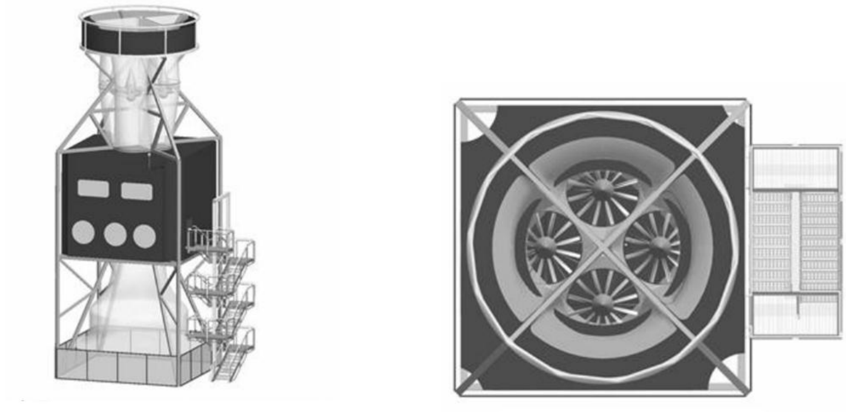


Fig.27. Design of aerodynamic stand

This simplified model allowed only evaluating the nature of flow in the nozzle and discovering the places of zones of reverse flow, evaluating uniformity of the velocity field in working area of the installation and the following it diffuser.

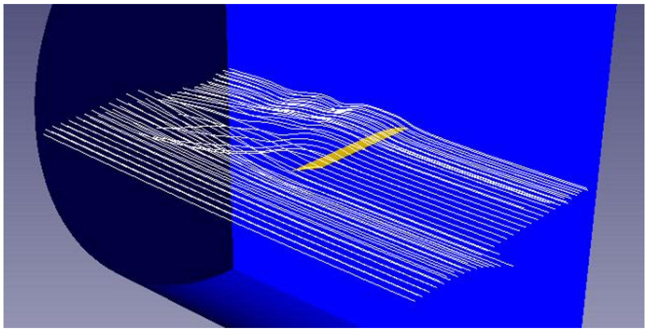
Engineering (enough simple) programs are used less often as in most cases they yield the qualitative result. For example SolidWorks + FlowWorks, SolidWorks+FlowVision.

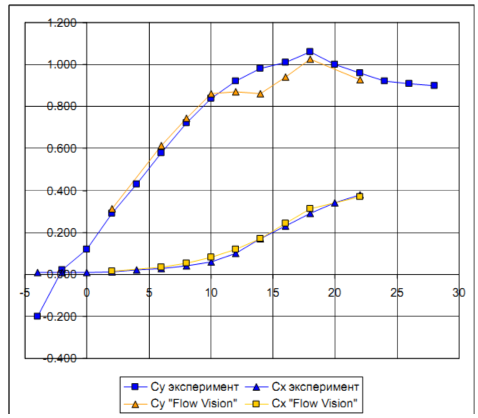
Use of such programs in the course of engineering designing is very claimed now. In this work analysis application of such programs.

In the work [47] were carried out numerical researches stramlining of screw with an aerodynamic profile NACA-2406 in a virtual wind tunnel using the software package FlowVision [48].

Natural experiment of blowing of NACA-2406 profile, was performed by laboratory LMAL - NACA in the tunnel with variable density and diameter 1.52 m (the critical Reynolds number is 150000) on a rectangular model of wing with dimensions 127 × 762 mm ($6 = \lambda$). velocity of incident flow in experiment was 21,1MPS, Re = 3120000, M = 0.06. Results of field experiments were published in the proceedings [18]. Geometry of computational variants of "Flow Vision" is shown in Fig.28 a

The calculations were performed for three-dimensional problem at $6 = \lambda$, as shown in Fig. 2, and for two-dimensional problem with $\infty = \lambda$ in order to be able to compare the results of three-dimensional calculations with experimental results, and the two-dimensional - with calculations of force characteristics of the wing according to theoretical dependencies. In the numerical simulation was used the model of an incompressible fluid with a standard k-ε turbulence model.





a) b)

Fig. 28. a) Computational variant of "Flow Vision", a three-dimensional task. b) lift and drag coefficients of the wing according to results of numerical simulation and experiment

The calculations were performed for three-dimensional problem at 6 = λ, as shown in Fig. 28 a, and for two-dimensional problem with ∞ = λ in order to be able to compare the results of three-dimensional calculations with experimental results, and the two-dimensional - with calculations of force characteristics of the wing according to theoretical dependencies. In the numerical simulation was used the model of an incompressible fluid with a standard k-ε turbulence model.

Results of numerical simulation of stramlining of a wing with the finite span in three-dimensional formulation of the problem and compared with experimental data of NACA.

Corresponding diagrams of dependence of the lift and drag coefficients of the wing on the angle of attack are shown in Fig. 28 b.

From Fig. 28 b it is clear that coincidence of the lift and drag coefficients of the wing takes place at all angles of attack, except for the interval 10º - 18º, which corresponds to the development of separation of boundary layer on the upper surface of the wing (angle of attack 18º corresponds to the maximum lifting force).

In this interval the calculated range lift coefficient was below the experimental, what could indicate about more developed flow separation in numerical calculations compared with the experiment.

In the work [49] using the software package FlowVision special interest represents the problem of simplification the calculation model of jet formation from the mixing screw intended to remove deposits of oil tanks.

Near the bottom of the tank is placed (for example, by module of moving body) screw, which rotates with the specified required frequency. As a result will be got a picture of liquid flow the tank and as a result, the mixing of oil with densified sediment. Rotation of screw was defined using a special module of programm for rotating bodies. Feature of this module consists in constructing around the screw area that rotates together with it and also with computational mesh. Note that the use of this module in this formulation of the problem allows significantly increase accuracy of calculation.

In this way created computational model adequately describes flow in the tank, and formation of the jet from the screw. As showed the results, such formulation of task requires significant computational resources - large RAM combined with a considerable number of iterations to carry out the calculation itself.

Obtained result can be explained by existence in the task two physical processes, which are different in spatial scales and time:

- Small-scale and rapid process - rotation of screw and the formation ofstream of liquid behind it.
- Large-scale and slowly developing process - formation of flow in the reservoir itself.

Consequently, in the full volume formulation of the problem are present both - large cells for description of flow development in the reservoir as a whole, and small cells - in the area where is located the screw. This significantly different scale computational model usually has a high dimensionality (total number of computational cells), which sets high requirements to the amount of RAM of computer.

As a result tor the calculation of task in the complete formulation are required tens and maybe hundreds of thousands of time iterations, that taking into high dimensionality of model by the number of computational cells may require several weeks of continuous computation of one variant.

Based on the performed analysis of full volume formulation of the problem are identified ways to simplify the task.

In particular:

- Substitution of the jet formation procedure behind a screw by the ready solution, which describes main characteristics of the flow (total flow rate of the jet, velocity distribution, dynamic pressure);
- Substitution of mixing device with the disc endowed with integral characteristics (flow rate) of the mixing device. As a simplified model of screw have been considered disc with the same diameter and the same liquid flow rate as the screw. The performance characteristics of the mixing device can be taken either from documentation, or calculation of flow behind the screw.

PROBLEM ACTUALITY

In the world there is already a large number of aerodynamic stands for free flight of human, which vary in design, dimensions, range of speeds, and ways of creating of a stream.

Known aerodynamic stands have closed (i.e. limited with side panels) or an open work area, and should create a long range vertical stream (altitude can reach 20-25 meters from the outlet section of the tube). In the cross-sections of work area of a jet flow must be sufficiently uniform and without swirling. Average vertical velocity in a work zone is approximately 60 -70 m/s at standard atmospheric conditions.

Disadvantages of known systems of creation of vertical air jet in open type devices are inappropriate design of the above mentioned elements and increased operating costs due to the high losses of flow energy as in separate elements, as well in the system as a whole at the assembly into a single set of elements without taking in to account their reciprocal aerodynamic effects.

Standard engineering methods for calculating of wind tunnels represent a sequential selection of parameters of their individual sections based on hydraulic losses generally without taking into account their mutual influence in a common aerodynamic system. As a result is the consistent composition of individual construction elements of wind tunnels which are not agreed among themselves as a unique aerodynamic system. Various assumptions, as well as the lack of information about the full picture of currents of flow and mutual influences of the considered sections do not allow optimizing the design of the wind tunnel during the engineering process and getting defined characteristics. In this way calculated patrameters are approximate and require fine-tuning during field tests, resulting in extensive material costs.

As a result, many stands (especially open stands with an open work area), have in the working section of the jet significantly uneven speed profile, high flow swirl and as well as large required power supply.

Modern constructions of aerodynamic stands have large dimensions, complex forms, making difficult and expensive implementation of natural or semi-natural experiment, especially if we are talking about an aerodynamic experiment. Therefore, the creation of structures and their optimization is not possible without improving and automating the design, calculation and experiment. Thus the use of modern methods of computer modelling and design of these stands are relevant.

THE PURPOSE OF THIS WORK AND THE FORMULATION OF THE TASK

The aim of this work:

Develop a methodology of computer modelling and numerical experiment to optimize geometric and aerodynamic parameters of individual elements of aerodynamic stands, as well as in the whole system: the propeller (fan) - rectifying apparatus - gas-dynamic channel. The results of numerical calculations, executed through the developed methods require the satisfactorily matching with the results of natural experiment.

In order to achieve the aim in the work must be the performed the analysis of:

- The general directions of constructions development of simulators (stands) for sport flights in vertical air jet;
- Methods of engineering calculation of elements of aerodynamic stands;
- Existing programs for calculation of gas flow interaction with moving objects in complex gas-dynamic channels.

And to solve the following problems:

- Development of methods of computer simulation and calculation of parameters of the open wind tunnel with vertical air stream of large diameter.
- Development of methods of computer simulation of geometric and aerodynamic characteristics of individual elements of aerodynamic stands in particular airscrews, inlet devices, rectifying apparatus - gas-dynamic channels.
- Development of methods of computer simulation and optimization of geometric and aerodynamic parameters of aerodynamic stands of different versions (single screw, open type multi-screw).
- Experimentally to prove the possibility of applicability of the suggested method and its individual parts for the calculation and optimization of aerodynamic parameters of aerodynamic stands.

CHAPTER 1.

METHOD OF CALCULATION OF PARAMETERS OF OPEN WIND TUNNEL WITH LARGE DIAMETER VERTICAL AIR JET USING SET OF CAD/CAE SIMULATION PROGRAMS. [64, 66, 82, 83]

Numerical solution of any particular task requires obligatory preliminary elaboration of a phased approach of its solution [50-54]. In the case of analysis of a certain class of tasks (in this work is the class of tasks of aerodynamic characteristics of vertical air jets created by ground stands of open type wind tunnels), can be selected the main part of a method which will include steps common for this class of methods. Method of resolution of a particular task will contain only specific parts inherent to this task.

Main stages of the basic methods for resolution of tasks on simulating CAD/CAE programs set used in this paper, can be represented as follows:

- The physical formulation of the problem of study.
- Development of a simplified model of original research object.
    - Creation of an electronic geometrical model of the object.
- Mathematical formulation of the task, boundary and initial conditions.
- Selection of applicable CAD/CAE programs.
- Creation of discrete calculation model, optimization of computational mesh.
- Development of strategy for resolution of task:
    - ✓ Formulation of the goals of computation and criteria for termination of the computation;
    - ✓ Method of calculation control and monitoring process;
    - ✓ Method of data visualization and processing of the digital calculation results.
- Solution of validation tasks, comparison with known data, assessment of the accuracy of solutions.
- Interpretation of the calculation results to optimize the properties of the investigated object.

Purpose of this chapter is formulation basic method of computer simulation and calculation of parameters of generators of large diameter vertical air jet (or open wind tunnels, generating the air jets). The method must enable parametric computer calculations which pretty accurately model researched physical phenomena and allow defining their interesting the practice characteristics. In addition, calculations must satisfactorily match with the experimental data or known theoretical calculations confirming the adequacy of the developed method. Calculation of developed aerodynamic stands must be done by means the known CAD/CAE engineering programs that ensure compliance with all the necessary work to be done in a reasonable time period.

1.1. Physical formulation of the task

Computational physical model includes a simplified geometric construction of aerodynamic stand, without the minor parts, with has a little effect on its aerodynamic characteristics [51, 54] (for example see Fig. 1.1.).

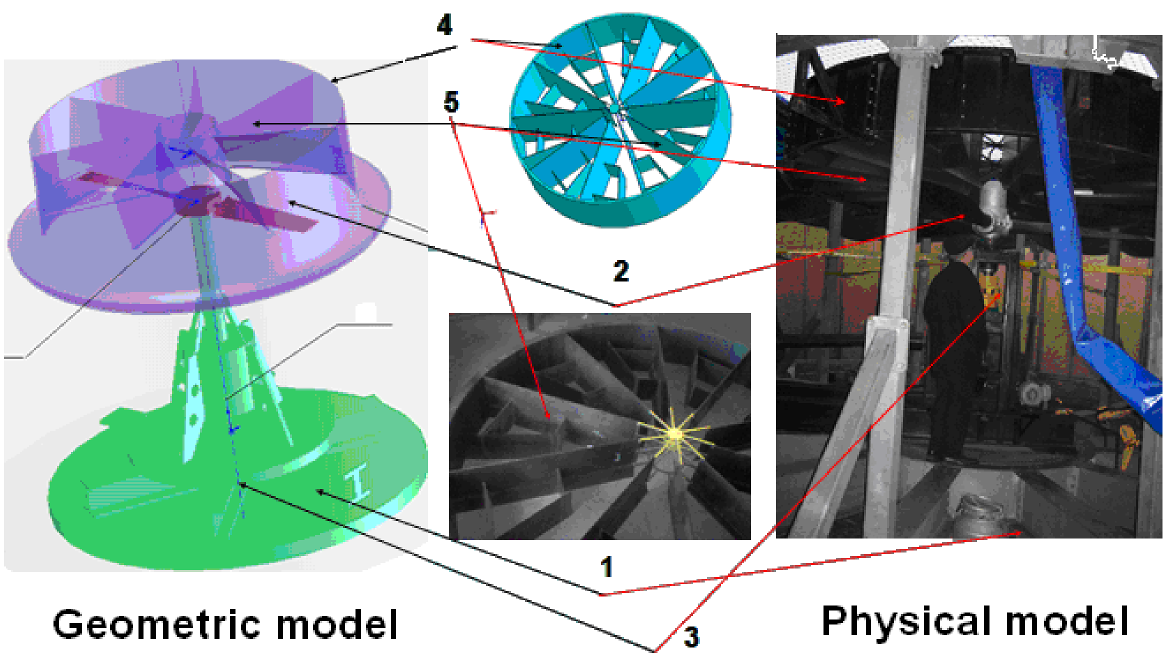


Fig.1.1. Simplified geometric construction of aerodynamic stand.
1- Ground; 2- Airscrew; 3- Axis of engine; 4-Ring; 5-Directing vanes.

Thus the physical model can be provided with only a part of properties of real construction. This will simplify the geometric model and mathematical description. How successful chosen physical model of construction, depends laboriousness of calculation and the accuracy of its results. Here much depends on understanding of the features of the construction and ability to highlight specific details, which generally determine its work.

For the stand, which generates a vertical air jet, the main feature of the task is the necessity to calculate formation process of submerged vertical air jet created by the aerodynamic screw, which rotates around a fixed vertical axis. Besides it is necessary to take into account the impact of closely located solid or permeable surfaces, as well as the creation of flow zones, induced by submerged jet and low pressure area in front of the screw (see Introduction, paragraph 1, flow diagram on Fig. 1.b). Generally flow in the jet is unsteady and turbulent, and in adjacent areas of the induced flow can be transitional or laminar.

Geometric models of considered open type aerodynamic stands contain the following major structural elements: aerodynamic screw or fan connected to motor; channel, which forms air flow entering on the screw or fan (in some prototypes doesn't exist); cylindrical lead channel, inside of which in general are installed guide blades and rectifier for elimination of flow swirling behind the screw (fan) and formation of a uniform vertical air flow at the exit of lower air permeable protective grid from which begins the working area of the free vertical air jet.

Scheme and principle of operation of these installations more detailed are described in Introduction, paragraph 1.

1.2. Mathematical formulation of the task. Formulation of boundary and initial conditions.

For the mathematical simulation of medium's motion and heat exchange are used non-stationary Navier—Stokes equations, the energy equations (the first law of thermodynamics) and the state equation.

For turbulent flows initial equations are averaged by the Reynolds method and taking into account the additional turbulent shear stresses, caused by turbulent pulsations of parameters. Obtained not closed system of equations is closed by the additional equations for the turbulent kinetic energy k and the turbulent energy dissipation ε according to the known k — ε model of turbulence. The system of equations for the momentum, mass, and energy conservation describing the turbulent, laminar, and transient flows of a compressible fluid and the heat exchange can be written in the following form [51, 54-58]:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j - \tau_{ij}) + \frac{\partial P}{\partial x_i} = F_i; \tag{1.1}$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0; \tag{1.2}$$

$$\frac{\partial (\rho E)}{\partial t} + \frac{\partial}{\partial x_i}((\rho E + P)u_i + q_i - \tau_{ij} u_j) = F_i u_i + Q_H; \tag{1.3}$$

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_i}\left(\left(\mu_l + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right) + S_k; \tag{1.4}$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_i}\left(\left(\mu_l + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right) + S_\varepsilon; \tag{1.5}$$

$$S_k = \tau^R{}_{ij}\ \frac{\partial u_i}{\partial x_j} - \rho\varepsilon + \mu_t P_B; \quad \rho = \frac{P}{RT}\ ; \quad q_i = -\left(\frac{\mu_l}{\Pr} + \frac{\mu_t}{\sigma_c}\right) c_P \frac{\partial T}{\partial x_i}; \tag{1.6}$$

$$S_\varepsilon = C_{\varepsilon 1}\frac{\varepsilon}{k}(f_1 \tau^R{}_{ij}\frac{\partial u_i}{\partial x_j} + \mu_t C_B P_B) - C_{\varepsilon 2} f_2 \frac{\rho \varepsilon^2}{k}; \quad P_B = -\frac{g_i}{\sigma_B}\frac{1}{\rho}\frac{\partial \rho}{\partial x_i}\ ; \tag{1.7}$$

$$\tau_{ij} = (\mu_l + \mu_t)(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_l}{\partial x_l}\delta_{ij}) - \frac{2}{3}\rho k \delta_{ij}\ ; \quad f_1 = 1 + \left(\frac{0.05}{f_\mu}\right)^3; \ f_2 = 1 - \exp\left(-\left(\frac{\rho k^2}{\mu_l \varepsilon}\right)^2\right) \tag{1.8}$$

$$\tau^R{}_{ij} = \mu_t(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_l}{\partial x_l}\delta_{ij}) - \frac{2}{3}\rho k \delta_{ij}; \tag{1.9}$$

$$\mu_t = f_\mu \frac{C_\mu \rho k^2}{\varepsilon}; \ f_\mu = [1 - \exp(-0.025\frac{\rho\sqrt{k}y}{\mu_l})]^2 \cdot \left(1 + \frac{20{,}5\mu_l \varepsilon}{\rho k^2}\right), \tag{1.10}$$

where u, P, ρ, T - pressure, velocity, density, and temperature of the fluid; R - gas constant; t - time; F_i - total force per mass unit; E - total energy of mass unit of fluid; QH - the heat source per volume unit; q, - the diffusion heat flow; δ_j - the Kronecker symbol; τ_{ij} - viscous shear stress tensor; $\tau^R{}_{ij} \equiv -\overline{\rho u_i u_j}$ - the stress tensor in the Reynolds model; μ - coefficient of dynamic viscosity; μ_t - coefficient of turbulent viscosity; y - the distance from the solid wall; g_i - components of the gravitational acceleration in the direction x_j; σ_c, σ_B, σ_k, σ_ε, C_B, C_μ, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$ - empirical constants; c_P - specific heat at constant pressure; λ - coefficient of thermal conductivity of gas (liquid); $Pr=\mu cp/\lambda$ – Prandtl's number; for the laminar flow the parameters

k, μ ε are equal to zero; x, y, z are the changing coordinates; summation is carried out according to subscripts i,j = x,y,z.

Upon analyzing the conjugate heat exchange between the gas flow and the solid body, heat transfer in the body is simulated by the known heat transfer equation:

$$\frac{\partial \rho e}{\partial t} = \frac{\partial}{\partial x_i}\left[\lambda_S \frac{\partial T}{\partial x_i}\right] + Q_H\,, \tag{1.11}$$

where $\boldsymbol{e} = \boldsymbol{cT}$, $\boldsymbol{c}$ - the specific heat, $\boldsymbol{T}$ - temperature, $\boldsymbol{\lambda_S}$ - thermal conductivity of solid body, $\boldsymbol{Q}$H - heat source per unit of volume of the body. The above equations (1.1 - 1.11) are very general. Later in solving of individual tasks are specified values of constants, as well as the dependent and independent variables.

Let's note that Low Re *(k - ε)* model is adjusted for solution of tasks, which contain (they have) zones both with the large and with the low velocities. In particular, this model is convenient for the calculation of the high-velocity submerged jets, which enter into large volume. Air jet interacts with the ambient air, implicating it into the induced movement, which ensures entering into the jet of air mass from the surrounding area. For the induced flows in the outlying zones are characteristic low Reynolds numbers Re ~1500… 5000. Advantages *(k -ε)* models are exemine in works [31, 60-64].

Formulated a system of differential and/or integral equations is nonlinear and, in general, has no analytic solutions. Therefore, it should be solved numerically. It involves transition from differential (integral) equations to their finite difference analogue. As a result, mathematical task of solving a system of differential (integral) equations is reduced to a mathematical task of solving of algebraic (usually nonlinear) system of equations. Instead of a continuous solution of the task will be obtained a discrete set of digital values at certain points (or cells) in the space and for certain points in time (if time-dependent problems are solved).

In this work was used engineering CAE software (see paragraph 1.3.2.), in which the Navier-Stokes equations were solved numerically by finite element method using adaptive movable mesh [55].

For the mathematical description of motion of a solid rotating body in the flow, is established concept of so-called Rotating Region. Rotating Region is a closed, permeable for flow area that surrounds rotating device. In the process of calculation Region rotates relative to a defined axis, and solid body inside the region rotates with it at the same angular velocity. Rotating Region lets to simplify and speed up the calculation of aerodynamic screws and rotating machinery such as pumps, blowers, turbines and mixers [49, 51, 55, 65, 66].

The rotation of the Region can be defined specifying angular velocity, torque or initiated by applied aerodynamic forces, which are defined during the calculation process. A detailed mathematical description of motion of a solid body, induced by the flow is given in [67].

Formulation of boundary and initial conditions

To relate mathematical model to a specific physical task and to an area of space in which it is solved, it is necessary to specify the initial and boundary conditions [51, 54, 55, 57, 68, 69].

Necessity to define initial conditions, that is, values of physical parameters of the medium (the fluid and solid, if calculated heat transfer in solids) in the computational domain, follows from the mathematical model of nonstationarity, i.e., its basic equations (1.1.-1.11.). If task is unsteady, and its solution is not periodic, then the initial conditions, together with the boundary conditions are determining solutions of the task, that is, cannot be arbitrary, but must perfectly match with the task. If the task is stationary or unsteady, but with the periodic solution, then the solution is found after the establishment in time - in this case, initial conditions can be arbitrary. It must be taken in to account that level of definition of arbitrary initial conditions depends on the problem, but in any case initial conditions must be physically correct. Some tasks may have several stationary solutions corresponding to different ranges of values of initial conditions. So on initial conditions depends not the solution, but the speed of finding of this solution (the closer initial conditions to the solution, the faster solution will be found).

Specification of boundary conditions, i.e. conditions at the boundaries of the computational domain is required for all tasks - both stationary and non-stationary. In fact, the boundary conditions determine the relationship of physical processes inside the computational domain to the physical processes outside. Depending on way how you set the boundaries of the computational domain, and thus the conditions on them, all the tasks are divided into internal and external.

In internal tasks computational domain is filled with fluid and limited by walls of model (if calculated heat transfer, these walls are included in this computational area).

In external tasks must be created a fictitious domain, to be filled with fluid which is completely surrounding a solid.

Number of initial and boundary conditions depends on order of equation, and therefore boundary conditions and their number are different for different models.

1.3. Complex of programs for calculation of interaction of the gas flow with moving bodies in complicated gas dynamic channels.

Main base for the numerical solution of problem of formation of submerged vertical air flow and realization of virtual blowing of aerodynamic stands - are CAD/CAE/CFD programs (CAD - Computer Aided Design, CAE - Computer Aided Engineering, CFD - Computer Fluid Dynamic.).

In CAD program is created a simplified electronic geometrical model of aerodynamic stand. Then this model is exported to the CAE/CFD program for numerical analysis.

It is known that usually during data transfer from one software system to another in a standard format part of data might be corrupted or lost, what can greatly affect result of the task [51, 55].

Therefore, special attention should be given to proper selection of interoperable CAD/CAE/CFD programs. Pre-selection of complex of CAD/CFD programs includes two phases:

- Selection of CAD programs;
- Selection of CAE/CFD programs.

1.3.1. Selection of optimal CAD program for solid parametric simulation

First step is selection of optimal (simple and accurate enough) CAD program for solid parametric simulation for construction of three-dimensional electronic model of investigated aerodynamic stand.

Based on analysis of number of possible solid parametric simulation programs such as PRO/Engineer, CATIA, Autodesk Inventor, Solid Edge, Unigraphics, SolidWorks, was selected program SolidWorks [50, 70-73]. Its main advantages are:

- Effectiveness of system to create and modify parts and assemblies;
- Possibility to perform drawings with 3D-models;
- Possibility of interaction with other CAD systems, the presence of additional applications - for strength calculation, modeling of kinematics and dynamics, calculation of flows of liquid and gas;
- Availability of tools for project data management (PDM);

Besides its advantages consist in the fact that the program has wide variety of translators from popular CAD - systems and universal graphic formats. Special attention deserves ability to read files created by other applications. For example, files created with CATIA V5 (they have the extension. cgr), are only illustrative and are not suitable for modification or use as part of assembly. For programs, Mechanical Desktop and PRO/Engineer it is possible complete or partial (depending on what tools were used by base system) re-creation of tree of construction of model.

Very important is relative simplicity of organization of interface between application programs and SolidWorks. In particular, CFD programs, that have enough computing resources, but do not have their own graphics subsystem, follow the path of integration of its mathematical tools in SolidWorks, using procedures of API.

1.3.1.1. Features of methods of creation of digital models.

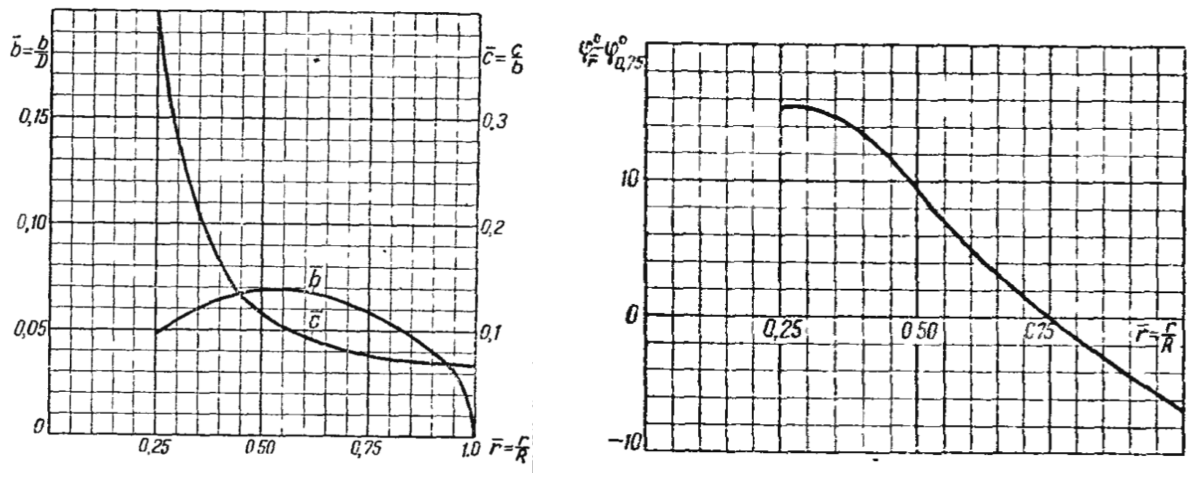
1.3.1.1.1. Geometric modeling of the blade surface of screw using known data.

To construct a contour of profile normally are used geometric characteristics in form of drawing or a table of ordinates. Quite often profiles of screws are built based on the relative ordinates, i.e. respectively to series of ordinates of maximum thickness. It's facilitates transition to sections of different thickness [9, 10, 17, 20, 74].

In this work was constructed three-blade screw of Clark Y type with known [17, 20, 21] geometrical characteristics (Fig. 1.2.a, b), which will be necessary for the calculation of the test task (the task will be discussed in chapter 2).

In Excel by the known formulas (1.12.-1.14.) are recalculated relative geometric characteristics of blade to the real values of coordinates X, Y_{uper}, Y_{lower} for each section of the screw blade.

Where X corresponds to the value of chord of the blade element (width of the blade), and Y_{uper}, Y_{lower} to their thickness.



a) b)

Fig.1.2. Geometric characteristics of screw blade: a) The relative chord $\bar{b}$ ($\bar{r}$), and the relative thickness $\bar{c}$ ($\bar{r}$); b) Twist of blade (φ - $\varphi_{0.75}$) $(\bar{r})$.

Relative radius of section of blade $\bar{r}$

$$\bar{r} = \frac{r}{R} \tag{1.12.}$$

Were ***r*** - radius of section of blade (m) , ***R*** - radius of screw (m)

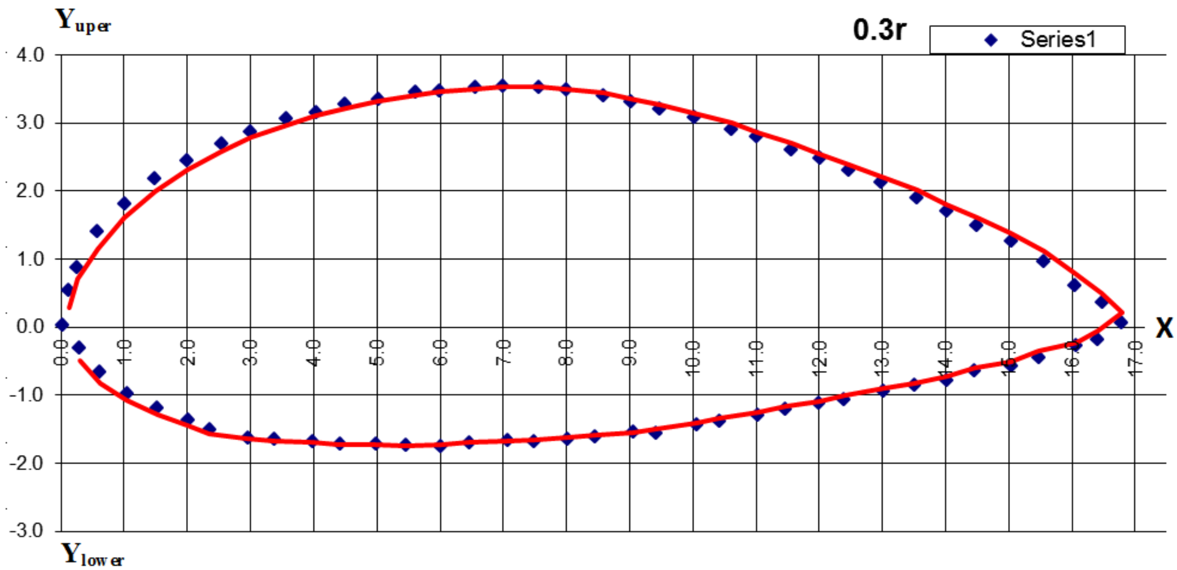
The width and thickness of the blade. Value of chord of the blade elements (blade width) and the thickness are given by following equation:

$$\bar{b} = \frac{b}{D}; \tag{1.13.}$$

$$\bar{c} = \frac{c_{\max}}{b}. \tag{1.14.}$$

Once obtained values of coordinates of the blade section is necessary to build a graph to view and verify accuracy of the coordinates. Fig. 1.3.

After averaging of obtained profile using the moving average method, we can see that obtained coordinate points don't lie on one curve and is necessary to make their adjustment. Adjustment of coordinate points is a mandatory process associated with the fact that during transferring of data into SolidWorks, these errors can worsen accuracy in the created solid model of screw blades. As a result, a solid model of screw blades might not be uniform, and that accordingly affect calculation process and final result of calculation.



1.3. Profile of the blade section on the *r*=0,3.

There are existing two methods of data transferring to the solid modeling program SolidWorks:

Obtained coordinate points are entered directly into the program sketch. In this case, the XY coordinates of each point are defined individually.

In the program SolidWorks is drawn Centerline, which is specified by the angle relative to the horizon. This angle is the angle of the installation of the blade section (φ - $\varphi_{0.75}$). This line also will be used as a chord, and it is defined by a linear dimension, whose value is equal to the chord of section.

Afterwards along the length of the chord are arranged points in the way, that they are approximately shaping the future profile. Their number must match the number of data in the table.

After adding of points with Smart Dimension tool in each point are defined coordinates by two axes: X and Y, beginning from nose of the profile. Fig.1.4.

When all points in the plane of drawing are fully defined, in order to create a profile directly, the points must be approximated by a smooth curve using the **Spline** function.

This procedure is repeated for each individual section of the screw blade, respectively, at each relative radius $\bar{r}$ of the blade. After entire screw blade sections are defined, in the program using the **Loft** function, is built a solid model of screw blade.

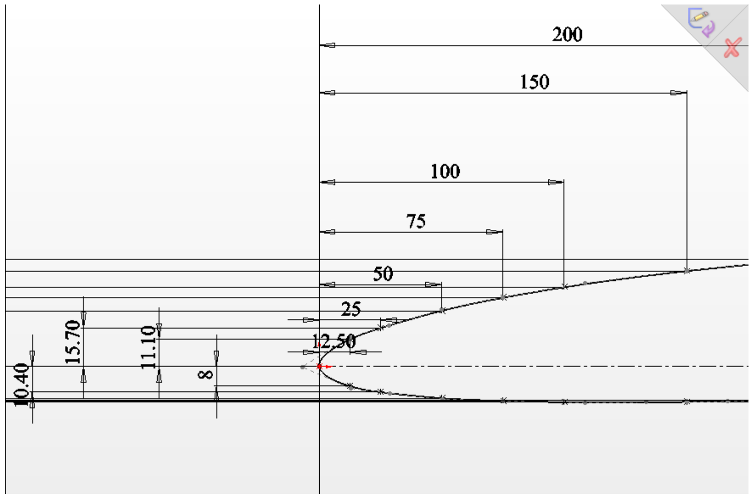


Fig.1.4. Coordinates of ponts of the profile.

Transfer of coordinate points using the Curve Through Reference Point tool in program Solid Works.

Coordinates of the points calculated in Excel are saved with an extension .txt, and for each section of the blade should be created a separate text document. After that in program Solid Works using the *Curve Through Reference Point* tool can be unpacked numerical values of points. After that the program automatically the according to defined coordinates will build the profile of blade section on the specified the relative radius $\bar{r}$ of the blade.

Must be noted that these methods are very time consuming and not accurate. Errors can occur during the reading from the graph relative values of geometric section of blade.

1.3.1.1.2. Geometric modeling of surface of screw blade using the program Profili 2.18a.

Profile [75] is designed for aircraft model constructors, aircraft manufacturers, developers of wind turbines or propellers. This program is free software. Database of the program has more than 2000 profiles.

Therefore if the screw profile is known (if it exists in the database of the program), this program allows quickly and accurately to determine the coordinates of the cross-section of profile of the screw. To determine the coordinates must be entered values of chord of section and maximum thickness of the profile.

Then saving the resulting coordinates and using the method of transfer of coordinate points using the function Curve *Through Reference Point* in program Solid Works (discussed in paragraph 1.3.1.1.1.), can be created solid model of the screw blade.

Must be noted that the Profile does not consider installation angle of section of screw blade (φ - $\varphi_{0.75}$). Therefore since is built in Solid Works profile of screws section using the Rotation tool, is set desired angle of the section.

1.3.1.1.3. Geometrical modeling of the imported surface of the refurbishment airscrew blade.

This section discusses basic steps of creation of a three-dimensional geometrical computer model of the five blade screw, whose aerodynamic characteristics and exact geometrical characteristics initially were not known. Screw was designed for use in open circuit wind tunnel, which creates free vertical air jet. Geometry of refurbished screw blade relatively easily can be obtained by means of scanning devices [76, 77]. In this work, in order to simplify creation process of a solid model, scanned blade with length R = 0.5D was divided into sections with a step 0.1R. All cross-sections were located in the vertical plane perpendicular to the rotation plane of the screw.

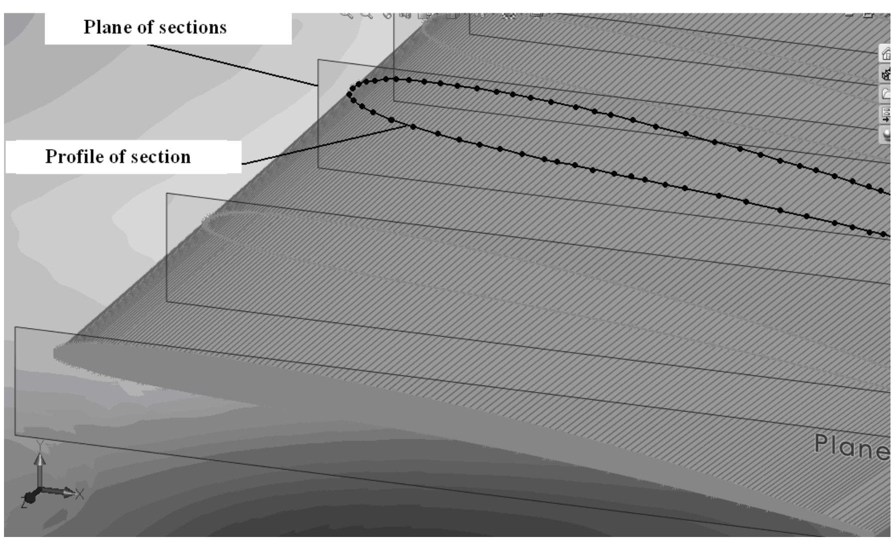
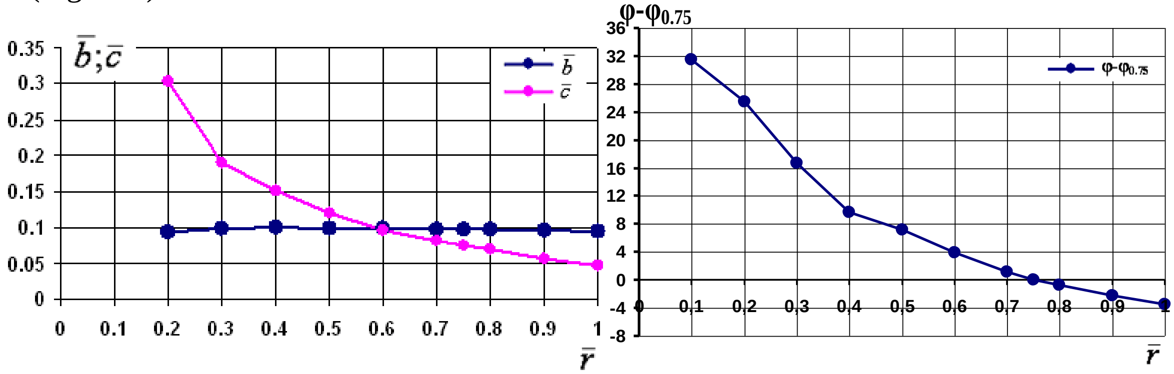


Fig.1.5. Imported surface of refurbished airscrew blade with vertical planes of sections and created a profile of section of the blade

Intersection of each vertical plane with imported surface of scanned screw was approximated by smooth curves, which formed the profile of considered section of blade taking into account its twist (Fig. 1.5.).



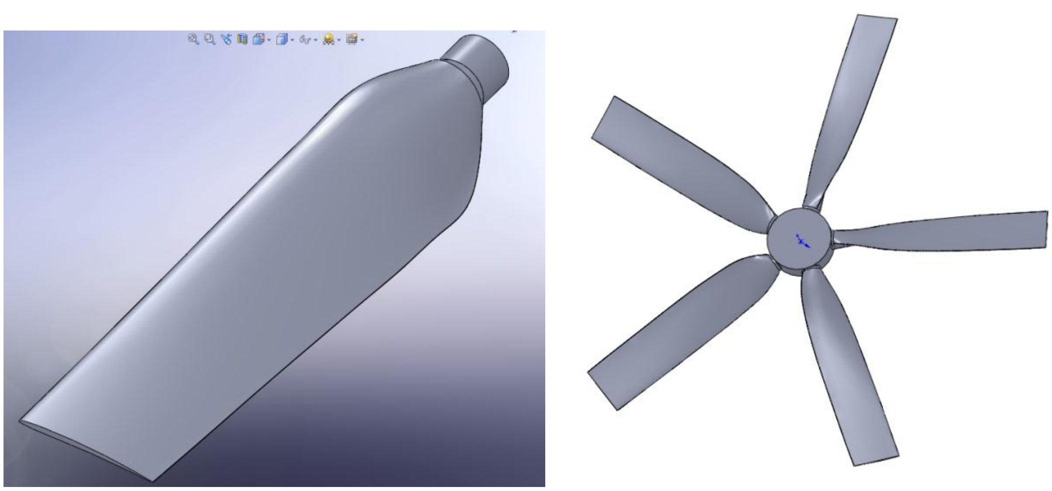
a) b)

Fig. 1.6.a) Change of relative chord $\bar{b}$ of thickness $\bar{c}$ along relative radius of the refurbished blade; .b) Change of twist along relative radius of the refurbished blade.

In Fig. 1.6.a, b is shown data about changes of renewed geometric parameters of blade: relative chord $\bar{b}(\bar{r})$, thickness $\bar{c}(\bar{r})$ and twist of blade $(\varphi - \varphi 0_{.75})(\bar{r})$.

Twist of the blade end elements are equal to approximately 3.5º, and rutt element 32º. General twist of blade from rutt to its end is about 35.5º.

Using the data obtained in SolidWorks by known geometry of sections was created a solid model of blade (Fig. 1.7.a).



a) b)

Fig.1.7.a) Computer model of the refurbished blade of screw; b Computer model of the refurbished five-blade screw

In the full-scale five-blade screw blades are fixed to cylindrical hub with a diameter 0.228m. In Fig. 1.7.b is shown a computer model of refurbished five blade screw with a hub, which must be installed in an experimental model of aerodynamic installation.

1.3.2. Selection of CAE/CFD software

In selection process of CAE/CFD programs (second task), was performed a comparative analysis of several CAE programs for calculation of gasdynamic parameters of single and multiphase flows (in particular, programs like COSMOS FloWorks, Ansys/CFX, Fluent, etc.). Based on a comparative analysis was selected program CFDesign [78], which is based on finite element method.

This program has been successfully used to calculate a wide range of problems: streamlining of cars, flows in nozzles and pipes, engineering of HVAC systems, calculations of convective heat exchange, etc.

Its main advantages: the ability to work with a wide range of graphic CAD programs (CFDesign does not have its own block of geometric modeling), a sufficiently high accuracy of calculations; relatively fast solver; ability to solve problems with moving solid boundary, solid bodies and mobile mesh; ability to analyze unsteady gas flows with variable density and heat transfer.

A key feature of the program CFdesign, is simplicity and ease of use. In the system is implemented a user-friendly Windows-based interface that allows calculating engineer to communicate with the system in "engineering" language. The whole process of model development, analysis and processing of the results is a subject to strict logic, therefore training period is significantly reduced compared to the time it takes for many modern CFD-programs.

CFdesign was created for developers of products, who are using various software for solid parametric modeling, such as PRO/Engineer, CATIA, Autodesk Inventor, Solid Edge, Unigraphics, SolidWorks and others. Supported by patented digital technologies and advanced computational methods, Cfdesign ensures proper associative relation with parts and assemblies.

Advantages of CFdesign software should also include:

- Ability to use the software by CFdesign engineer, and not CFD-specialist;
- Minimum time necessary for input data preparation and review of results;

- ➢ CAE software CFdesign is integrated into CAD software SolidWorks as an additional module and is running on parallel geometrical model, which supports all changes of geometry, made in the CAD program.

Functional possibilities of CFdesign:

- ➢ Compatibility with 3D CAD software: Autodesk Inventor, SolidWorks, Pro/ENGINEER, SolidEdge, CATIA and Unigraphics, support of formats ACIS (. sat) and Parasolid (.x_t) for compatibility with other systems (such as COMPAS-3D);
- ➢ Automatic creation of finite element mesh;
- ➢ Wide range of physical models and materials;
- ➢ High-speed computation on personal computers;
- ➢ Collaboration on projects (data transfer for the analysis of the stress-strain state in the strength codes - ANSYS, Abaqus, COSMOS, Mechanica and Nastran);
- ➢ Multifunctionality of system of analysis of results and reporting.

Process of using of CFdesign can be summarized as follows: digital prototype of product (3D model) is developed in 3D CAD, and then transferred and analyzed in CFdesign. In addition, CFdesign allows generating computational domain (for internal and external problems.) Definition of initial data is realized by introduction of required values on respective tabs. Intuitive algorithm of introduction of initial data and performing of calculations allow to engineer with basic knowledge of the English language very fast to perform necessary calculations.

Because during installation CFdesign is integrated in CAD system (e.g., Autodesk Inventor, SolidWorks, etc.), this allows to transfer the model directly from the CAD working window without additional creation of computational domain. This link is directed to get fast results, analysis of options and selection of optimal solutions. In addition it is possible not only change the calculation parameters, but also to make constructive changes.

CFdesign Motion allows studying of performance characteristics of mechanical prototypes with moving parts (pumps, compressors, turbines, fans). In order to use CFdesign Motion must be licensed for CFdesign Base Analysis System and CFdesign Advanced.

CFdesign is suitable for a wide range of users, from beginners to advanced, and for those who have to analyze the flow and heat transfer of designed products from time to time.

Application of CFdesign in technical preparation of production of fittings, pipeline components, pumps, compressors, heat exchangers allows passing from manufacturing and testing of prototypes to research of digital prototypes of new products, thus reducing the costs and quickly implementing high-quality products in market.

1.4. Features of formation of computational mesh and completion criteria of calculation.

One of the most important phases in finite element analysis is to build on the model the finite element mesh [78-81].

In the program CFDesign for three-dimensional tasks are used tetrahedral, hexahedral, pyramidal elements and wedge elements. Each node of element has 5 degrees of freedom for laminar flow: U, V, W, P, T, and 7 degrees of freedom for turbulent flows: U, V, W, P, T, K, ε. For construction on finite-element model can be used not only one, but several types of elements [55].

1.4.1. Main features of the construction of finite element mesh.

Main features that should be considered during construction of finite element mesh:

- If size of the linear finite element is smaller, then number of elements in the model is greater, the time of computation increases significantly, and errors of analysis decrease. However, errors do not decrease to zero, since an increase of number of elements accumulate rounding errors of calculation results made by computer.
- Finite element method is an approximate method, the accuracy of which depends on correct selection of types and sizes of finite elements. For example, higher density of a mesh is required, where large gradient of each of the dependent variables is expected, or in areas of significant change in the curvature of the surface of a solid.

At the same time, a rare thin mesh can be used in areas with a more or less constant gradient of each of the dependent variables, as well as in areas that do not present special interest.

- Very important point is that in general, optimal size of finite element mesh is achieved experimentally. That means, first must be made analyzes with mesh with not more than 50,000 elements, then evaluated the features and the identification of obtained (required) parameters in selected areas of high gradients. After that finite element mesh in the identified areas either could be increased or decreased. These test calculations are carried out as long as the results do not depend on the parameters of finite element mesh. CFDesign program has two primary methods of constructing of a mesh [55,78]:

- *arbitrary,*
- *ordered.*

Arbitrary mesh is constructed automatically and adjacent elements may significantly differ in size. Ordered mesh is constructed by dividing the geometric models to some number of parts.

To save time, power and RAM for resolution of problems investigated in the thesis, the best option of finite element mesh generation is ordered method.

Very important point in formation of finite element mesh is an implementation of diagnostic model. Diagnostic function allows identifying the problematic surfaces or surfaces that are, for example, extremely small relative to rest of the model. These problematic areas are automatically indicated by arrows in the graphic window CFDesign (Fig. 1.8.).

In these problematic edges or surfaces, as in other parts of the model, it is necessary to specify finite element mesh, which can significantly increase number of elements. For example, on surface of the model rectifier was defined mesh with size 0.1m, but identified problematic surfaces have a maximum size of 0.03 m, what respectively increase number of elements by 30%.

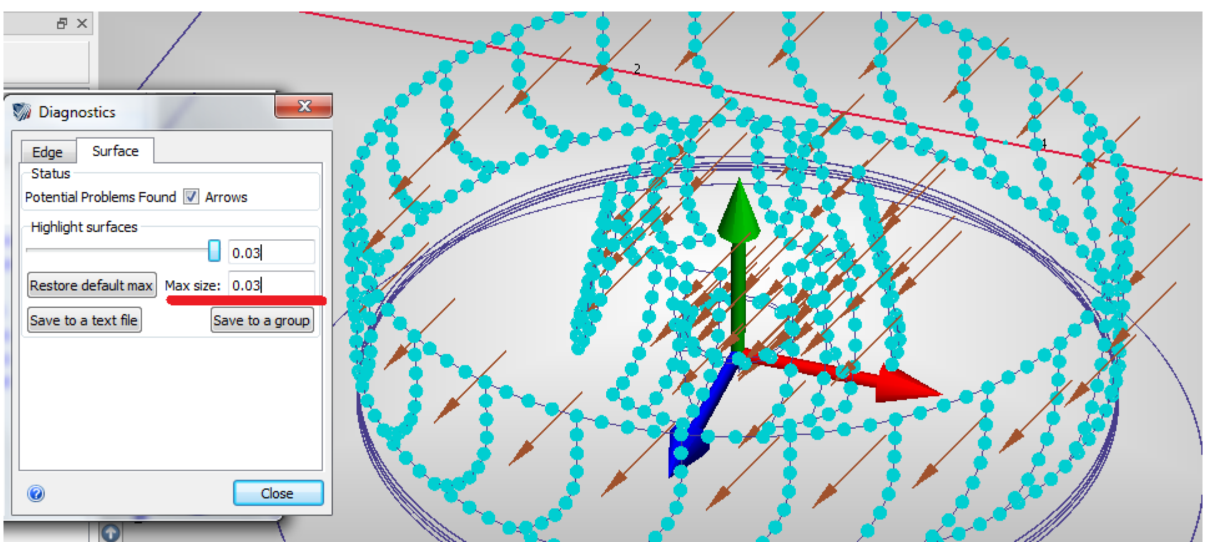


Fig. 1.8. Diagnostic function

Folowing the results of diagnostics it is necessary to make a decision:

- ➢ To reduce the overall mesh of the whole element to the maximum size of the problematic surfaces or edges;
- ➢ To specify the local mesh on these problematic surfaces or edges;
- ➢ To simplify analytical model and to eliminate these surfaces or edges from calculation.

➢ It should also be taken into account impact of these problematic surfaces or edges on physical process of the problem being solved. In our case, if there do not occur flow separation phenomena worsening flow pattern at the outlet of rectifier, these edges do not affect physical process of the problem. Therefore, in the absence of flow separation the best option is exclusion of edges from calculation process.

Note that for each element of model of aerodynamic stand (earth, engine, aerodynamic screw, rotating region, rectifying device, space that surrounds the installation and includes area of distribution of air jet) are defined their own parameters and dimensions of the finite element mesh.

Considering that in these problems interested are aerodynamic parameters of air movement, and not the internal stresses of structural elements, in areas of distribution of air flow are defined parameters of volume mesh, but on solid-state mobile or fixed elements - surface mesh.

Moving Solids

The fluid region surrounding a moving solid (and in the intended path of the solid) are areas in which the mesh should be focused. The fluid gradients that occur as a result of a moving solid can be quite severe, and the mesh must be fine enough to capture them.

The moving solids modeling, assume that the fluid that comes in contact with a solid will take on the instantaneous velocity at the point of contact that is applying a no-slip boundary condition in this situation.

To allow for arbitrary motion, a local coordinate system is placed within each moving part. It is assumed that the part does not move with respect to this local coordinate system. Rather, the local coordinate system moves relative to the global coordinate system [55, 78].

Rather than making a new mesh at each time level, the solid mesh is allowed to pass right through the fluid mesh. When a fluid node is “masked” by a solid element, then the velocity of the closest solid node is applied to this masked fluid node. In our diagram, the fluid node marked with the arrow is being masked by the solid element which is at the corner of the moving part. All the fluid nodes within the boundaries of the moving are considered “masked” and as such have their velocities controlled by the moving solid.

When models are initially constructed, many times these moving solids are embedded within the fluid. When the solid starts moving, we need a fluid region to appear in the area that the solid vacates. To handle this situation, all solids that are constructed such that they are in contact with the fluid, will have fluid elements and nodes automatically added. That is referring to this action as replication. When models are constructed such that the moving solids are outside the fluid region, then no replication will take place

As an example, below are given optimal parameters of finite element mesh of structural elements of particular stand:

Volumetric elements:

Rotating region (overall dimensions D = 3,8 m): 0,08 m

Computational domain 1 (overall dimensions D = 8m. H = 16m): 0,46 m

Computational domain 2 (overall dimensions D = 4m. H = 16m) (area of distribution of the jet, which is of special interest, so finite element mesh size is about a two times lower than in the computational domain 1): 0.28 m

Surface elements:

Ground (dimensions D = 8m): 0.46 m

Engine (overall dimensions H = 2m): 0,46 m

Aerodynamic screw (overall dimensions D = 3,6 m): 0,08 m

Rectifying device (overall dimensions D = 3,6 m, H = 0,6 m): 0,1 m

It is important to note that the reported values of finite element mesh are applicable only to this particular task. If there are implemented changes in design of geometric model of aerodynamic stand, or made some changes in overall dimensions, then respectively whole process of optimal mesh size selection must conducted again.

Usually the total amount of elements depending on the task varies in range 800.000-1.200.000. Examples of computational meshes for structural elements are shown in Figure 1.9.

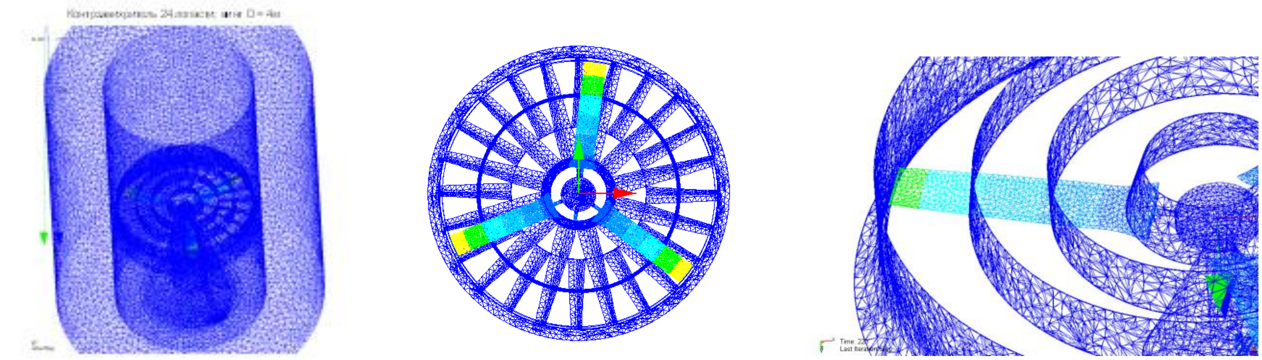


Fig. 1.9. Examples of computational meshes for structural elements

1.4.2. Definition of Boundary Conditions

For numerical simulation of operating mode of aerodynamic stand must be specified area of volume or so-called domain - space that surrounds installation and includes area of distribution of air jet Fig.1.10. Since theoretical value of domain should be much more than model, in practice, it is selected depending on capabilities of computer, its RAM and allowed computation time based on trial calculations.

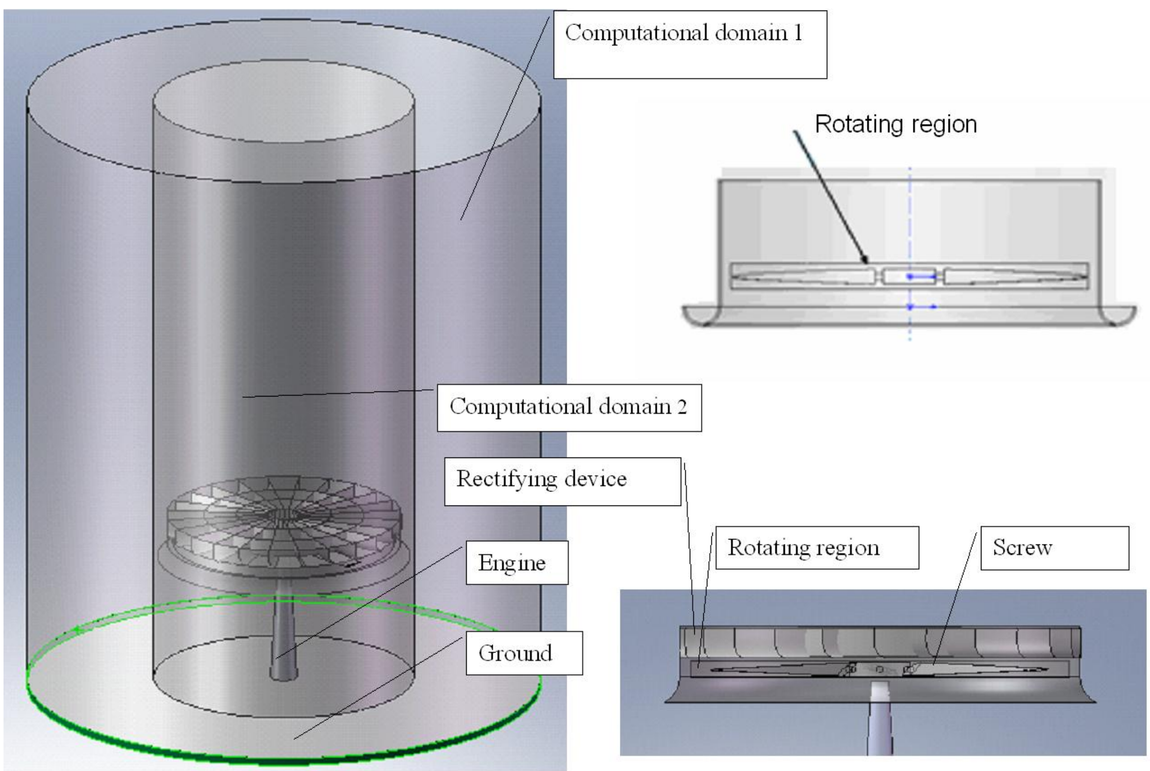
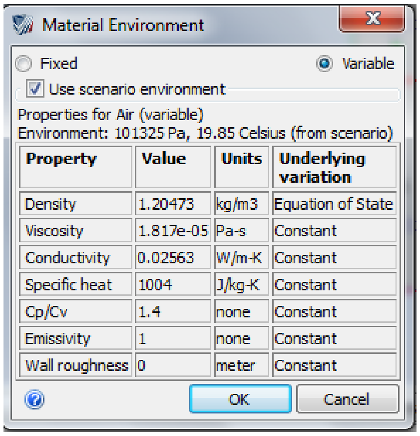


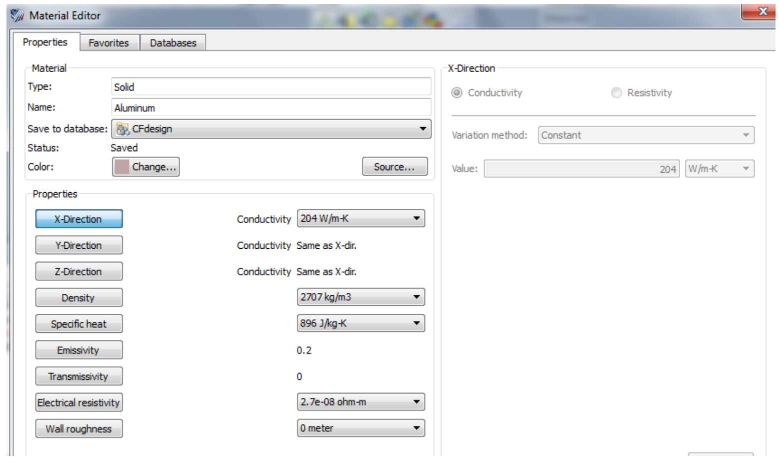
Fig. 1.10. Simplified geometric model of aerodynamic stand.

On all walls of domain is defined relative pressure P(Gage) = 0 Pa, that is equal to atmospheric pressure of 101325 Pa, at a standard temperature T = 19.8^{0}C.

In the volume of domain is defined slightly compressible air, with all its physical properties (density, viscosity, temperature, etc.). Fig. 1.11a.



a)



b)

Fig. 1.11. a) Physical properties of air; b) Physical properties of material - Aluminium

Earth, engine, inlet device and rectifier are simulated defining solid material (aluminum). As well as liquid, solid has its own specific properties Fig. 1.11.b.

Program CFD on all solid walls automatically defines condition of adhesion of viscous fluid (Vx = Vy = Vz = 0), it means that velocity is equal to zero.

1.4.3. Features of calculation methods of problems with moving solid boundary

Main feature of CAE software CFdesign is its ability to solve problems with moving solid boundary, solid bodies and movable mesh (paragraph 1.4.1.). Program has a function of *Motion*, which makes it possible to analyze an interaction between moving solid bodies and fluids surrounding its body [55, 78, 82, 83].

CFdesign allows simulating different types of motion - linear, angular, combined, and rolling. Movement of structural elements can be caused by the flow (for example, rotation of turbine blades), or created by a flow (rotation of pump impeller). In Fig. 1.12. is presented configuration dialog of the rotating region (impeller), which allows to define speed of rotation, driving torque or free spinning.

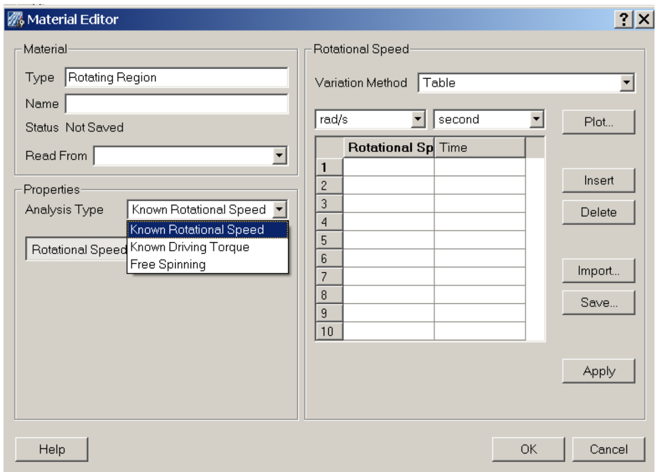


Fig. 1.12. Configuration dialog of the rotating region.

The CFdesign rotating machinery capability analyzes rotating devices using a locally rotating frame of reference. This region completely surrounds a rotating object, and is called the *rotating region.*

Areas in the model that are not rotating are analyzed in a static (absolute) frame of reference. These regions are called *static regions*. (Obviously fluid in a static region can move, but the volume itself does not rotate in space.)

The following points summarize the geometric considerations for setting up rotating analyses Fig.1.13.:

- All rotating objects must be completely immersed in a rotating region. Such a region will rotate using its own relative rotating frame of reference.
- The mesh that is generated in a rotating region will physically rotate along with the parts that are immersed.
- Immersed parts can be modeled as voids in the rotating region, or they can be included as solids. (Solid objects in a rotating region will rotate at the same speed as the rotating region.)
- The interface between a rotating and a static region is called the periphery zone. Within a periphery zone, the outer element faces of the rotating region will slide along the neighboring element faces of the static region.
- The shape of a rotating region needs to correspond (loosely) to the shape of the rotating device. Rotating regions are usually fairly simple cylindrical shapes. This allows the element faces on both sides of the periphery zone to "fit" together easily.

- The rotating region should extend to roughly the mid-point between the outer blade tips and the closest point of the surrounding non-rotating wall.
- Do not apply any boundary conditions to nodes on the periphery zone. Care should be exercised when constructing fluid geometry to avoid such a condition.
- Rotating regions from multiple rotating components must not overlap. Devices such as gear pumps or the beaters of a kitchen mixer cannot be modeled with the rotating machinery capability because their rotating regions overlap.
- All rotating devices must have a rotating region and a static region that interact via the periphery zone. In other words, a rotating region cannot directly contact a non-rotating solid region, even if the solid is not inside of the rotating region. An example is a solid annulus surrounding the outside of rotating region. The result will be that the solid annulus (which is supposed to be static) will rotate. The resultant images will be much unexpected.
- Objects within a rotating region that have a uniform cross-section that satisfy the requirements for mesh extrusion can be extruded. The mesh inside of the rotating region, however, cannot be extruded.

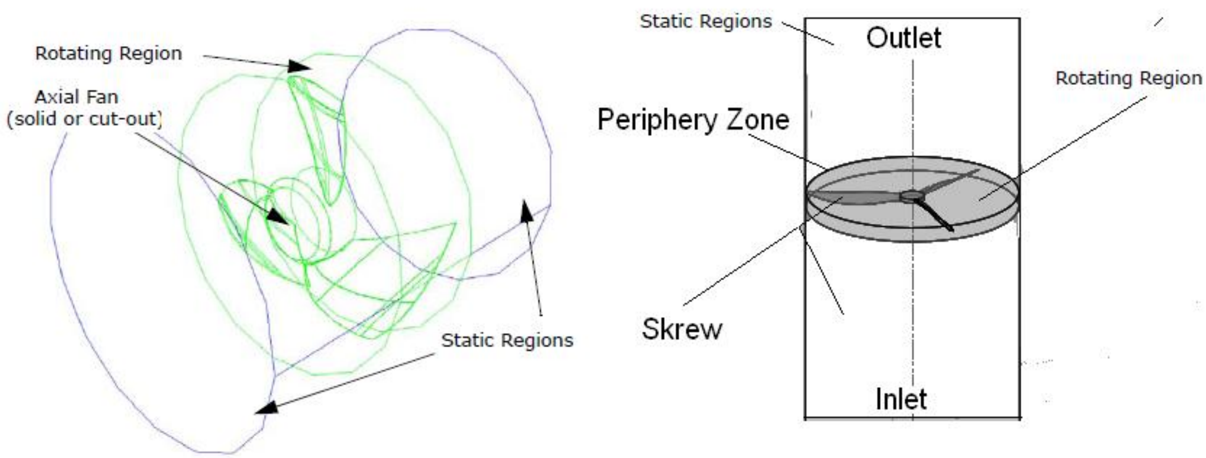


Fig. 1.13. Graphics illustrate these principals.

In case of rotation of a screw with a constant angular velocity, computation time of the task can be significantly reduced the by placing the screw in the so-called Rotating Region - closed cylindrical region fig.1.10, whose axis coincides with the rotation axis of the screw. In process of calculation (analysis), it rotates relative to a specified axis and any solid bodies inside the region rotate together with it.

Screw inside rotating air-permeable region is modeled as a solid body. In the process of calculation aerodynamic screw and region rotates relatively to the axis with the same angular velocity, which is defined by a given number of screw revolutions n RPM.

The rotation of the region can be defined by specifying an angular velocity, torque, or initiated by applied aerodynamic forces. All rotating equipment must be placed into rotating regions and modeled as hollows or solid bodies. Rotating region can not directly contact with the immovable area (stator). Transition zone between the rotating and static regions is called peripheral zone.

In the process of creation of computer model with rotating elements should be considered that the program CFdesign does not allow contact of rotating and stationary parts of installation, as well as changes in boundaries of solid bodies and shape of channel walls during calculation process.

Also for some tasks instead of screw can be considered a model fan. This model is a rotating disk for which is defined axial velocity along the radius of rotation of screw blades. Disc is placed inside the ring.

Rotating region helps to analyze operation not only of aerodynamic screws, but also of rotating machinery, such as pumps, blowers, turbines, and mixers.

After all the boundary conditions and materials are specified, geometrical parameters of mesh are defined, for nonstationary problem must specify computational time step, as transient mathematical model is discretized both in space and in time.

All problems with rotating mechanisms (bodies) are calculated as unsteady (Transient), and for calculation is using the following formula:

$$\Delta\theta = 6 n_m \Delta t \qquad (1.15)$$

Where, $\Delta\theta$-body rotation angle (grad); n_m – number of revolutions per minute (RPM); Δt - time step (s)

Therefore, for calculation of investigated model with natural geometrical parameters and a cylindrical domain with dimensions ~ 8x8.5 meters are required ~ (10^5 - 10^6) nodes of mesh and initial time step is about 4×10^{-4}s. In case of a three-bladed screw with number of revolutions 1200 RPM one revolution of model of the screw takes 0.05s. During one time step Δt blades are fixed, and then rotated by an angle $\Delta\theta = 6 n_m \Delta t$.

1.4.4. Criteria of completion of calculation

After defining all input parameters of the task, now it is necessary to define the criteria on the basis of which calculation of task will be completed [51, 55, 69].

For stationary problems main criteria is maximum number of iterations, which must be performed in the calculation. This number of iterations is determined for each task individually. For the stationary task it depends on computational mesh.

Since solution of stationary task is determined by its steady state-in time, it is important to select right moment to complete the calculation, i.e. to determine on time whether the decision will change when the next time step (iteration) will be performed (as mentioned above, in the solution of stationary problems is used a local time step, i.e. in the same time step time in different points in space generally is different) or not, and if not, then stop the calculation

In case of premature termination of calculation the solution still would not be obtained, for example, the values of the physical parameters will depend on the defined initial conditions, while for stationary solution such dependence, of course, is absent, and at too late termination of calculation time since steady state a solution until termination of calculation will be spent for nothing.

For unsteady tasks the main criteria is maximal computational time of task (duration of calculation in physical time). For those tasks this time depends on properties of fluid and flow, and exactly in what time period flow will cross computational domain. (If the task is unsteady and has time-dependent and non-periodic solution, calculation is usually performed until a specified moment of physical time).

Since the problem is solved numerically, discretization of the task is realized not only in space but also in time, i.e. the first problem is solved at some time layer, and then is realized next step in time, which is equal for the whole computational domain. These time steps in CFdesign are called iterations. After each iteration are updated visualized pictures of fields of physical parameters.

When searching for a stationary or non-stationary periodic solution of the problem, appears problem of identification of reaching of desired solution, which can be solved based on an analysis of performance of a certain criteria - as soon as it is executed - can state that such a solution is reached, and accordingly complete the calculation after the next iteration in time.

If, from a certain moment of time, the solution of the nonstationary problem is steady state, i.e., becomes stationary or periodic, then the calculation is carried out until achieving this moment of time, more precisely, to the definition of this stationary or periodic solution based on the performance of certain criteria, which suggests that such a solution is reached.

In observing process of calculation can intervene in decision making process of completion of the calculation, interrupting the calculation before automatic completion, if for some reason it is not satisfactory. For example, the CFD has sent warnings, which indicate that the continuation of the calculation is meaningless, that is, at the exit of a domain is a vortex, which significantly reduces correctness of the calculation, therefore it is necessary to extend the domain. This cause is important, because in some cases it reduces the computation time.

Therefore it is convenient to use in “Convergence Monitor”, to observe the progress of the analysis. When the analysis is complete, can be used the dialog box “Review” and there click "View Convergence Monitor” to assess the ultimate convergence and verify that the solution converged. Convergence of solutions is an important criterion for the ending of the process based on an assessment of movements of interested physical parameter. In the “Convergence Monitor” are shown plots of total values of each physical parameter. In these graphs in abscissa are indicated this time steps, and in ordinate value of any physical parameter (pressure, temperature, etc.).

The fastest way to determine whether that decision has converged is review of graph of interested physical parameter. However, since the analysis is time-dependent (transient), then the values of the physical parameters almost never become constant (unchanging over time). The best is if the convergence curves are repeated for several iterations. A good sign of convergence is that if the value of the physical parameter varies by less than 5% during the last 20% of total number of time steps. To estimate the interested physical parameter individually, it is necessary to select it from the list of physical parameters (by default all physical parameters are displayed).

Since the problem is solved numerically on a certain computational mesh, which may be too rough for this task, and in addition there are not excluded human errors in setting of initial data, then are possible cases of absence of steady state at all. These cases also must be as soon as possible identified in the process of calculation and analyzed for causes of divergence solution.

1.5. Methods of data visualization, processing and verification of results of computer experiments.

Unlike a physical experiment in which it is possible to receive only values of physical parameters measured at specific points using special sensors, computer experiment has much more possibilities. In computer experiment can simultaneously receive not only the numerical values of investigated parameters, but also to visualize them as two-dimensional and three-dimensional graphs or colored pictures, as well as the flow lines, trajectory, etc. These data can be obtained at any point on the lines, surfaces or selected volumes, located in the computational domain. It must be noted that in computer experiment formation process of digital results of calculation and visualization play as important a role as measurements and their processing in a physical experiment. A main feature is a visualization of the flow parameters, which in some cases allows reveal characteristic physical features of the flow (forms of separation zones, zones of intense vorticity, etc.), which are quite difficult to identify in a physical experiment.

Therefore in computer experiment for successful analyzing of obtained results of calculation must be performed method of data visualization, taking into account the characteristic features of considered class of problems.

The method of for in this work investigated tasks should provide possibility and ability of interpretation of results in following ways:

- Ability to present results in different types of images (wireframe, shaded, grid, etc.).
- Distribution of values of interested physical parameters (velocity, pressure, temperature, etc.) in sections and on the surfaces of the model.
- Construction of trajectories and flow lines
- Ability to determine values of parameters at point, integral and local (minimum, maximum, average) values of parameters at surface and in volume. Numerical values of investigated parameters in a table are displayed in CFD or displayed in Microsoft Excel.
- Plotting of graphs of changes of parameters along trajectories or any given direction XY during computation, and after its completion.
- graphs of changes of parameters along trajectories or any given direction XY are displayed in file of Microsoft Excel.
- Generate reports in HTML.

- Creating pictures of the distribution of parameters and trajectories in JPEG, Bitmap, GIF, TIFF or VTF formats.

Review calculation results starts by booting of corresponding file of results in CFD post-processor. This file with extension res. is saved in folder of project. For unsteady problem also are stored files with results obtained at moments predetermined during definition of settings of unsteady problem or when specifying settings for saving of results. File, which contains final results is called n.cfd, where n-name of folder of project. Intermediate results of solution of task are saved in file n.res.t x, where x - number of iterations, after which the intermediate results have been saved.

Results of operation of generator of initial computational mesh are stored in a file with extension *.set*. This file contains only information about obtained initial grid (if during process of calculation mesh doesn't adapt to the field of flow, initial mesh is not changed until the end of calculation).

It should be noted that reviewing and processing of results, is a very important aspect of proper resolution of the problem, because incorrect interpretation of obtained results can negate all efforts to elaborate and implement strategies to solve the problem.

When solver of task is running it is recommendable periodically to monitor process of calculation, at least changes of physical parameters defined as goals before the start of the calculation.

This must be done due to the following reasons:

- It is possible to intervene in decision-making process about termination of interrupting calculation before auto-complete, if for some reason it doesn't satisfy. calculation,
- If interested physical parameters do not change very much and already during the calculation is possible to estimate the values to be received upon completion.

This consequently will lead to savings in time, the process of solving of problem.

Representation in different types of images (wireframe, shaded, grid, etc.).

When visualizing results (in process of calculation or after loading of file of results) directly in the graphics window of CFD, model can fully or partially close a picture of the results, where the area of interest is inside the model. There are several ways to ensure that the model does not interfere with the perception of the results.

In the window of program CFD using Tool Bar Fig. 1.14. can be viewed pictures of physical parameters in different images (wireframe, shaded, grid, etc.).

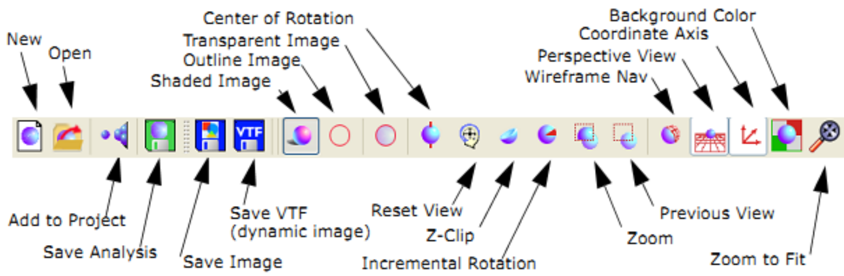


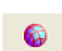
Fig.1.14. Tool Bar of CFD

 **Shaded image.** This image does not allow fully to see a picture of results taking place within the domain (model). Can be only analyzed ongoing process at the outlet of the domain. The model is shown filled. Tab.1.1.Fig. a.

 **View Lines**. **Grid view**. - applied when necessary to evaluate (analyze) parameters of obtaining finite element mesh on a whole model or its elements. The mesh lines are shown. Tab.1.1.Fig. b.

._**Outline Image -** this mode preserves understanding of e geometry of the model and at same time maximally opens to review results. Displaying only lines of intersection of surfaces in some cases does not disturb view of unnecessary details, in others may hinder the perception of results. The outline of the model is shown. Tab.1.1.Fig. c.

. **Transparent** – to define properties of transparency of models is possible as for the individual surfaces, as well for structural elements or entire bodies. This works in conjunction with a shaded image, and makes the model transparent. Tab.1.1.Fig. d.

 **Show Mesh** - this kind doesn’t allow to see completely picture of finite element mesh inside a domain (model). But when used with function **Z-Clip and Crinkle Cut**, allow to evaluate quality of mesh in three dimensions. Displays surface mesh. (Shaded Image should be enabled.). Tab.1.1.Fig. e.

 **Peel by Surface** - allows extinguishing of individual layers of surface that may hide the calculation results. Toggles between surface and volume blanking (with the right-mouse-button). Default is volume blanking. (Note: surface peeling is not available for parts with assigned motion

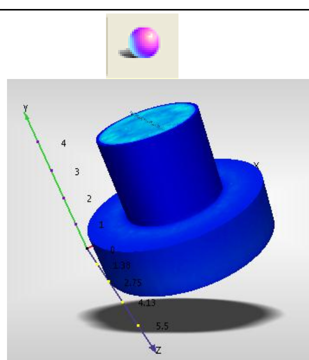
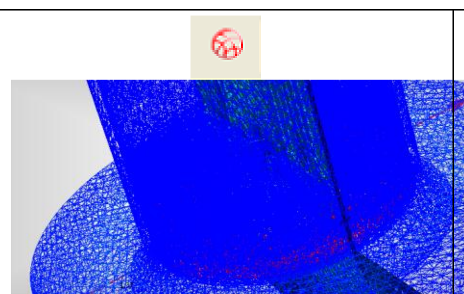
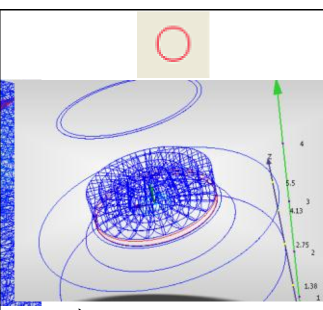
(Moving Solids). Toggles between surface and volume blanking (with the right-mouse-button). Default is volume blanking. (Note: surface peeling is not available for parts with assigned motion (Moving Solids).Tab.1.1.Fig. f.

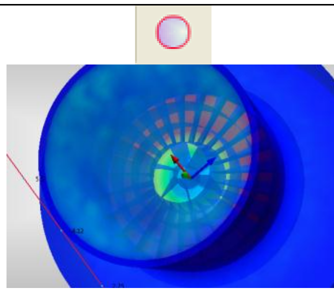
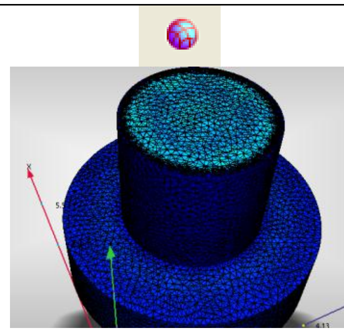
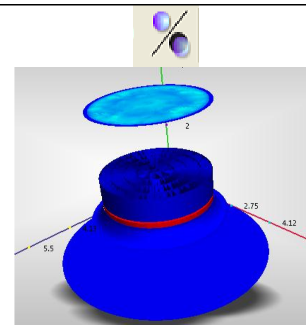
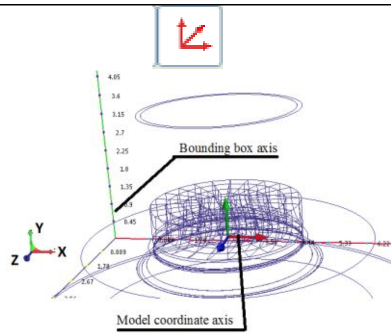
 **Coordinate Axis Toggle -** limiting axle of model surrounding the graphic object and specifying its size. Using slider is possible to specify a number of points on coordinate axes X, Y, Z (from 4-20 points). Numeric value of coordinate is calculated automatically, depending on the overall dimensions of model. The model also has a **Model coordinate axis**. This is a coordinate axis of model and is defined in CAD program during process of building and assembly of model. This axis is very important when solved problem with moving or rotating bodies. When enabled, the model oordinate axis and the axis bounding box are shown. Note: Several functions in the Results task as well as Monitor Points use model coordinates. These coordinates are referenced from the model coordinate axis, which is positioned at the model origin . Tab.1.1.Fig. g.

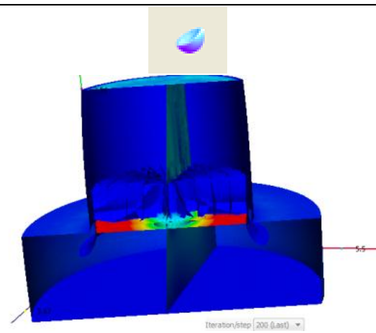
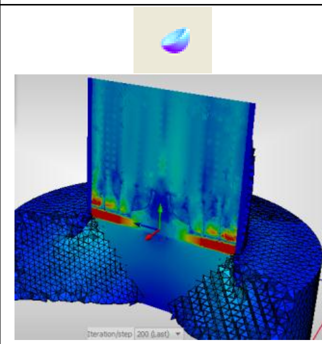
 **Z-Clip and Crinkle Cut** - allows to cut model in desired section, but it does not "cut" picture of results. Use of the cut makes picture particularly graphic when displaying parameters in sections and on surface of the model. Tab.1.1.Fig. k. When combining the **Show Mesh and Z-Clip and Crinkle Cut** is possible to analyze quality of the finite element mesh in three dimensions. Launches the Z-Clip dialog: Use the slider bars to clip into the model. Parts of the model that are between the plane and the user are made invisible. The following is an example of a clipping plane: For some models with close parallel surfaces, reducing the Mesh Factor increases the visual clarity of the clipped display. Crinkle Cut is a way to view the mesh inside of the model, and is available in Results Viewing. Tab.1.1.Fig. *l*.

Main types of used images are shown in Tab. 1.1.

Table. 1.1. Viewing Results

 a) **Shaded Image.**	 b) **View Lines**.	 c) **Outline Image.**

<table>
<tr>
<td>
d) Transparent.</td>
<td>
e) Show Mesh.</td>
<td>
f)Peel by Surface.</td>
</tr>
<tr>
<td>

g) Coordinate Axis Toggle.</td>
<td>
k) Z-Clip and Crinkle Cut</td>
<td>
l) Z-Clip and Crinkle Cut.</td>
</tr>
</table>

Distribution of values of physical parameters (velocity, pressure, temperature, etc.) in sections and on surfaces of model

Picture of distribution of values of physical parameters in sections and on surfaces of model

In the dialog box Results using the tool Cut Surface in the graphic area of program CFD can be viewed (visualized) distribution of values of interested physical parameter (velocity, pressure, and flow swirling, etc.) in a selected section.

Example of use of functions of the panel of tool Cut Surface (Fig. 1.15.).

Using the function **Add**, in the **Tools** on aerodynamic model of the stand were set two planes of section - **Section 1** and **Section 2**, with the turn respectively to Z axis by ± 135°. Part of model till given section planes were exstinguished reflection using the **Clip cuts**. In the Section 1 using the two functions **Shaded** and **Show Mesh**, convenient to analyze parameters of the finite elements computational mesh. In the Section 2 in the **Tools** was selected physical parameter of vertical component of velocity *W-Velocity*, and using functions **Color by Scalar** and **Show Mesh**, can analyze distribution of vertical component of velocity in computational mesh of finite elements. Each color according to the **Scale** has its numerical value given in front of each color.

Minimum and maximum values, as well as number of colors are defined automatically or manually in tab **Setting**. At the outlet section using parameters **Color by Scalar** and **Vector**, can view distribution of velocity vectors in a given section. Color of vector corresponds to the color and the numeric value of **Scale** of distribution of a physical parameter - in this case the velocity **W-Velocity**. In tab **Vector Spacing** is adjustable such a parameter as "distance between vectos" - the distance between the points at which are built velocity vectors on a picture of velocity distribution.

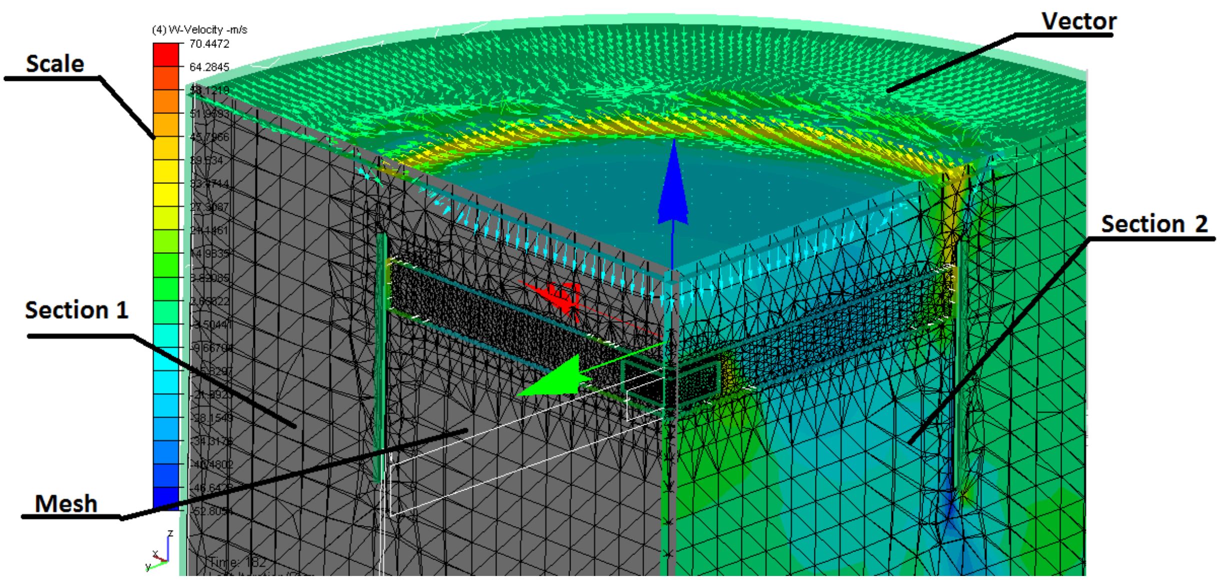


Fig. 1.15. Distribution of the vertical component of velocity W-Velocity (MPS)

Construction of trajectories

In the **Cut Surface/Tools** section window plane is constructed with respect to which is necessary to build a trajectory. Afterwards on the **Scalar** tab is defined interested physical parameter, such as the vertical component of velocity Vy-Velocity (Fig.1.16.). (Taking into account that in the work mainly will be considered axisymmetric tasks, the main interest presents only the Circular Grid). To construct trajectory go to the tab **Trace** and **Seed Point** window is selected **Circular Grid** and defined initial number of points through which will be run the trajectory. First value is number of points in the circumferential direction, and the second value is the number of points in the radial direction. In the graphical area on the active plane of section are marked two points: first point defines the center of the circular mesh, and the second - its outer radius. That means, if are specified parameters 20x20, then the circle is divided by lines into 20 parts (which determine the radius of the circular mesh) and on each line will be marked 20 points, through which will be run the trajectory. Using the **Add Trace Set** for each trajectory is defined its serial number **Trace 1, Trace 2 , Trace n**, where **n** - number of trajectories and time of its processing (building) **Residence Time**.

Color of trajectory is determined according to the color **Scale**, and particular for that color numerical value.

If necessary to remove the trajectory, the **Delete** button must be used.

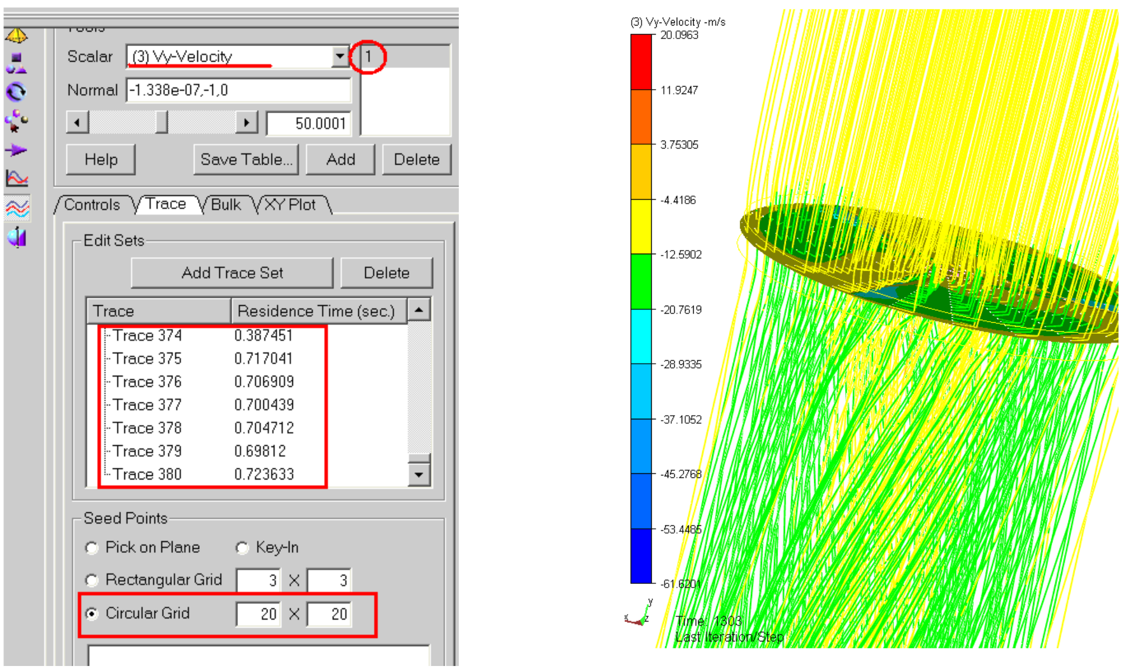


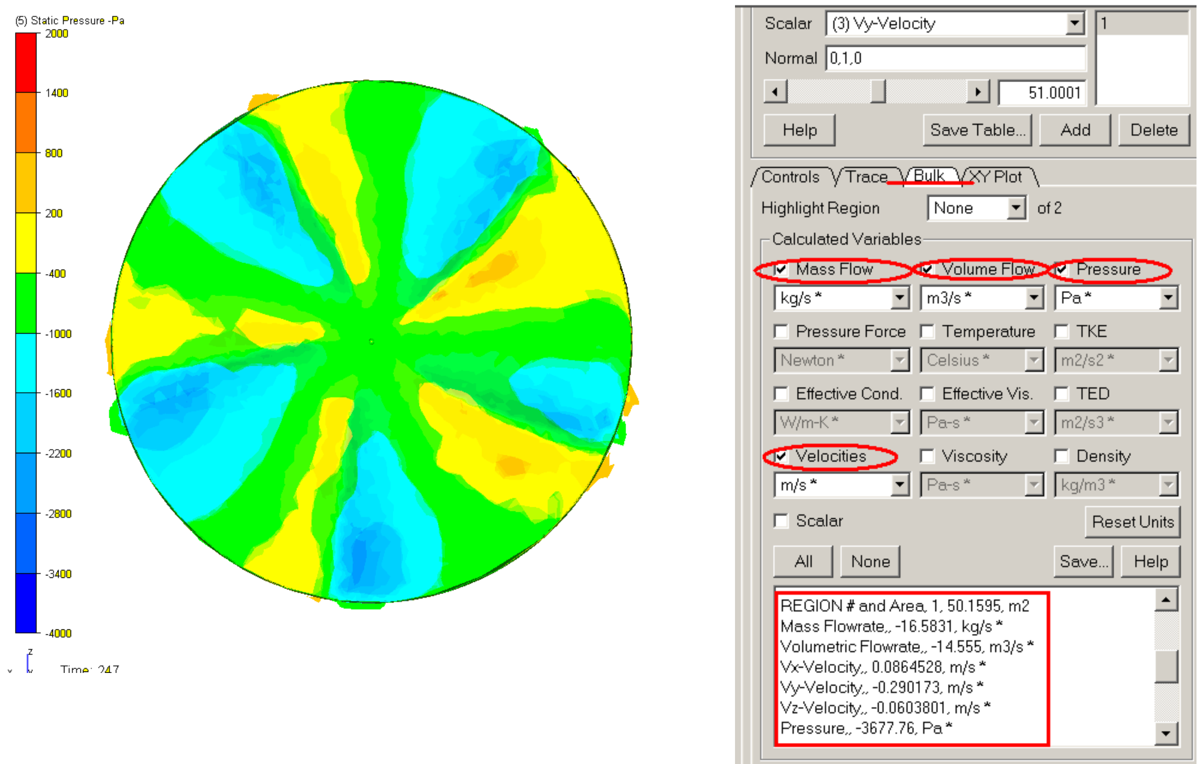
Fig.1.16. Construction of trajectories.

Quite difficult is on a visualized picture to determine distribution in area of the section, for instance Static Pressure Fig.1.17.a., the average numerical value of physical parameter. Picture is more a visual perception of the zones of high and low pressure, which need to be addressed.

Therefore the average numerical value of distribution of main physical parameters (mass and volume flow rate, pressure, velocity components, etc.) Fig.1.17 b. in a given plane of section is convenient to determine using a function **Bulk**.

After check the box in front of the desired parameter in the dialog box of function **Bulk** is obtained its average numerical value in a given section. During the process of calculating the value may vary depending on time (iteration) and physical processes occurring in a given time.

Once the calculation is complete, these values will correspond to the last time step (iteration). Therefore, in order to qualitatively evaluate dynamics of changes of values of given parameters, it is recommendable to save them in Excel. To save in Excel, CFD program uses extension *.csv*.



a) b)

Fig. 1.17. a) The visualised image of static pressure; b) Average numerical value of distribution of main physical parameters.

Example of using the function Bulk

To determine the pressure difference must perform the following steps:

Step 1. In the toolbar **Cut Surface** must install two plane sections: one plane Section 1 in front of aerodynamic screw and a second Section 2 behind it, respectively on a selected distance from the rotation axis of the screw.

Step 2. In the **Bulk** tab **Calculated Variables** check in checkbox in front of **Pressure**.

Step 3. to activate needed plane of section, and in the working window **Bulk** get the average in the plane of the section numerical value of pressure.

Step 4. Calculate the difference between the numerical values of pressure in Section 1 and Section 2, and thus the difference between these values is the required pressure drop.
In a similar way, we can determine the mass and volume flow rate, as well as changes in the parameters of velocity components Vx-Velocity, Vy-Velocity, Vz-Velocity.

Construction of graphs of parameters along the trajectories or any given direction XY during computation process, and after its completion.

In the toolbar **Cut Surface** using the tab **XY Plot**, can build a graph of interested physical parameters (Fig. 1.18.). The graph can be constructed in several ways: through the points selected graphically on the cutting plane (**Add by Picking**), through the points specified by coordinates (**Add by Key-In**) or through the points saved from the previous graph (**Read From File**).

To construct the graph using the **Add by Picking** necessary to activate at least two points. Coordinates of points (x, y, z) are shown in the working window of tab **XY Plot**. Number of intermediate points is indicated in the **No of Divisions per Segment** and their minimum number is two.

Once the coordinates of the points are determined, by pressing the **Plot** in a separate window appears resulting graph with the data. Graph (XY Plot) allows viewing the change of interested physical parameters (values are indicated on the y-axis) along the given points (lines) **Parametric Distance** (values along the x-axis) in Fig. 1.18. in the interested area.

In order to assure that points for creating the graphs in different analysis (projects) were located in the same places, they are saved using **Save Points**. The best is to save them in a separate folder. To save them in Excel, CFD software uses extension .xyr.

The following graphs or in subsequent analyzes (projects) for creation of graphs through the same points is used the function **Read From File**, clicking the **Browse** button from the corresponding folder where was saved the file with the coordinates of the points, it can be unpacked in the working window of tab **XY Plot.**

In the further work is necessary to determine such coefficients as traction factor α, power factor β and efficiency factor η. In the program CFD is not incorporated automatic determination of these values. Using the function **Wall** can be determined traction force of screw.

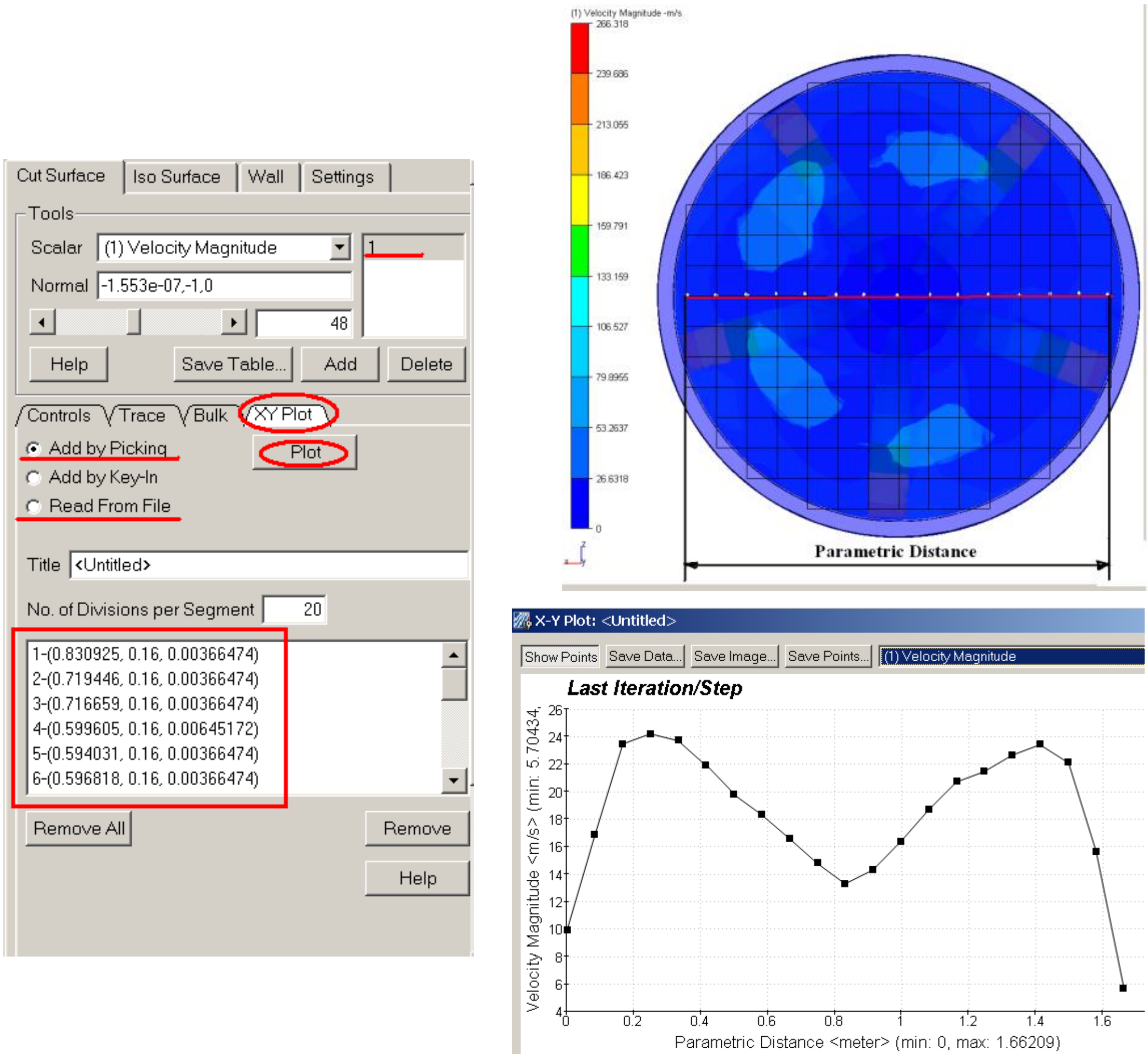


Fig. 1.18. Construction of graphs of parameters along the trajectories or any given direction XY.

In the window **Wall** (Fig.1.19.) necessary to select the tab **Selection and Result** where must check the checkbox in front of **Volume**. Then in graphical area necessary to select all screw blades and the hub. In the working **Wall** appear all marked volumes in digital form. The next step is selection of required parameters, in this case **Force** and **Torque**. Then press the button Calculate. The results can be viewed on the tab **Output**.Results are calculated for each of the selected volume separately, and average value of desired parameters is enabled in **Summary**. Further by well-known formulas are calculated relative factors of traction (14) $\overline{\alpha}$, power (15) $\overline{\beta}$, and efficiency η.

Power N_v expended in rotation of screw is determined:

$$N_v = \omega \cdot M \tag{1.16.}$$

where: M – torque expended in rotation of screw; ω –angular velocity of screw $\omega = 2\pi \cdot n_s$

n_s - momentary number of revolutions propeller

Efficiency factor

$$\eta = \frac{\overline{\alpha} \cdot V}{\rho \cdot n_s \cdot D} \tag{1.17.}$$

V –velocity of incident flow, D –diameter of screw, ρ - density of air.

To save the data in **Excel** format with the extension .csv. is used button **Write to file.**

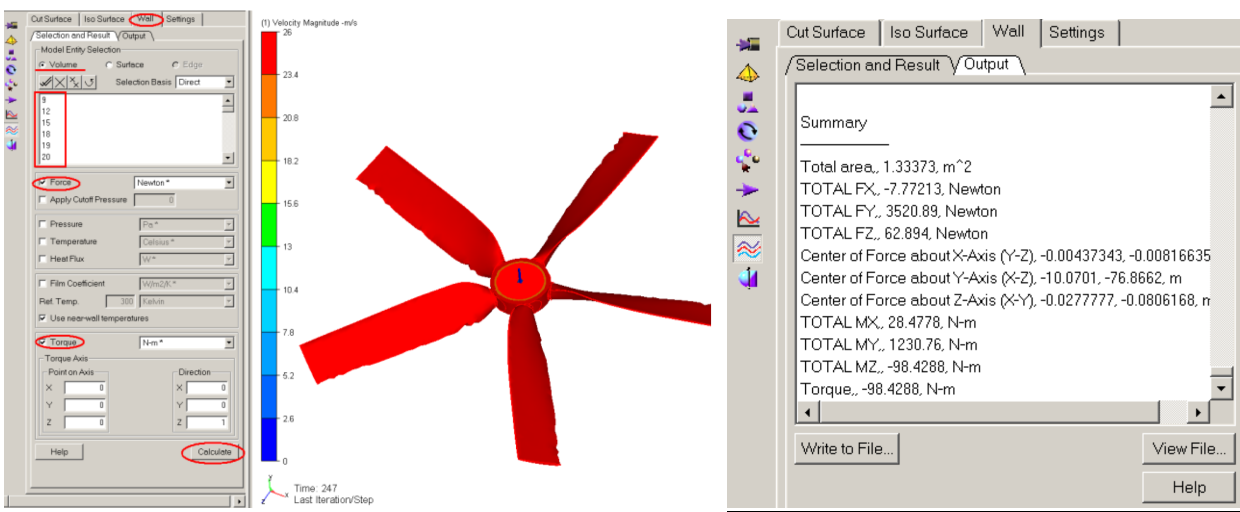


Рис. 1.19. The Wall tab of the Results dialog task provides a way to calculate flow-induced wall forces on wall surfaces of the model.

Generation of reports

Reports are generated in the dialog box "**Review**" in tab **Report.** To view the report, use the button **General Report**. The report is in HTML. If there is a need to add to report some textual information, then use the button **Create Text**, where in the new window can write, for example, the conclusions of the work done.

Report consists of:

➢ ***Introduction***, which contains information about location of the folder in the Computer, name of the project, name of its author and date of report.

- ***Model Description*** - image of the computational model. A detailed description of the given initial and boundary conditions of calculation. Information about the materials selected for each part of the model, with defined physical properties. As well as information on the finite element mesh.
- ***Analysis Summary*** - Contains tables of the results with physical parameters (minimum, maximum and average values) at the last time step (iteration). Convergence graphs of calculation results. As well as information on the elapsed time of calculation, and use the computer's memory.

Creation of pictures of distribution of parameters and trajectories in the JPEG, Bitmap, GIF, TIFF or VTF formats.

In horizontal Tool Bar, selecting Save Image button can save any of the interested types of model. Available are extensions JPEG, Bitmap, GIF, TIFF.

To create an animated form in the dialog box **Review** need to go to the tab **Results** and select the results needed for the animation. Then, go to the tab **Animate** and there set the interval of switching of frames of film (in milliseconds). Created animation can be saved in the format .vtf, by clicking **Save dynamic view** in horizontal **Tool Bar**.

Verification of results of computer experiments

Reliability criteria and validation (verification) of the solution obtained by the calculation method, consists of two evaluations:

- Evaluation of accuracy of solution of a mathematical problem, corresponding to used mathematical model of a physical phenomenon.
- Evaluation of accuracy of simulation of physical process used in computational method of mathematical model.

Accuracy of solution of a mathematical problem is defined by mathematical methods, regardless of the adequacy of mathematical model to calculated physical phenomena.

Evaluation of this accuracy is based on the analysis of the mesh convergence of solution of problem based on solutions obtained on different meshes.

Once obtained satisfactory in accuracy solution of mathematical problem, next step is to evaluate accuracy of simulation of physical process by the mathematical model used in method of calculation. For this, obtained solution must be compared with the experimental data (taking in to account their uncertainties combined from measurement error and the error of experiment purity, i.e. the absence of parasitic effects). Naturally, because of the limited amount of experimental data, for such verification it is recommendable to select them as close as possible to engineering problem being solved. Accordingly, in addition to this problem, have to be solved some additional test task, for which experimental data are available, and with which can be made a comparison of the calculated data. This significantly increases reliability of accuracy of numerical solution of standing in front of you an engineering problem, so that the additional costs of resources and time will be repaid in the future, including the for solution of similar engineering tasks.

CHAPTER 2.

NUMERICAL CALCULATION AND OPTIMIZATION OF CHARACTERISTICS OF MAIN ELEMENTS OF THE AERODYNAMIC STAND FOR THE FREE MANNED FLIGHT. [84,85,86,87,88]

2.1. The purpose of this chapter and the formulation of the task

Purpose of this chapter is development of methods of computer simulation and optimization of geometric and aerodynamic parameters of main elements of studied aerodynamic stands. These methods by means of 3D parametric computer simulation software SolidWorks and CFDesign packages allow creating of different modifications of 3D geometric models and performing their virtual blowing, ensuring optimal geometric and aerodynamic parameters of developed stands. Method must permit to obtain relatively quickly results satisfactorily matching with the data of natural experiment.

Aerodynamic screw is a main element of generators of large diameter free vertical jets. Computer simulation method, including creation of a numerical geometric model of the screw and calculation process of its aerodynamic characteristics significantly depends on the geometric parameters of elements of the stand and its composition scheme. In this chapter are discussed several versions of methods of calculation of individual main elements and recommendations for their design, based on which can be composed and analyzed the characteristics of different types of aerodynamic stands.

*Version **N1.*** Method of calculation and numerical validation of the aerodynamic parameters of the screw with the known geometry

*Version **N2.***Computer simulation of aerodynamic parameters of the refurbished screw.

- The use of a implified disc model of the fan (aerodynamic screw);
- Methods of computer simulation of system: inlet device - screw - gasdynamic channel.

*Version **N3***. Comparison of the results of numerical calculations with the results of field measurements.

*Version **N4***. Computer simulation of aerodynamic parameters of rectifier with profiled blades.

Version **N5**. Development of recommendations for designing of elements of the aerodynamic stand.

2.2. The method and numerical verification of the aerodynamic characteristics of the screw with a known geometry.

In practice, often arise cases when for an existing aerodynamic screw are not available data about detailed geometrical characteristics of the blade and the aerodynamic characteristics of the screw. Aerodynamic characteristics of a computer model of the screw can be calculated using known CFD programs based on solving of non-stationary Navier - Stokes equation systems [51, 54-59] , (Chapter 1, paragraph 1.2.).

To verify the developed method and the possibility of calculating the aerodynamic characteristics of the screw with a selected set of CAD/CFD software SolidWorks and CFdesign has been chosen three-blade screw of type Clark Y with known [17,18,20,21], geometric (Chapter 1, paragraph 1.3.1.1.1.) and experimental aerodynamic characteristics (Fig. 2.1.). Physical experiment in works [17,18,20,21] was performed in a wind tunnel with a diameter of working area 6.1m and with flow velocities V = 0 - 160 km/h (V = 0 – 44,4m/s.). In the experimental device, the screw is installed on the nacelle, fixed on the wing compartment, which geometrical parameters in works are not specified.

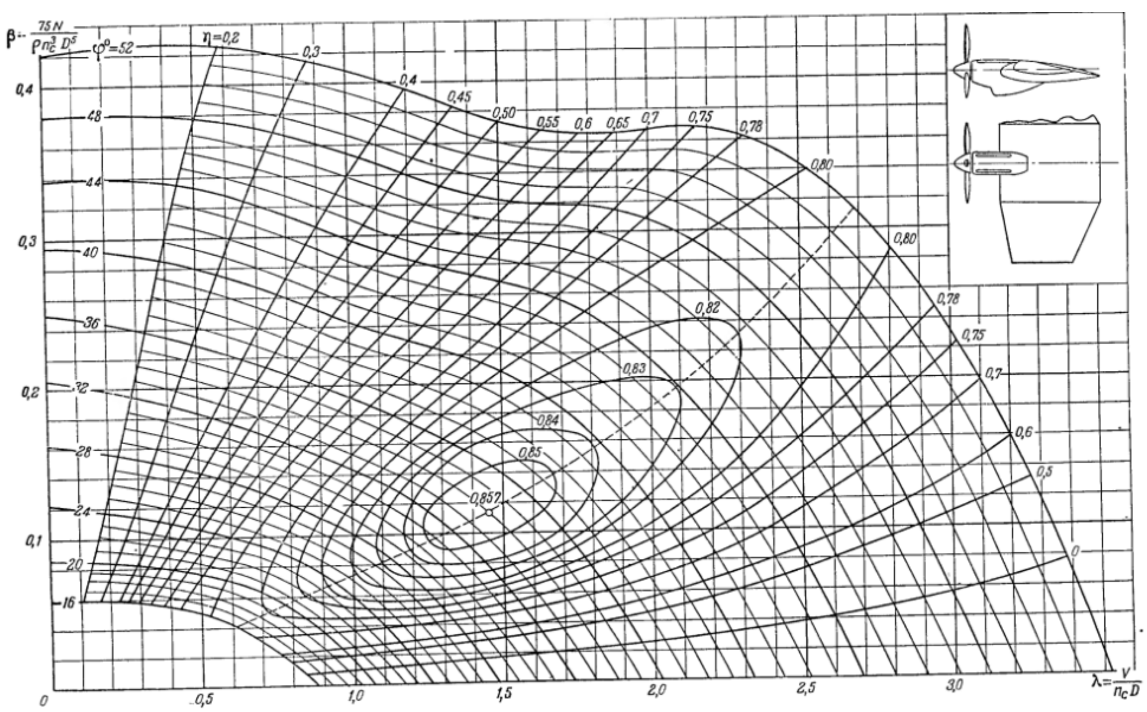
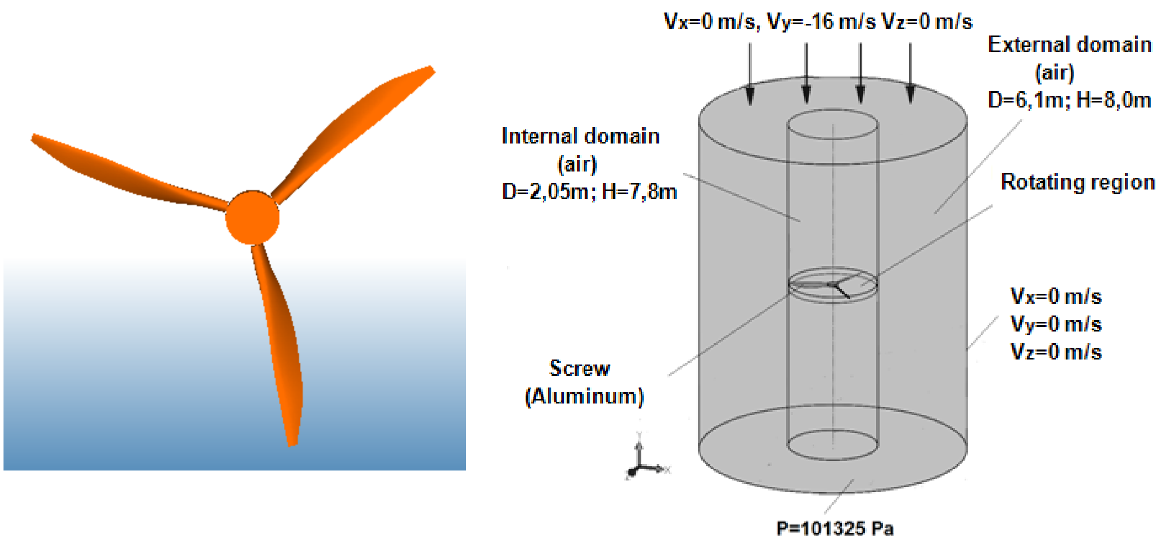


Fig. 2.1. Aerodynamic characteristics of three-bladed screw.

In computer experiment in CAD program SolidWorks based on the known geometrical data was created the 3D geometric model of the screw with Clark Y type profiles (method of construction of three-dimensional geometric model of the screw described in Chapter 1, paragraph 1.3.1.1.1.). It was studied the screw with diameter D = 2.0 m, diameter of bushing - 0.24 m at installation angle of the screw φ = 30° to relative radius r = 0.75. In a computer experiment advance ratio of the screw $\lambda = \dfrac{V}{n_s \cdot D}$ (n_s- momentary number of revolutions of screw) was selected in the range of V = 0-22 m/s, n_s = 10-11RPS. Unlike physical experimental installation, in the calculation model of screw was missing nacelle [84] Fig. 2.2. a, b.



a) b)

Fig.2.2. Models of three-bladed screw: a) Geometrical computer model of screw;
b) Computational computer model of the screw in a wind tunnel.

To improve the accuracy of numerical calculations, the computational domain, inside which is located the screw, should be large enough to inside the domain the jet behind the screw cuold be freely distributed. At the boundaries of the domain should the fulfilled following conditions: conditions of adhesion of the flow on the walls of the wind tunnel or "flow velocity at infinity” [69,78] (Chapter 1 paragraph 1.4.2.) for screw in a flow or in a immovable medium when the screw working "on –the- saite". It should be noted that the domain is "transparent" for air flow and is computational area inside which is constructed the finite element mesh. Configuration of the domain is determined by the shape of the model and the air flow.

The shape of the domain is usually not very critical, and can be a circle, semi-circle, rectangle, sphere, or box. [78].

In Fig. 2.2.b for screw D = 2.0 m is shown an example of a domain in a wind tunnel in the form of a cylinder with a diameter 6.1m and a height 8m. To improve the accuracy of calculation in a jet behind the screw inside the main domain is located cylindrical internal domain with diameter 2.05m. Size of computational mesh in the internal domain is significantly less than in the external. This makes it possible to significantly reduce the total number of computational cells, the time of computation and the required operational memory of computer (Chapter 1 paragraph 1.4.1.).

At the entrance and exit from the domain are set boundary conditions respectively for the velocity and pressure (Fig.2.2.b). On side walls of domain is not set any boundary condition, therefore the program automatically assigns the adhesion conditions of viscous liquid (Vx = Vy = Vz = 0) (Chapter 1 paragraph .1.4.2.).

Screw and bushing of the screw are modelled as solid material (aluminium). Everything else inside the domain is poorly compressed air.

In the case of rotation of the screw with a constant angular velocity it is possible significantly reduce the computation time of the task by placing the screw in the so-called Rotating Region - closed cylindrical region whose axis coincides with the axis of rotation of the screw (Chapter 1. Paragraph 1.4.3.).

Computational CFD program CFdesign except the aerodynamic characteristics and the detailed flow pattern allows to determine the traction force and torque on the shaft of the screw, based on which respectively are calculated the relative coefficients of traction α, power β and efficiency η. (Introduction formulas are calculated relative factors of traction (14) $\overline{\alpha}$, power (15) $\overline{\beta}$, and Chapter 1, efficiency η (1.17)).

As described in Chapter 1, paragraph 1.2. for the numerical solution of the task initial system of nonstationary Navier - Stokes equations with additional equations describing the turbulent transfer, is discretized both in space of the computational domain and in time. As a result, the entire computational domain is covered with computational mesh, which size and numbers of cells are user-defined or automatically defined. For the discretization of differential equations and resolution of obtained algebraic equations in the program CFdesign is used the finite element method. Depending on the type of task for satisfactory accuracy of results in this work were required about 800000-1200000 liquid and solid elements. All tasks were calculated in two CPU computers with RAM ≥ 3.5 GB.

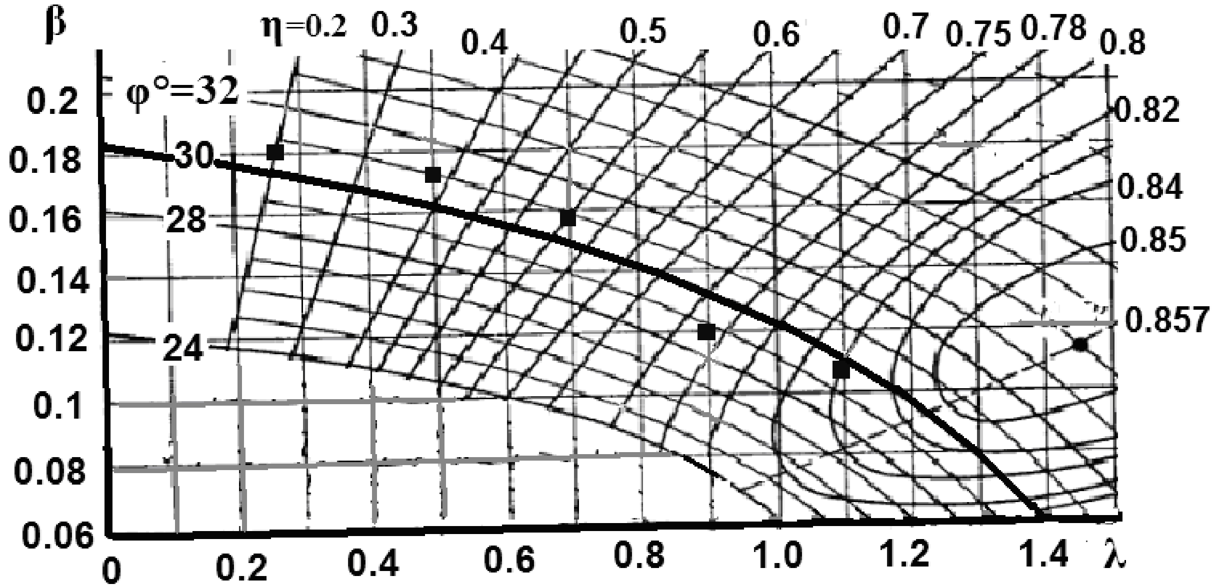


Fig.2.3. Dependences of β (λ) and η (λ) at installation angle φ = 30° for three-bladed screw (points- calculation, line - experiment).

In Fig. 2.3. are presented results of numerical calculation of dependence of power factor β ($\beta=75N/\rho n_s^3 D^5$) from relative advance ratio λ at the installation angle φ = 30°. There are also presented natural experimental data [17, 18, 20, 21] for β and η. From the presented graphs is clear that in case not taking in to account in the calculation the influence of nacelle and wing compartment, computational results are satisfactory matching with the experimental data with an accuracy of ~.5-10%. The obtained results confirm the possibility to use SolidWorks and CFdesign programs for numerical calculation of aerodynamic characteristics of aircraft screws with known geometry.

2.3. Method and results of computer simulation of aerodynamic characteristics of the refurbishment screw.

In practice often arise cases when there are no available data about the aerodynamic characteristics of aerodynamic screw, as well as the detailed geometric characteristics of the blade. A similar situation occurs in case of need to restore the damaged screw or to create a approximate analogue of known screw. For physically existing blade modern technologies [76, 77, 84] allows fast enough by 3D scanning methods to obtain required number of blade section profiles comfortable to create a three-dimensional computer model of the screw.

Method of creation of solid geometrical model of known geometry blade of the screw in SolidWorks based on known geometry of section was considered in Chapter 1. p. 1.3.1.1.3**.** The developed computer model of screw can be used in aerodynamic calculations of a computer model of aerodynamic stand.

In this chapter is discussed the process of simulation of aerodynamic properties of the screw by means of simplified disc model of the fan (aerodynamic screw) [49, 51, 55, 84, 85].

At the beginning were investigated features of formation of jets behind the screw with inlet device and a short cylindrical exit nozzle located in the immovable air. Calculation model of this intermediate task is shown in Fig. 2.4. Model of engine, screw, inlet device, cylindrical exit nozzle, modelled as solids (aluminium) are placed in rectangular domain (domain dimensions 15000x6000x6000 mm). Inside the domain is defined generally poorly compressed air, but on the side walls and at the inlet to and exit from the domain - atmospheric pressure P = 101325Pa. At the bottom of the domain (ground) is not set any boundary condition, therefore the program automatically assigns the adhesion conditions of viscous liquid (Vx = Vy = Vz = 0). Screw with the diameter 1.8m and installation angle $\alpha=16^0$ is placed in rotating region which rotates along with the screw with speed ω = 2180 RPM. Angle of installation and angular velocity of rotation of the screw were chosen approximately the same as the corresponding values in natural experiment, described below.

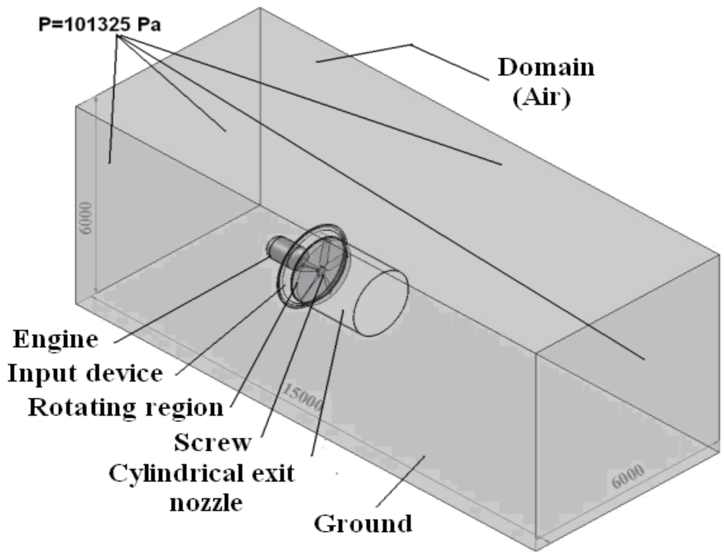


Fig. 2.4. Screw in a short channel with inlet device and cylindrical exit nozzle.

Aim of this task is to determine the flow rate of air flowing through the plane of refurbished screw, as well as to evaluate the estimate the swirl of flow behind the screw. These data will allow approximately substitute the screw in the calculation model of experimental installation with simplified model of internal fan (disk model of fan) which exists in program CFdesign.

Disk model looks like air-permeable rotating disc, which creates a flow with the specified flow rate and can rotate with a constant speed which generally is equal to zero or not exceeding the screw rotation speed. Parameters of simplified internal fan model depend on the accuracy of the restored geometry, installation angle of the blade, screw speed and accuracy of calculation of aerodynamic characteristics of a screw.

As a result, the task of calculation of aerodynamic parameters of the experimental device with the model of internal fan will contain mainly the slow in time and large-scale in space processes, so the time of calculation would be significantly reduced [49, 51, 55, 78, 84, 85].

Below are shown the results of calculation of axial and linear circular components of velocity behind the screw needed for approximate determination of parameters of model of internal fan: volumetric air flow Q m^3/s and number of revolutions of flow behind the screw n_b RPM.

High-quality picture of currents in the area behind the screw (horizontal plane of section) and twist of flow near the plane of rotation of the screw (vertical plane) is shown in Fig. 2.5.

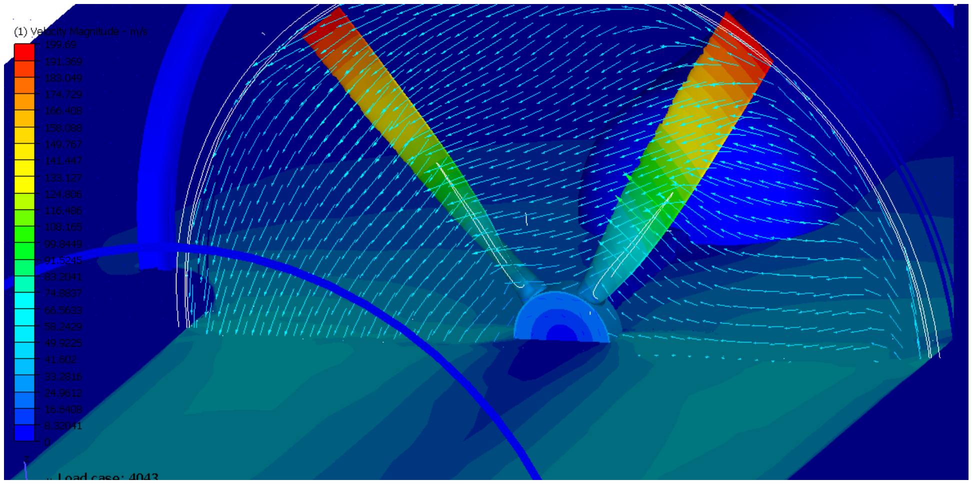
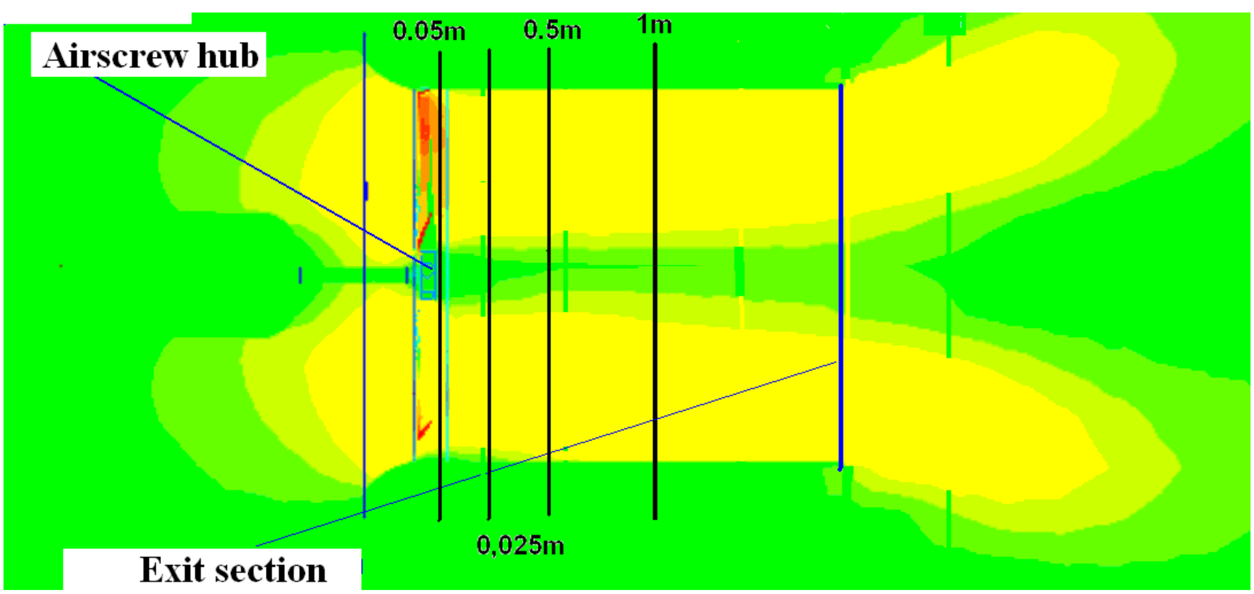
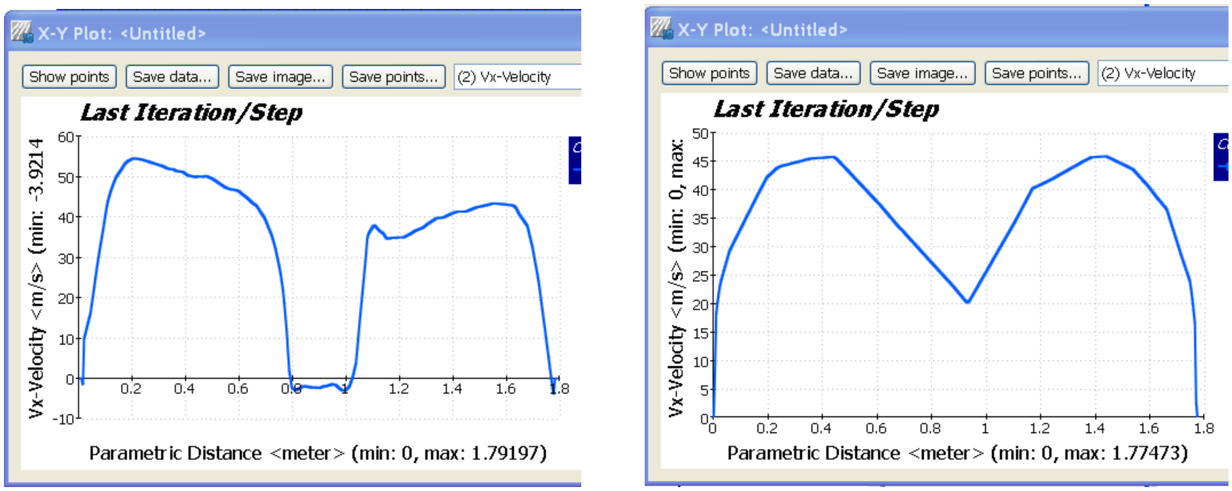


Fig. 2.5. The flow pattern in the area behind the screw (in horizontal plane of section) and twist the flow near the plane of rotation of the screw (in vertical plane).

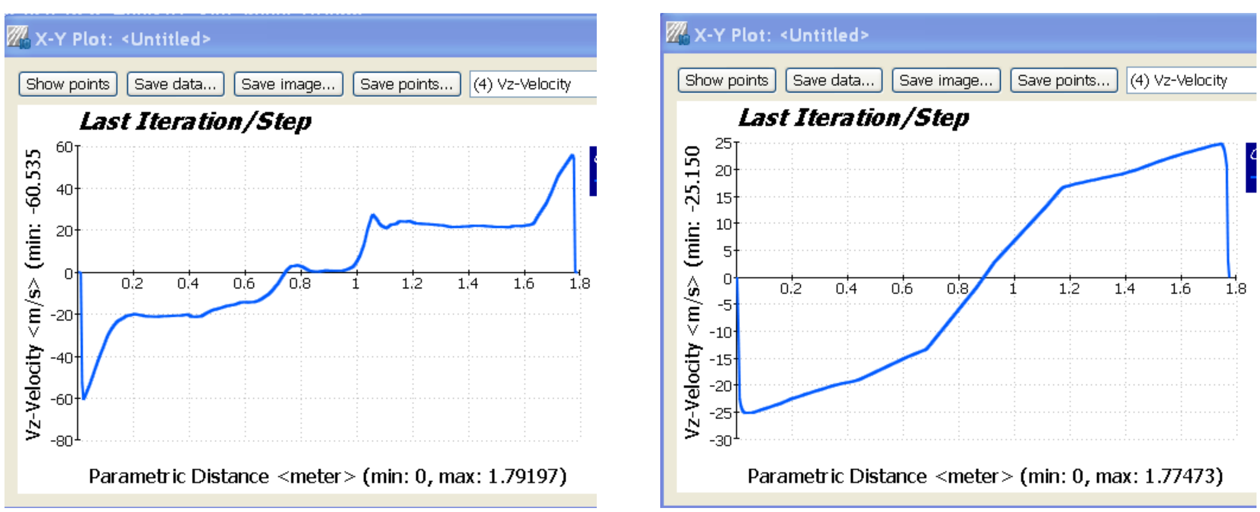
Obviously that behind the cylindrical bushing of the screw appears a zone of reverse flows, which dimensions downstream rapidly reduces. Already at distances of x > 0.125 D from the bushing profile of longitudinal velocity Vx has a "failure" of velocity only near the screw axis, indicating an absence of return currents in this section the flow (Figure 2.6.).





a) b)

Fig.2.6. Dependence of the longitudinal velocity Vx (y) in the section of the jet:
a) - x = 0.05m ≈ 0.025D, b)- x = 1m ≈ 0.5D.



a) b)

Fig. 2.7. Nature of changes of linear rotation velocity of the flow along the radius of the screw.
a) - x = 0.05m ≈ 0.025D, b)- x = 1m ≈ 0.5D.

In general, the flow behind the screw has a variable swirl, which nature of changes along the radius of the blade is shown in Figure 2.7. It should be noted that close to the axis of the screw at a distance 0<r=<0.2R tangential flow velocity V_z increases along the radius of the blade approximately according to the law of solid body, then it is approximately constant 0.2R < r

<0.7R and at r > 0.7R practically linearly increases at approaching of the cylindrical wall of the nozzle (fig. 2.7a). At the distance from the rotation plane of the screw more than x > 0.5D character of rotation of flow is close to the rotation of a solid body (Fig. 2.7.b).

Taking into account of the given data and to reduce the time of numerical calculation of gas-dynamic parameters of experimental installation (fig. 2.8.b, paragraph 2.4.) with a cross-section of exit 2.03m2, in order to obtain the specified average air outlet velocity Vy ≈ 60 m/s, five-blade screw with diameter D = 1.8m can be substituted with approximation model of internal fan with diameter D = 1.8m with volumetric flow rate Q = 120 m3/s. This model is a disk whose diameter is equal to the diameter of the screw and which generates the flow with the specified volumetric flow rate (or distribution of the vertical and radial velocity), and with swirl of flow in plane of model of fan.

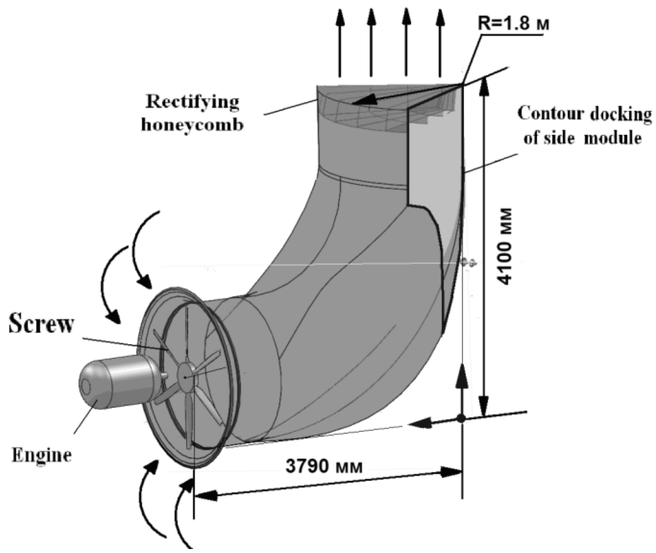
Analysis of results of the systematic calculations showed that the swirl of the flow has little effect on the average velocity in the exit section. Therefore, to compare the results of numerical calculations and experiments was selected fixed number of screw revolutions (n_b = 900 rpm), which corresponds to sliding of the screw s ≈ 0.36 - 0.4 for the range of numbers of screw revolutions of the experimental device n=2180- 450 rpm (s - ratio of the angular velocity of the flow to the angular velocity of the screw).

High value of sliding s was chosen with a purpose of confirmation of its weak impact on the average main parameters of investigated installation. It should be noted that this condition does not occur in case if close to the screw for elimination of twist is installed rectifier, on whose blades within noncomputational regime might appear flow separation that will reduce flow velocity.

2.4. The method of computer simulation of system: inlet device – screw - gas-dynamic channel. Comparison with the experiment

The following chapter discusses the method of computer simulation of symmetric element of real aerodynamic stand. Aerodynamic installation is an open wind tunnel for generation of a large diameter free vertical air jet [87, 88]. Tunnel consists of five identical adjacent to each other modules, in each of which as fan is used above described five-blade screw (a more detailed description of this installation will be discussed in Chapter 3). General view of such a module with the dimensions is shown in the Fig. 2.8.a.

In front of the screw is located inlet device, followed by complex form air channel, turning the flow by 90°. Electric motor installed in front of an inlet device rotates the screw at 2180 RPM. Vertical velocity of the jet coming out from the installation is 50-60 m/s.



b)

a)

Fig.2.8. module of wind tunnel with a fan and motor, a) - computer model, b) –natural tunnel

Feature of numerical solution of task about creation of the air jet in the considered installation consists in the following. Close to the rapidly rotating five - blade screw computational time step must be equal to one-hundredth fractions of the rotation period of the screw (i.e. thousandth fractions of seconds), and the characteristic size of finite elements mesh– around one millimetre.

The process of formation of flow in the air channel and in a free jet is much more slowly and more large-scaled in space, therefore, the desired time step - tenths of a second or more, and the size of finite elements mesh - decimetres and meters.

Taking into account the mentioned characteristics, the average time of calculation of task about the formation of jet behind the screw, motion of flow in the channel and the escape of the free vertical jet might be very large and reach up to several days (with RAM of computer ≥ 3.5 Gb). It should be noted that in comparison with the integral characteristics of screw – flowrate and pressure behind the screw, detailed picture and features of the flow near the screw, has a weak influence on the parameters of the free jet.

Purpose of this phase of the work is to compare the results of numerical calculation of gas-dynamic parameters of experimental installation, with the results of the measurements of distribution of vertical velocity of escape in outlet section of the installation. The module contains five-blade screw with refurbished geometrical and aerodynamic parameters of screw. Measurements were performed using the measuring stick with a length 2 m, on which with the gap 0,2m were installed 8 sensors of air pressure [1,3,4]. The rod was mounted along the radius in the symmetry plane of the sector of outlet section (Fig.2.9.).

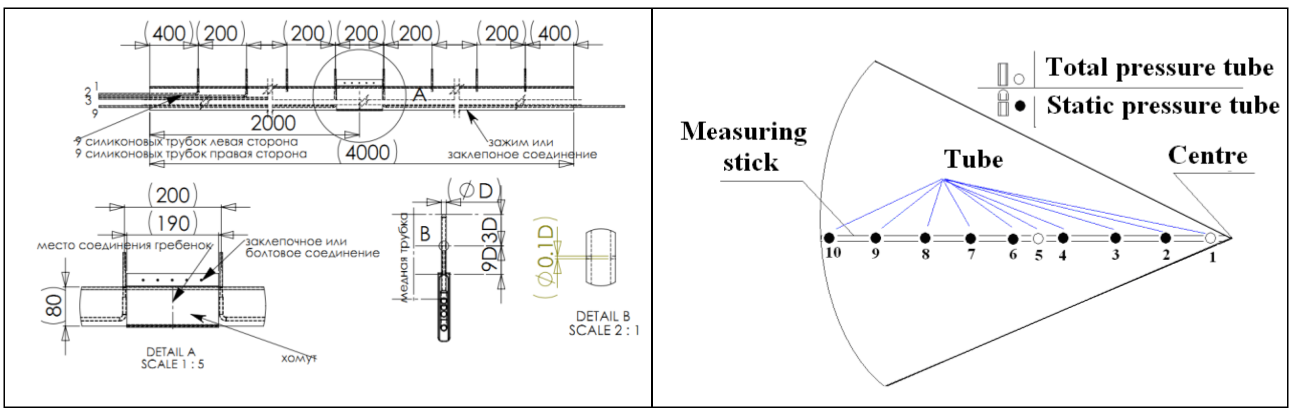


Fig. 2.9. Measuring stick

In this part of work in order to reduce computation time instead of the screw, was considered the model of internal fan. Use of the fan is justified in the previous paragraph. In this case, this model is a disc with a diameter equal to the diameter of the screw and generating a flow with the specified volumetric flow rate (or the distribution of the vertical and radial velocities) and swirling of the flow in the plane of the model of fan. The nature of the changes of vertical flow velocity Vy and linear velocity of the rotation Vz (swirling) along the radius of screw (twist of the flow) for analyzed task is shown in Fig. 2.10.

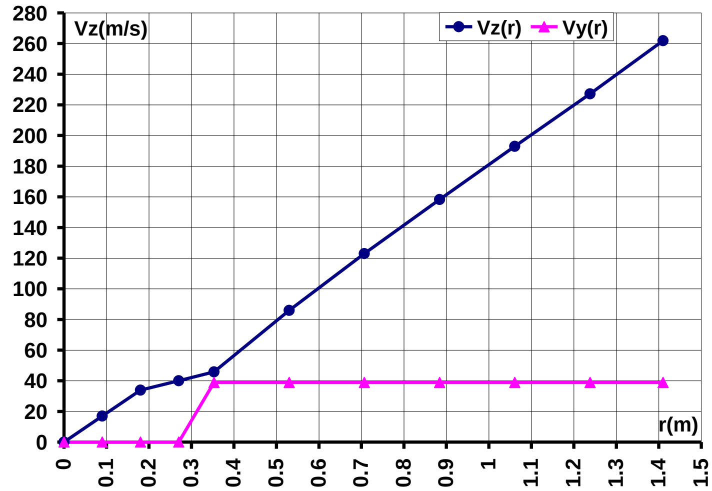


Fig.2.10. Character of the changes of vertical velocity Vy (-▲-) and the linear rotation velocity Vz (-•-) of the flow along the radius of the screw

Results of calculations and measurements are presented in Fig. 2.11., what confirms that calculated data satisfactory coincide with the results of the natural experiment. Note that the picture of flow and distribution of velocity Vy(r) generally in installation and, in particular, in the outlet section is not symmetric (see Fig. 2.11.a). Therefore average vertical velocity in the plane of symmetry of the outlet section in Fig. 2.12. is equal to 47 MPS. At the same time, the average vertical velocity in the whole outlet section is equal to ≈ 60 m/s, what is confirmed by results of the calculation in Cfdesign program, taking into account uneven distribution of V_y (r) in the section (see Fig. 2.11.b).

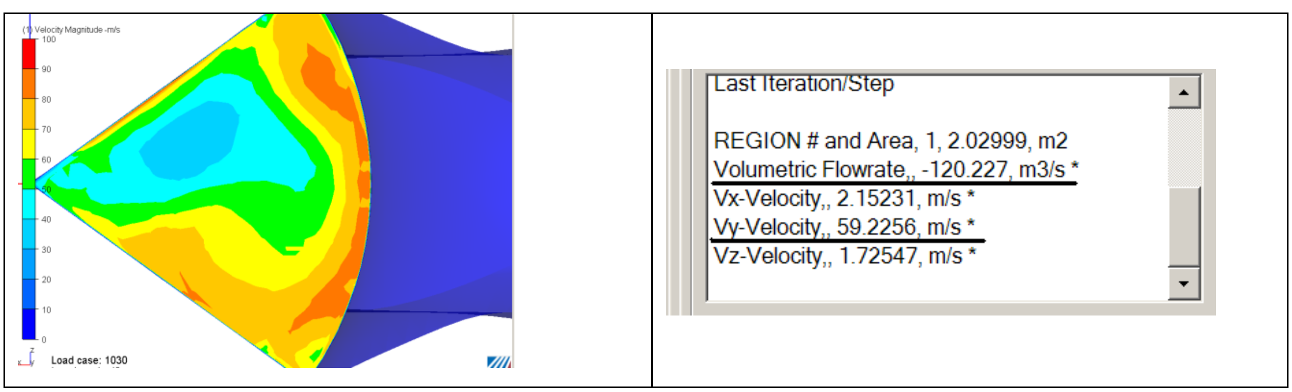


Fig.2.11. a) Picture of distribution of velocity Vy (r) in the outlet section of computer model of the experimental device; b) The results of calculation in the program CFdesign of the parameters averaged in cross-section

Results of the comparison of numerical calculations of the vertical velocity V_Y (line) with the disk model, a five-blade screw with diameter 1.8m and experimental measurements (points) at the exit of the element of investigated natural installation ("Aerodium" type) with a curvilinear channel without honeycomb are shown in Fig. 2.12. Obviously, that calculated data satisfactorily coincide with the results of a natural experiment. This confirms the possibility of using of a developed method of calculation and numerical simulations, and selected approximate disk model of screw as well.

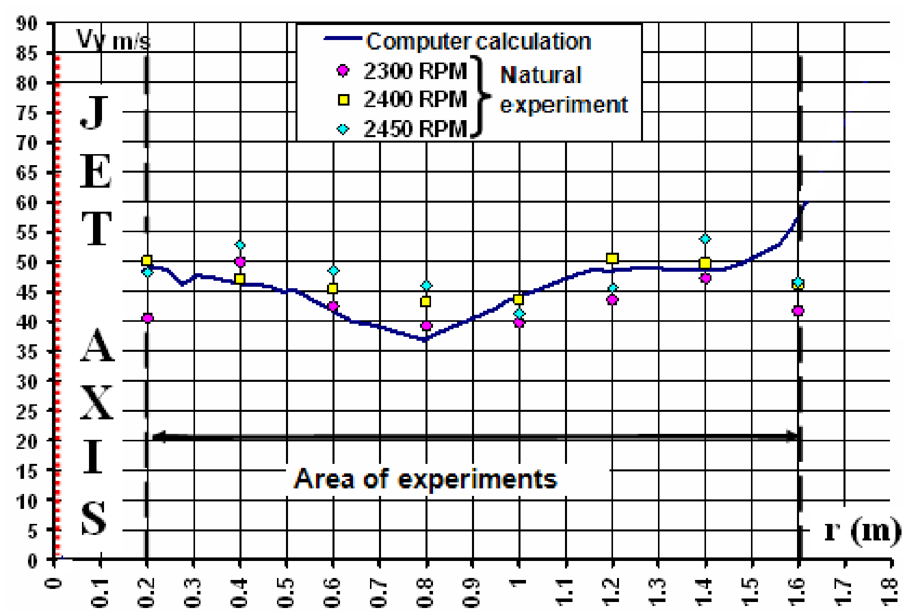
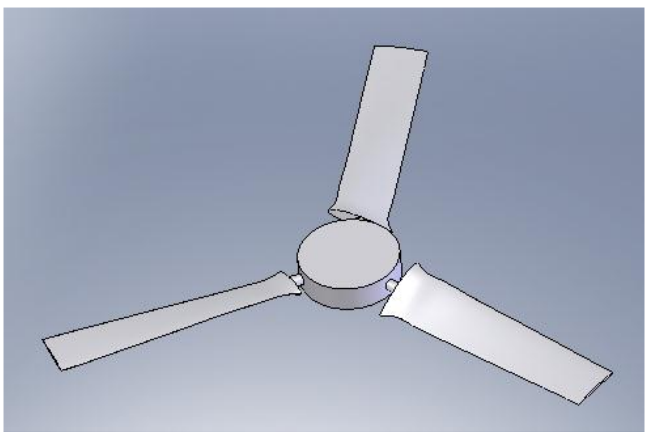


Fig. 2.12. Comparison of results of numerical calculations and experiments of determination of vertical velocity Vy (r) in jet in the exit of the module of the experimental device.

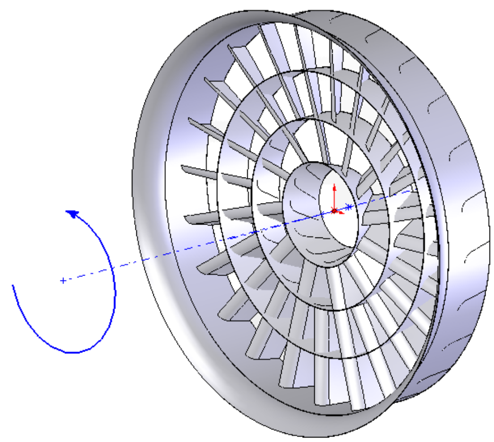
2.5. Computer simulation of aerodynamic parameters of rectifying apparatus with profiled blades.

Purpose of this task: - to develop the method of the computer simulation of the aerodynamic parameters of the submerged vertical air jet, created by system "rotating propeller in the ring – directing vanes with the profiled blades". Method must allow for the possibility of changing of the rotation the frequency of propeller, taking in acount of the influence of the geometry of propeller, on the form of the blades of directing vanes, and to the influence of the geometry of other elements of device, limiting walls and ground on the parameters and the level of non-uniformity of the air flow in the working zone of jet as well. Aim of method is the optimization of the designing process of the system " propeller - directing vanes", the guarantee to defined velocity, twist and the uniformity of flow in the working area of jet [85] .

Design model of the three-bladed propeller, with a diameter of 4 m with the rectangular form of blade is shown on Fig. 2.13.a. The form of the blade sections profiles and the law of variation of the twist of sections were calculated by the theory of N.E. Zhukovskiy [9]. For an improvement of the working conditions of propeller near the ground and to prevent the contraction of the jet it was installed into the profiled ring with the directing vanes (Fig. 2.13.b).



a)



b)

Fig. 2.13. a) Three - dimensional geometric model of three-bladed propellers with the rectangular blades. b) Calculation model of the directing vanes with 24 blades (D=4m).

Directing vanes (counter vanes swirler or honeycomb made from the sheet material) was installed on the exit of the ring coaxially with the propeller. The radial profiled blades of counter vanes swirler have geometric twist (geometric chords of sections are turned relative to the chord of root section, see. Fig. 2.13.b). Sectorial honeycomb is executed without the blade twist in two versions: with the radial profiled blades (Fig.2.14 a, b) and by the radial unprofiled blades (rectangular plates). Honeycomb is installed under the fixed angle of attack for decreasing of the losses on the inlet edge of blade (Fig. 2.14 c). Both types of directing vanes are intended for eliminating the twist of jet, destruction of large vortexes and decrease of the nonuniformity of the profile of the longitudinal velocities near the axis in the working area of the jet. Calculation and layout of the profiled blades of directing vanes were made according to method of B.N. Yurev [10], based on the vortex theory and the considering special features of the velocity distribution in the sections of propeller blade.

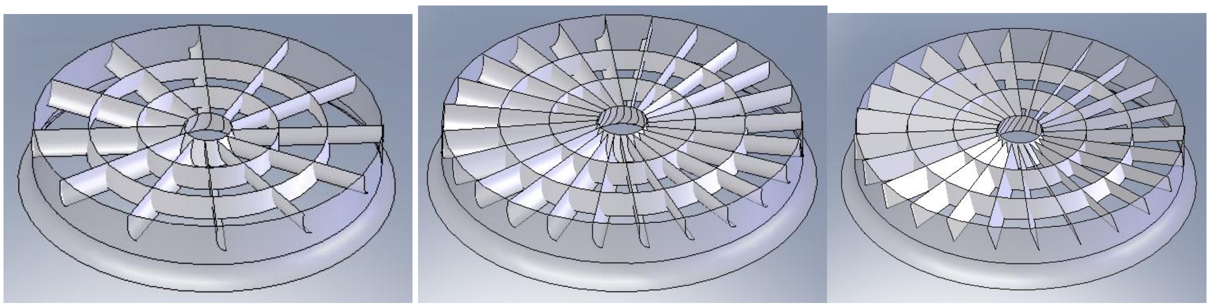


Fig.2.14. Modifications of the directing vanes: a) honeycomb with 12 profiled blades; b) honeycomb with 24 profiled blades; c) honeycomb with 24 rectangular blades without the twist.

Because of the constancy of the blade curvature the residual twist of flow and the nonuniformity of the profile of axial jet velocity in sectorial honeycomb is litle more than curvature of counter vanes swirler, what is disadvantage. At the same time the blades of sectorial honeycomb are simpler both in the production and in the ring assembling process in comparison with the blades of counter vanes swirler.

For creating of the computational mesh of finite elements the electronic geometric model of aerodynamic stand (scale 1:1) was placed into the cylindrical computational region (domain).

On all walls of external domain it is necessary to assign the appropriate boundary conditions. Adhesion conditions on the solid walls are assigned automatically. Because of the presence of the revolving propeller, all tasks are calculated as nonstationary.

For the calculating model with the full-scale geometric parameters and the cylindrical domain being investigated with the size of -8x8.5 m is required required ~ (10^5 - 10^6) calculated mesh nodes and initial time step of the order of $4*10^{-4}$s. In the case of three-bladed propeller with the number of revolutions n_m=1200 rpm one revolution of the model of propeller occurs in 0,05s. The blades are fixed during the one time step Δt, and then they are turned to the angle of $\Delta\theta = 6n_m\Delta t$. (1.15.), (Chapter 1. p.1.4.3.).

The results of calculations for the propeller, with a diameter of D = 4m are given below. Radius of the propeller hub 0.255m, the chord of the rectangular blade b_m= 0.3m, the number of revolutions of the screw n_c=1200rpm = 20rps, the diameter of the ring D_k= 4,1m, height of the ring 0,9m. Atmospheric conditions for the work of propeller - normal: $\rho_н$ =1,225kg/m^2, $P_н$ = 101325 N/m^2.

Twist significantly influance the level of non uniformity of longitudinal velocity in the workng zone of the jet. It is shown that behind the propeller with the infinite number of blades which are installed inside the profiled ring without directing vanes, near the axis of jet is formed the zone with the intensive reverse flow, caused by the significant twist of flow and by the formation of rarefaction zone.

Calculations confirmed the utility of installation of directing vanes in the form of counter vanes swirler or sectorial honeycomb behind the propeller, which it makes it possible to preclude not only the twist of jet, but also to significantly decrease the level of non-uniformity of the flow.

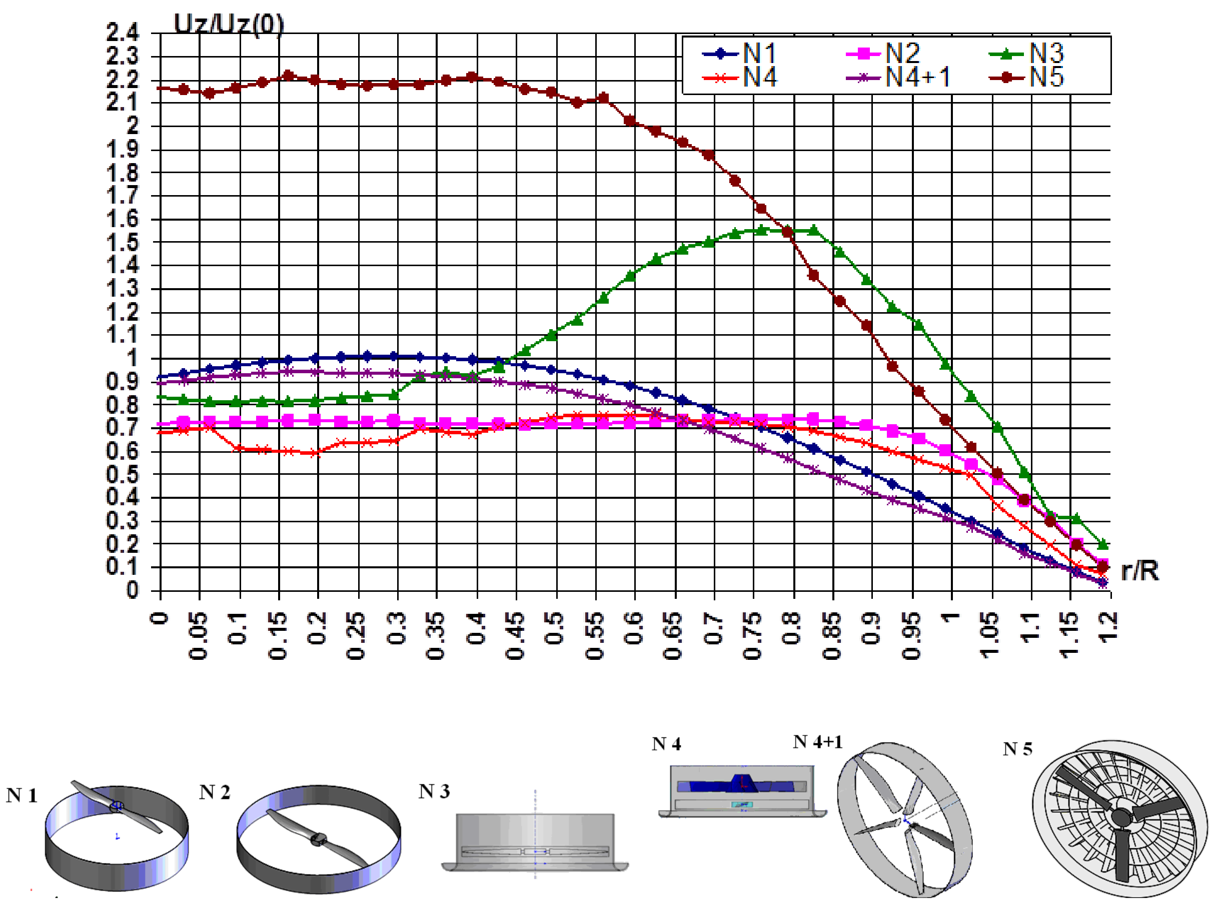


Fig.2.15. Plot of relative longitudinal speed in cross- section section of a jet z =lm for variants of system the propeller - ring: Nr.l - basic version: propeller before the cylindrical ring without the counter vanes swirler. Nr.2 - propeller inside the ring without the counter vanes swirler. Nr.3 -

propeller inside the profiled ring without the counter vanes swirler. Nr.4 - propeller before the cylindrical ring with the counter vanes swirler. Nr.5 - propeller, inside the profiled ring with the counter vanes swirler.

For an increase of velocity in the working part of the jet of the evaluated type of Aerodium of most efficient is the system “propeller, inside the profiled ring with the counter vanes swirler” [12], but not “the propeller, before the cylindrical ring with the counter vanes swirler”, which was previously realized on existing aerodinamic stand.(Fig.2.15., curves 1-5).

The nature of the influence of the directing vanes construction on the flow velocity in the working part at the identical propeller’s working regime is possible to estimate on the basis of the comparisons of the relative longitudinal velocity profile in the fixed transverse section of jet (Fig.2.15.) It is evident that the best parameters have a counter vanes swirler with 24 profiled blades with the twist. Its main advantages:

It is shown that the influence of engine block with the streamlined cover and the ground surface do not worsen, but in certain cases even improve the conditions for incoming flow into the ring (Fig.2.16.). In the case if the electromotor of propeller drive is also installed near the inlet into the ring, the radius of raised edge r > 0.08Dk ensures the practical absence of separation zones. For the defined relative height of ring (H_k/D_k ~ 0.22... 0.25.) it is useful to locate propeller, inside the ring before the cylindrical part at a distance approx. 0.1 D_k from the front edge of the ring.

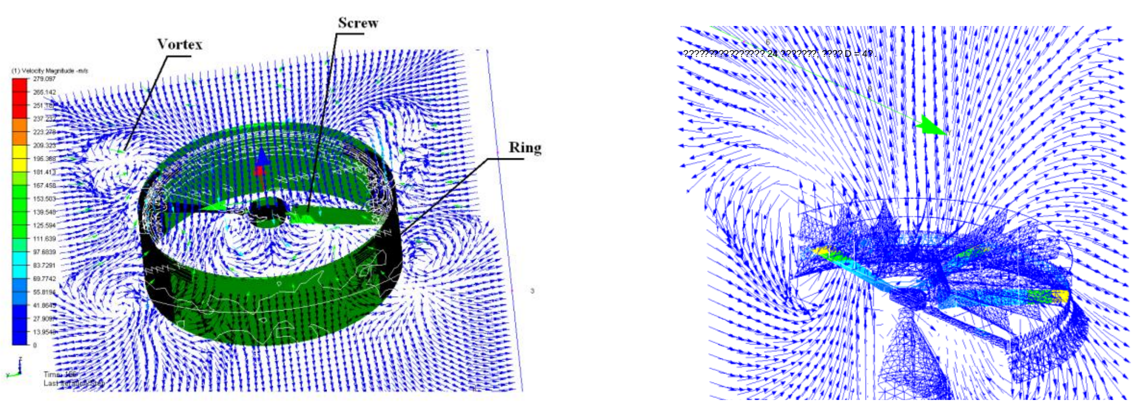


Fig.2.16. Improvement of the conditions for incoming flow into the ring and to the propeller at the installation of cone on the engine and consideration of ground effect.

- Counter vanes swirler successfully liquidates the twist of the flow already at a distance of approximately 0,7 to l,0 m from the outlet section of ring,

- Satisfactorily smoothens velocity drop near the axis of jet at a distance approximately 1,0 m from the outlet section of ring, what makes it possible to install in this cross-section area of jet the shielding - trampoline.

In comparison with counter vanes swirler sectorial honeycomb less efficiently removes the twist of flow and smoothens velocity drop at the identical distances from the outlet section of ring (Fig.2.15.).

The developed method was used for the development of the design of aerodynamic stand with the submerged free vertical air jet, which was prepared and installed in the town Sigulda (Latvia) [86].

CHAPTER 3.

COMPUTER SIMULATION AND OPTIMIZATION OF PARAMETERS OF AERODYNAMIC STANDS FOR FREE FLIGHT OF HUMAN [87, 88, 89, 90,91]

3.1. The purpose of this chapter and the formulation of the task

Aim of this part is development of computer simulation and numerical calculation methods of aerodynamic parameters of three-dimensional computer models of different modifications of installations with purpose of their optimization (decrease of dimensions and required power keeping requirements for air flow quality with defined velocity in working area of jet).

Main tasks of this chapter:

- Development of method of computer simulation and numerical calculation of aerodynamic parameters of three-dimensional computer models of installations with generator of air jet with the purpose of their optimization.
- Optimization of one of designs of aerodynamic stand for generation of large diameter vertical air jet.
- Development of computer simulation and analysis method of geometric and aerodynamic parameters of complex aerodynamic installations, which generally contain system of fans (or aerodynamic screws), gas-dynamic channels and rectifiers for formation of large diameter free vertical jet with defined parameters.

3.2. Universal generator of vertical air jet

As it was mentioned in the introduction, disadvantages of known systems of creation of vertical air jet in open type wind tunnels are following:

Irrational construction of elements and increased operational expenses due to high flow energy losses in separate elements and in the system as a whole at the junction (combination) of elements into common complex without taking into account their mutual aerodynamic effect on each other.

3.2.1. Features of construction of air jet generator

In this part of work instead of known design of wind tunnel for generation a vertical air jet, generally containing inlet channel, aerodynamic screw with engine, outlet channel and rectifier, is investigated air jet generator by methods of computer simulation .

The computer simulation of open type aerodinamic installations by means of CAD/CAE programs allows to create different modifications of three-dimensional geometric models and to carry out the virtual blowing of air jet generators, thus obtaining the optimal geometric and aerodynamic parameters of developed tunnels with the low material expenses. Indicated generator is considered as a single system, whose all elements have non-linear mutual influence on each other and on final result a vertical submerged air jet [88, 89, 91].

Elements of system are compactly placed (one of the main requirements) inside a short vertical axisymmetric gas-dynamic channel of special form in whose cylindrical part is coaxially installed aerodynamic screw. Behind the screw at a short distance from it is immovably fixed device that simultaneously straightens and aligns flow, forming initial part of free jet created by generator.

Construction of the gas-dynamic channel with elements for generation and shaping of air jet consists of the following elements [88, 89]. (Fig. 3.1.):

- Inlet device with special curvilinear shape;
- Straight cylindrical channel coaxially connected to the inlet device, at whose entrance is coaxially installed aerodynamic screw;
- Rectifying and aligning device, which is immovable and coaxially installed in the mentioned cylindrical channel, located behind the screw at a small distance from it, and does not exit beyond the outlet section of the cylindrical channel. Mentioned device consists of concentric rings of truncated cones and radial curved vanes generally with varying geometric twist along radius and curvature of sections of profiles adjacent to the leading edges of profiles.

The generator of vertical air jet works as follows (Fig.3.1 a,b). Rotating screw 1 creates in front of itself the zone of the reduced pressure and forms airflow 2, which moves in the inlet device 3 of gas-dynamic channel 4 upwards and reaches the screw 1.

Inlet device 3 with curvilinear generating line of Bernoulli's lemniscate forms a smooth axisymmetric flow without formation of local zones of flow separation and with low energy loss factor in front of screw. Short gasdynamic channel 4 with a special form (average height of the channel is approximately half of screw diameter) is designed to reduce energy losses in the generator and to increase its efficiency by creating the flow in which there are no sharp turns, areas of flow separation and stagnation zones with parasitic makrovortexes.

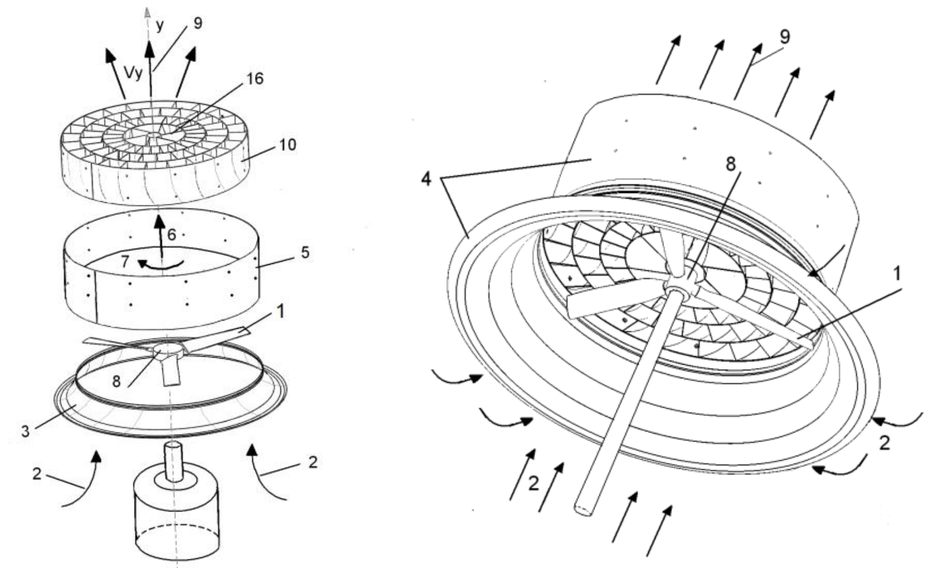
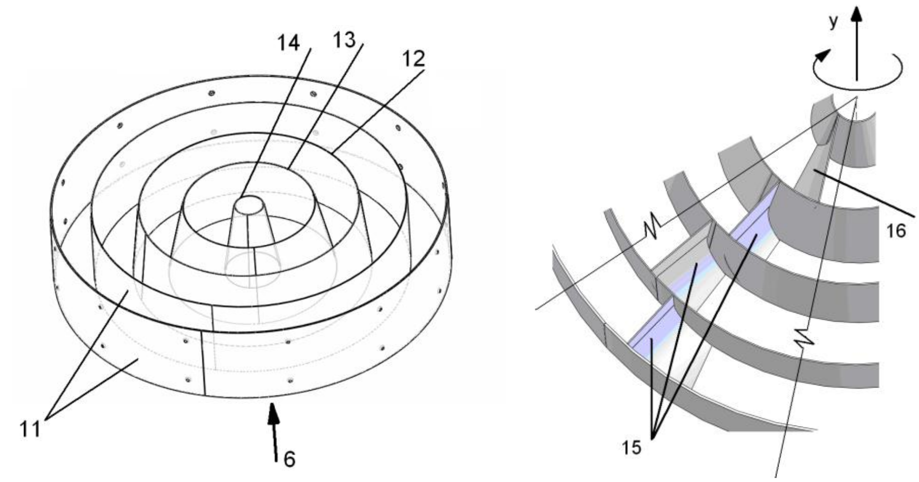


Fig. 3.1. a) Schematic view of the vertical air jet generator (exploded view); b) General view of the vertical air jet generator.

Aerodynamic screw is placed in the adjacent to the inlet device 3 cylindrical part 5 of the gas-dynamic channel 4. Depending on restrictions on the height of the gas-dynamic channel 4, aerodynamic screw must be installed just behind the inlet device in the initial section of a cylindrical channel or at certain distance from it not exceeding the half of diameter of the screw. To increase velocity of flow generated by screw, the clearance between ends of screw blades and the internal surface of cylindrical part 5 must be not more than (2.0-2.5) % of the screw diameter.

Influenced by the screw is formed flow 6, which moves upwards in a cylindrical channel 5 with a great vertical velocity and certain angular velocity of rotation 7 (flow swirling) respectively the vertical axis "Y" of gas-dynamic channel. Vertical velocity profile in the cross section of the flow 6 usually is substantially uneven due to flow swirling by the screw, as well as due to formation of separation zone behind the screw hub 8 and appearance of velocity "failure" zone near the axis of channel. To create in the outlet of the gas-dynamic channel air flow 9 without swirling and with sufficiently uniform vertical velocity profile, behind rotating screw at a small distance from it coaxially is installed device 10 which rectifies and smoothens the flow 6.



a) b)

Fig. 3.2. a) Alternative arrangements of cylindrical rings and truncated cones of rectifying apparatus; b) Variant of layout of one line of vanes in the device, which rectifies and smoothens the flow behind aerodynamic screw.

Device 10 is made of sheet material and as shown in Fig. 3.2a, b, consists of several concentric rings of equal height 11, truncated cones 12-14 and radial curved vanes 15 with a constant or variable geometric twist along the radius. To reduce energy losses when flow attacks the vane, in places adjacent to the leading edge, profiles of vanes are made curvilinear while keeping linear remaining sections of profiles. Curvature of vane profile near the leading edge depends on direction and magnitude of velocity vector of flow coming down from the trailing edge of corresponding section of the screw. Vanes 15 eliminate swirling 7 of air flow 6 behind the screw 1 and partially smoothes zone of "failure" of vertical velocity near the axis of jet. They are installed between the rings 11 and cones 12, 13 along segments of radius shifted by a certain angle relative to each other so that in a rectifying device they are generally located in staggered order. To reduce excessive blockage of free space, in the central area of rectifying device number for vanes is less than at the periphery. Such an arrangement of vanes facilitates assembly of rectifying device and facilitates smoothens velocity profile in jet 9 of generator, inside which does not penetrate makrovortexes formatting behind the screw since they are crushed into smaller vortices in channels formed by vanes and rings or cones.

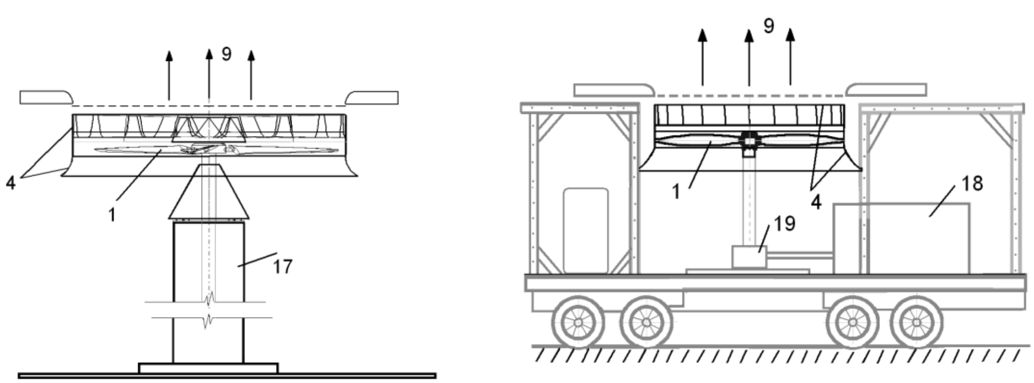
External cylindrical ring 11 Fig. 3.2.a of rectifying device 10 is immovable and coaxially fixed inside a cylindrical channel 5 Fig. 3.1.a, not exiting outside of gas-dynamic channel. Overall height of rectifying and smoothing device is selected depending on diameter of screw hub and restrictions on height of gas-dynamic channel 4.

To fix zone of flow separation behind the screw hub 8 and reduce the "failure" of velocity close to the axis of the jet created by the generator, in the center of rectifying and smoothing device 10 Fig. 3.1.a, in front of screw hub 8 are coaxially installed two downstream tapered thin truncated cones 13, 14, between the walls of which are fixed several short vanes 16 with a geometric twist for swirling of flow between the cones in the direction opposite to rotation of screw Fig. 3.2.a, b.

Diameters of larger base cones 14 and 13 should be respectively slightly larger than diameter of hub 8 of screw 1 and the diameter of the non working zone of vanes, adjacent to the hub. Maximum opening angle of the cones of device 10 is selected from condition of absence of separation of flow inside the formed circular cone channels.

Rotation of aerodynamic screw of air jet generator is realized by means of electric motor 17 or internal combustion engine 18, that are connected to the screw shaft directly or through a reducer 19. In both cases engine and gearbox are mounted coaxially in front of or at the inlet of gas-dynamic channel in area of low-velocity flow Fig. 3.3.a, b.

Proposed construction of air jet generator is universal and designed to be used as a separate module in stationary (Fig. 3.3 a), sectional and mobile (Fig. 3.3.b) open type installations for human flight in a vertical air flow.



a) b)

Fig. 3.3. a) Stationary aerodynamic installation with vertical air jet generator.

b) Mobile aerodynamic type installation with vertical air jet generator.

3.3. Features of calculation method and analysis of influence of geometry of rectifying device on output parameters of jet of generator.

Results of this chapter are generalization of the numerical calculations of various models of open type vertical air jet generators. The main purpose of calculations was to create a model with minimum height at a fixed diameter of jet at the outlet from generator and maintaining quality requirements of air flow with specified velocity in working section of the jet. Generator of air jet must be universal enough to be able to be used as a separate unit in stationary, sectional and mobile open type aerodynamic stands for human flight in a vertical air flow [88].

In Fig. 3.4. are shown some of the 23 variants of geometric models of rectifying devices for vertical air jet generators.

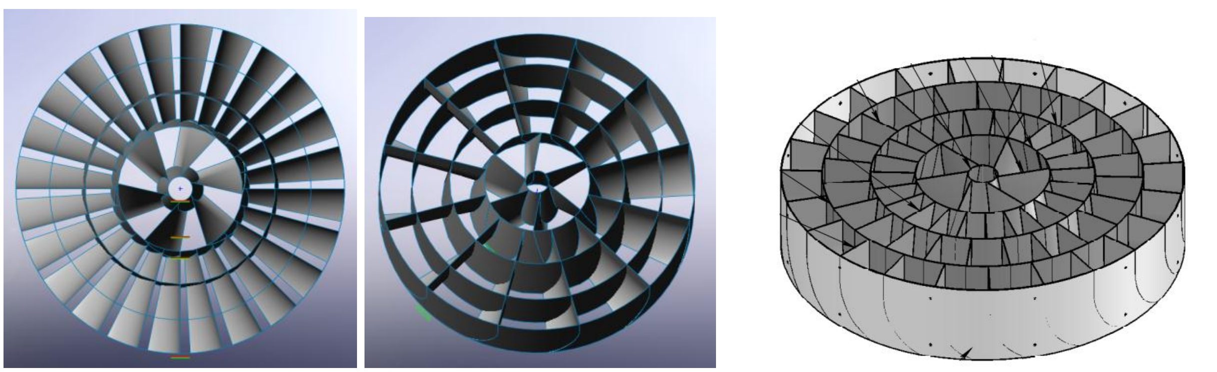


Fig. 3.4. Variants of models of rectifying devices.

a) Rectifying devices of 24 vanes with cylindrical curvature and with geometric twist along radius; b) Rectifying devices of 12 vanes with cylindrical curvature and with geometric twist along radius; c) Rectifying devices of 24 vanes with cylindrical curvature and with geometric twist along radius and located in staggered order.

Radial curvilinear vane of proposed rectifying apparatus of air jet generator has variable along the radius geometric twist and curvature of the initial sections of profiles adjacent to the front edges of the blades. In Fig. 3.5. is shown one of the simplified versions of manufacturing of vanes with cylindrical curvature and without geometric twist along radius.

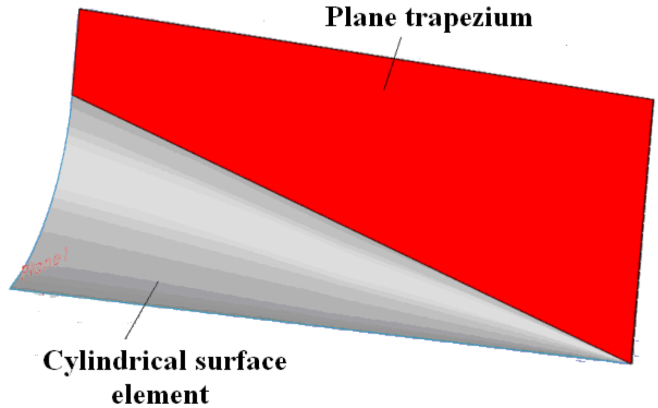


Fig. 3.5. Vane of rectifying apparatus of air jet generator

To create a universal generator were required more than 20 calculations whose aim was optimization of individual elements of generator. During computational process were changed the number and angles of installation of vanes of rectifying apparatus, number and disclosure angles of cones, as well as height of the rectifying apparatus.

Configuration of domain for numerical calculations and generation of finite element mesh, determined by the form of investigated model and the air jet is shown in Fig. 3.6.

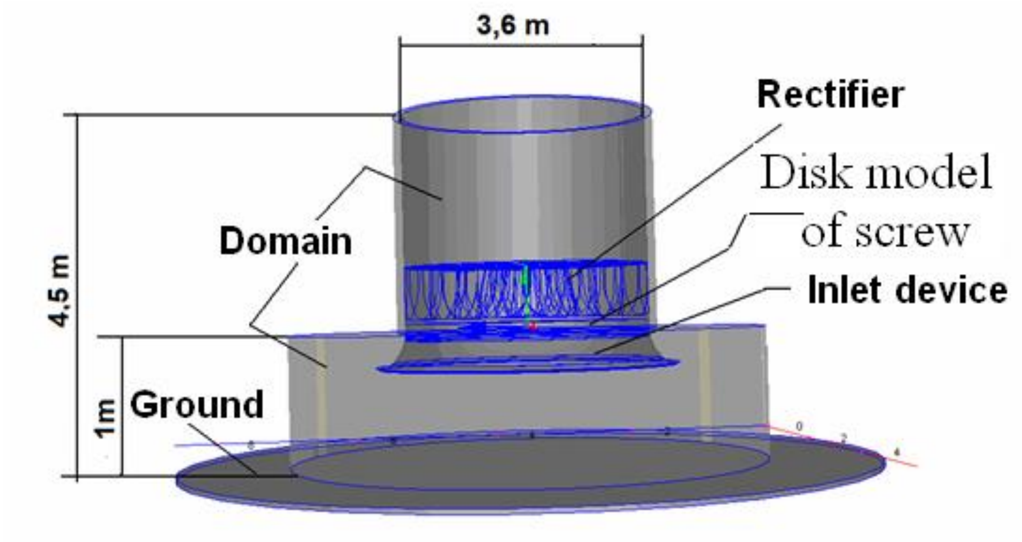


Figure 3.6. Calculation scheme of air jet generator with domain

On all solid walls of model, including ground surface, program automatically sets the boundary conditions of adhesion (air velocity is zero.) On all exterior walls of the domain was set atmospheric pressure.

In order to reduce computation time in the computer blowing instead of the screw was considered disk model of screw (model of internal fan, see Chapter 2. p.2.3.). Confirmation of possibility of using a method of calculation and numerical simulations, and selected approximate disk model of screw was justified in Chapter 2.

Qualitative picture of distribution of flow swirling, vertical velocity V_y in vertical plane of section of generator, and velocity profile V_y in the cross section of jet at a distance of 1m from outlet section of generator without rectifying device are shown in Fig.3.7. a.

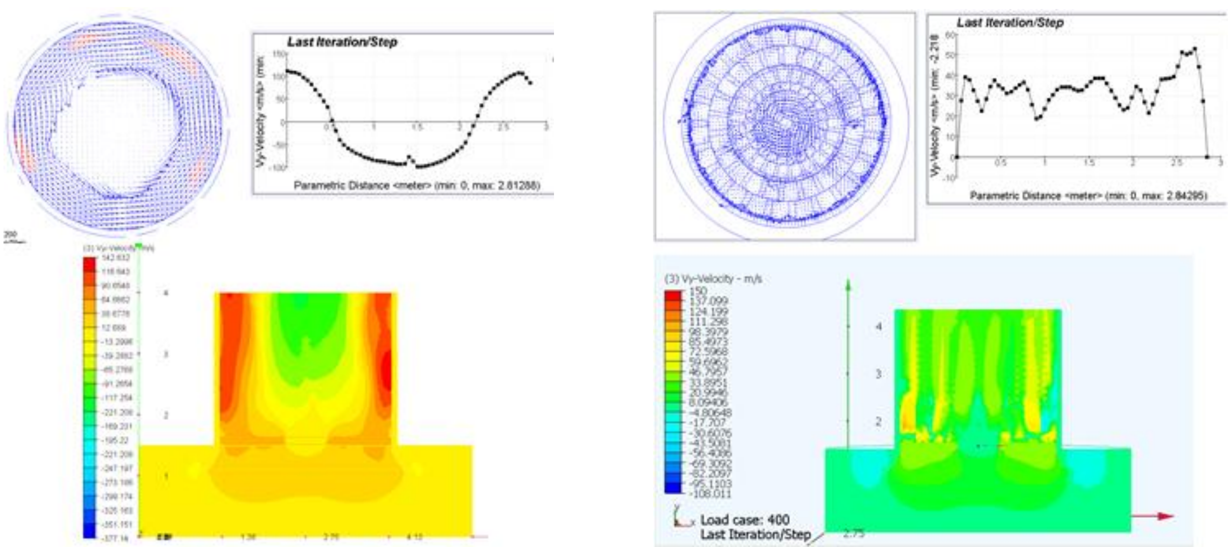


Fig.3.7. a) Distribution pattern of flow swirling and vertical velocity component (V_y) in absence of air flow rectifying device. b) Distribution pattern of flow swirling and vertical velocity component (V_y) of air flow after installation of the rectifying device.

A similar picture of distribution of vertical velocity (V_y) for generator with rectifier is shown in Fig. 3.7.b It is obvious that in case of absence of rectifier in a significant part of the cross-section of the stream is present "failure", that is not acceptable for on ground simulators, simulating long skydiving or sport and entertaining attractions, using free manned flight. Installation of the developed rectifier significantly improves uniformity of flow in cross-section of the stream already at small distances from outlet section of the generator. With the increase of the location height of the cross sections, distribution of vertical velocity becomes more even while reducing the maximum vertical velocity on the axis of the wind tunnel.

Based on a series of performed calculations have been optimized parameters of the vertical air jet generator [88, 89, 91]. Proposed universal construction of vertical air jet generator with can be used as separate module in stationary, sectional and mobile aerodynamic stands for the free manned flight. Installation was designed and manufactured in Latvia Fig.3.8.

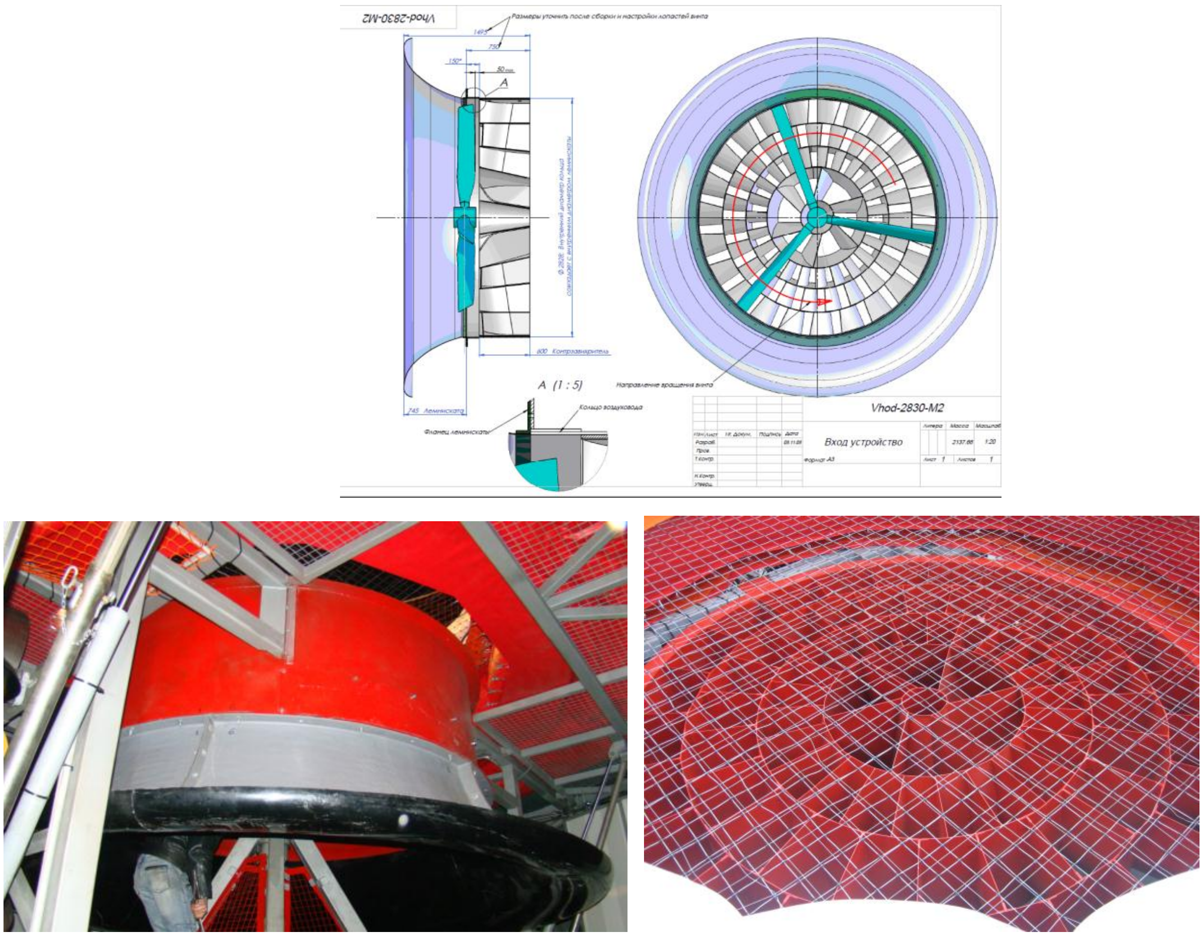


Fig. 3.8. Construction air jet generator.

3.4. Recommendations for engineering of air jet generator.

Based on systematically performed computer calculations have been developed recommendations for engineering of vertical air jet generator. These recommendations are atributable to the fixed (static) elements of aerodynamic stand, whose design can have a significant effect on aerodynamic parameters of work area of air jet.

Main recommendations for engineering of aerodynamic stands are following (see Fig. 3.9.):

➢ Inlet device should be performed with a special curvilinear shape (for example, thin-walled axisymmetric channel with smooth curvilinear generating line of Bernoulli's lemniscate) [22,23, 92, 93];

➢ Inner diameter of cylindrical channel should be no more than (2.0-2.5)% larger than the diameter of screw [10];

➢ Overall height of rectifying and smoothening device should be selected in the range (0.1 ÷ 0.4) of screw diameter depending on screw hub diameter and restrictions on height of gas-dynamic channel;

➢ Rectifying and smoothening device, fixed in a cylindrical channel, should be located on a small distance from screw (about 30-50 mm from structural elements of the screw) [22, 23, 92, 93];

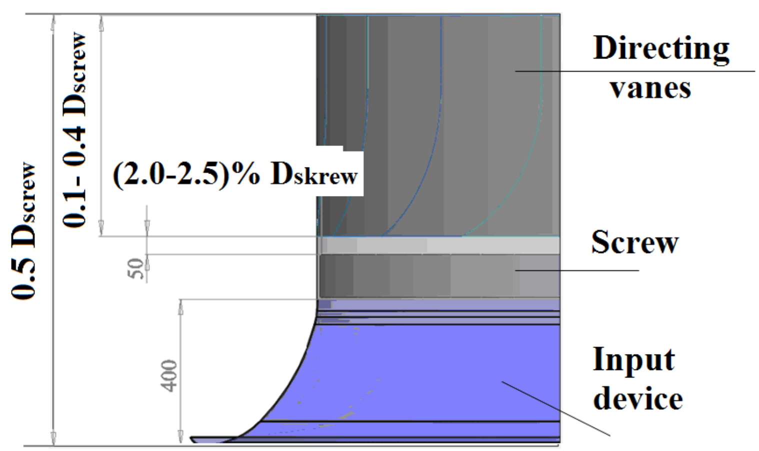


Figure 3.9. Main parameters of air jet generator

➢ Average height of a short gas-dynamic channel of a special form is about half of the screw diameter (D0.5);

➢ Aerodynamic screw should be installed just behind the inlet device in the initial section of a cylindrical channel or at some distance from it no more than half of the screw diameter (D0.5) [22,23];

➢ To reduce velocity "failure" in the central part of the work area of a jet is recommended to install inside the ring downstream tapering channel in the form of a circular truncated cone coaxial with ring. Diameter of main inlet cone, which is located in front of screw hub, must be greater than the diameter of non-working part of blades. According to statistics, diameter of non-working part of the blade is approximately equal to ~ (0.2 ... 0.3) D, where D - diameter of the screw. The opening angle of narrowing part of the cone can be quite large - from 14º to 38º. Cone height does not exceed the height of the cylindrical part of the ring and is equal to width of rectifier vanes adjacent to side wall of cone. If diameter of the ring of aerodynamic stand D_R is equal or less than 1.5m, for a satisfactory alignment of the velocity field is enough to install one small cone [27, 94].

➢ To reduce velocity "failure" behind the ring of large diameter rectifier (D_R ~ 4m), inside the ring should be installed several cones of different diameters. To prevent flow separation inside the tapered annular channels their opening angle must be selected in range of 10º ... 12º. If the opening angle of the annular channel is too large, then one or two cones (excluding cones adjacent to the axis or to the internal wall of the ring) can be replaced by a cylindrical channel.

➢ When installing screw of aerodynamic stand directly on the motor shaft (in this case the engine is located on the ground in front of the screw) is recommended to install on the upper end section of engine with some ring gap one conical dome, inside which is coaxially located shaft of the motor. This conical or generally other axisymmetric streamlined dome is designed to eliminate flow separation behind the end section of motor and improve conditions of formation of flow incident on the screw. Annular gap can improve the ventilation of end section of engine.

➢ Values of the curvature, angles of installation chords of sections of vanes depend on the geometric characteristics and the main operating mode the screw. Width of vanes is approximately equal to the length of the cylindrical part of the ring in which they are installed.

➢ To avoid excessive blockage of passage section of the generator ring near the axis of vanes of rectifier, it is recommended to use two types of blades - long and short, which are instaleld alternately. Long blades are mounted between the ring and the small cone, short - between ring and intermediate cone or cylinder.

➢ Safety grid is advisable to install in that section of the jet, in which velocity "failure" near the axis is small enough. Based on the calculated data it is recommended to install it in a distance approximately (0.35 - 0.5) D_R from outlet section of ring of rectifier.

3.5 Example of computer optimization of parameters of aerodynamic stand type with symmetric system of fans

Initial aerodynamic stand, which was described in Introduction fig.10 consisted of five symmetrically installed screws with diameter 1.8m, located inside short cylindrical channels, whose axes were directed at the angle of ~80° toward vertical axis of jet. Fans create five free collided jets, which according to the intention of designers had to create the required large diameter vertical jet. Trial natural experiments showed that the required parameters of vertical air jet do not realize.

In this part are presented results of computer optimization of geometric and aerodynamic parameters of construction aerodynamic stand, which was developed by one of the private companies and described in Introduction. Behind the screws is installed fixed rectifying device designed to eliminate swirling of the jet in work area Fig.3.10. Advantages and disadvantages of this aerodynamic stand are discussed in Introduction.

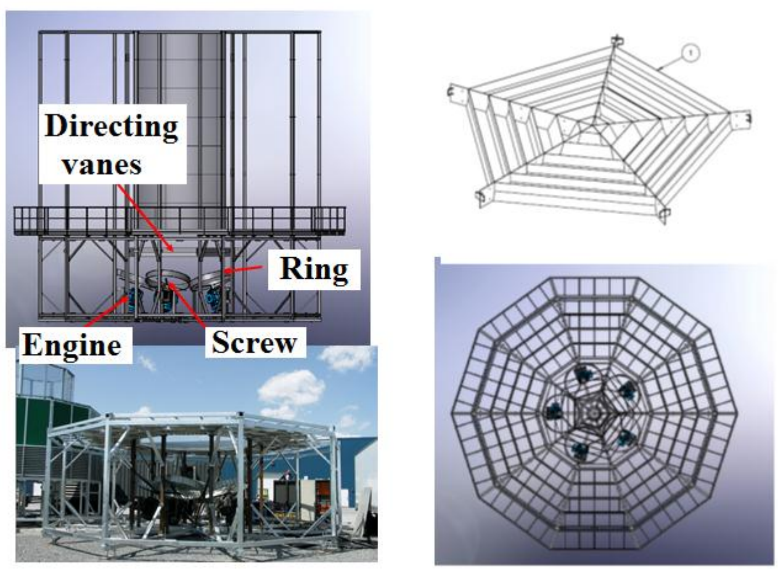


Fig. 3.10. Main constructive elements of design of aerodynamic stand developed by private company.

In the process of computer optimization were examined several versions of constructions (see Fig 3 – 11 a-g) of open type aerodynamic stand. One of the main conditions of optimized design was mandatory obtaining of flow velocity in the initial section of the free jet ~ 60 mps and use of the already existing aerodynamic screws, electric motors and electrical equipment [90].

Analysis of geometry and results of numerical calculations of output aerodynamic parameters showed that best of the developed versions, which are presented in Fig. 3.11, is version g.

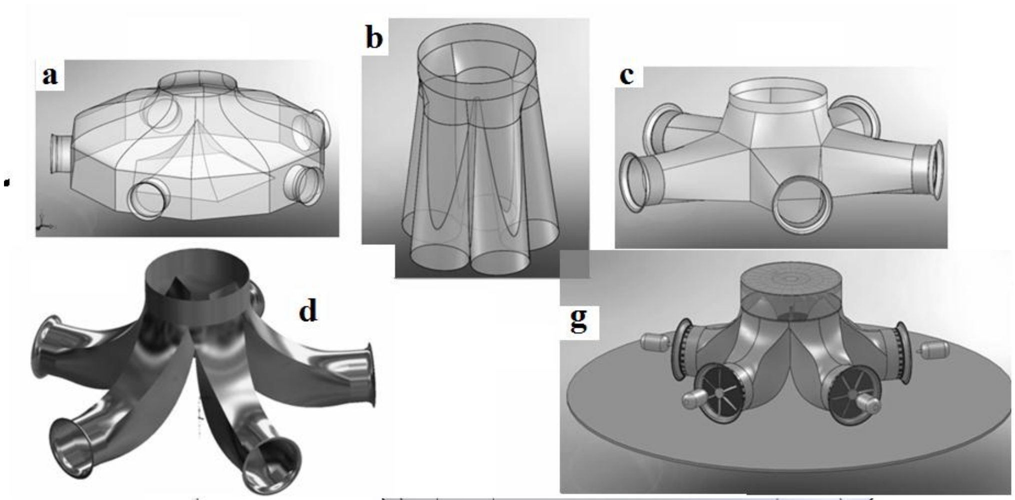


Figure 3.11. Examinated options of stand structures

Computational three-dimensional geometric model of stand and its elements were created in CAD program SolidWorks (Fig. 3.12).

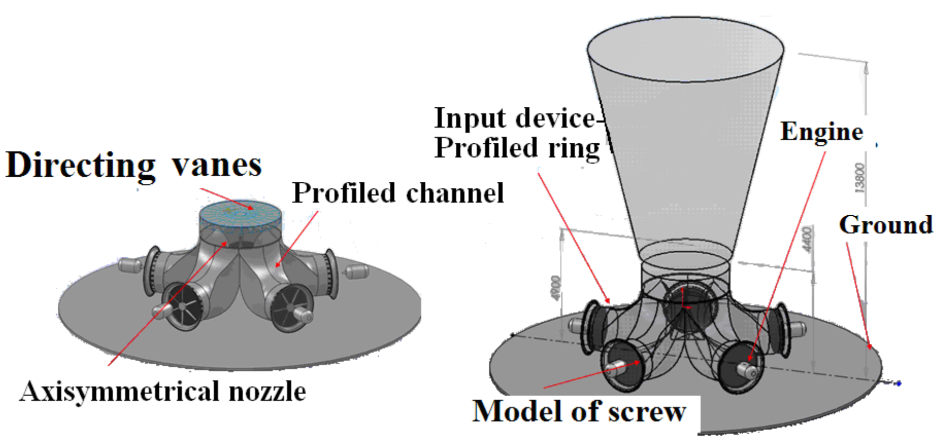


Fig.3.12. Geometrical and computational model of aerodynamic stand.

In the designed and investigated computer model of aerodynamic stand (Fig.3.12) vertical air jet creates five serial five-blade screws with the diameter 1.8m, whose horizontal axes are symmetrically located in the circle along its radii. Each screw is installed into the specially profiled ring, which behind the screw is transfered into the three-dimensional profiled channel of complex Γ - shaped form for the smooth and the nonseparable turning of flow at 90˚ in parallel to the vertical axis of installation. Behind the completion of the turning of all five channels they are junked into one axisymmetrical nozzle with the diameter ~3.6m, which forms free jet.

The finite-element computational model of the optimized full-scale tunnel with vertical jet is shown in Fig.3.13. Results of computer calculations of symmetrical 1/5 part of this installation, and also their comparison with the results of physical experiment are presented in chapter 2, section.

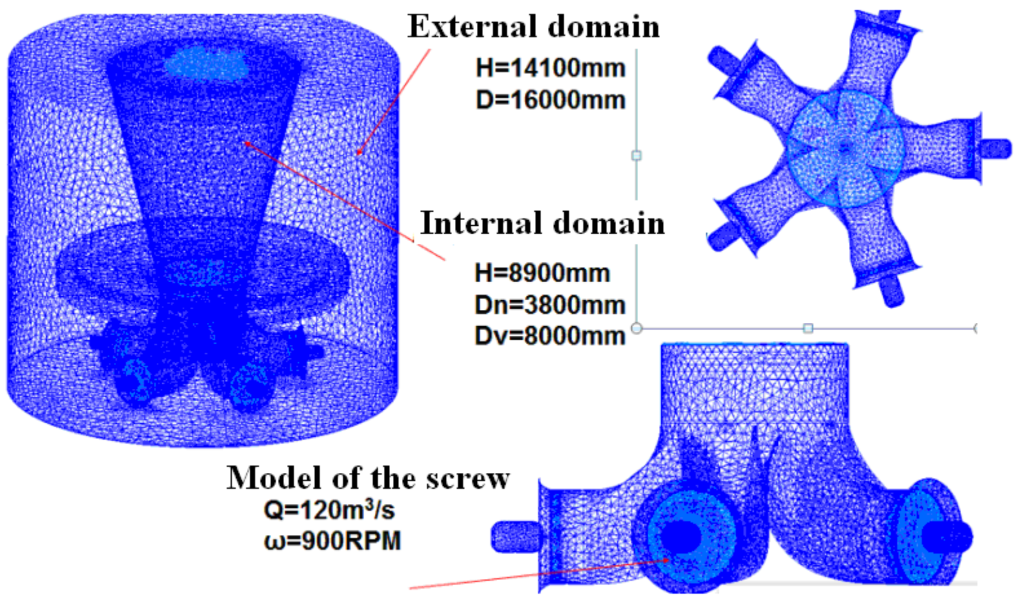


Fig. 3.13. Finite element computational model of optimized tunnel with natural dimensions

In this task instead of screw was considered its disc model, for which was defined number of revolutions, volumetric air flow rate and the level of slippage of blades relative to the air flow 50%, which corresponds to 2100 revolutions of screw per minute.

Models of screws do not have behind them rectifying devices, eliminating flow swirling. As shown by calculations, considered system has the feature that in process of mixing of jets macro-vortices, which are formed after them considerably, destroy each other because of the identical direction of rotation of all screws. After the turn of jets behind screw per 90° at the nozzle exit appears a jet with five zones with relatively weak swirls, whose centers are moved from jet axis and located on its periphery (Fig.3.14).

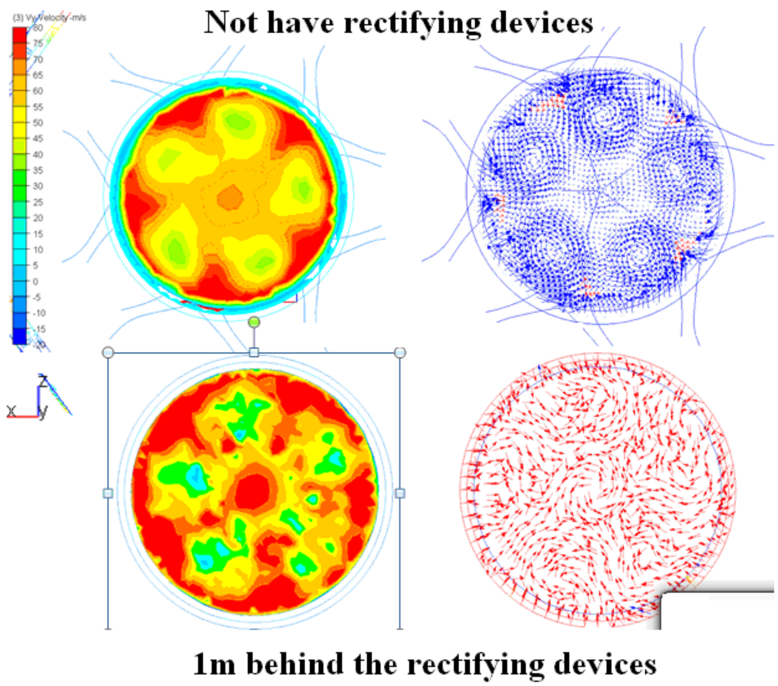


Fig.3.14. Distribution pattern of vertical velocity and swirling zones in the jet.

To eliminate residual zones of flow swirling was sufficient to install in the mixing chamber a simplified rectifier like circular honeycomb with a low coefficient of energy loss (Fig. 3.15). The results of calculations presented in Fig.3.14. confirm this conclusion.

Fig. 3.15. a) Simplified geometric model of rectifying device. b) Full-scale model.

Main advantages of the proposed version of optimized stand:

- Sufficiently low hydraulic losses in gas-dynamic circuit;
- Shaped channels and absence of the mixing chamber made it possible to avoid energy losses due to collision of jets, created by screws, which as a result increased efficiency of tunnel;

- Use of a system screw in the shaped ring, increased flow rate and velocity of flow behind the screw;
- Construction made it possible to obtain the maximum velocity (~60-70 MPS) in the initial cross-section area of vertical jet at the outlet from the gas-dynamic channel with the defined screw geometry and power of electric motor, and also the maximum uniformity of flow in working area of jet;
- Feature of proposed design of aerodynamic stand is absence of special rectifying devices behind screws and presence of circular honeycomb at the outlet of gas-dynamic channel [27, 95, 96].

Should be noted that in the optimized design computer simulation helped to solve complex geometrical problems related to junction of sections of air channels while respecting restrictions on narrowing and expansion angles, smoothness of transitions and turns of channels, junction of multiple channels.

Developed stand was designed, manufactured, assembled and experimentally investigated in Jelgava.

In order to determine optimal installation angle of blades of aerodynamic screws and evaluate required energy consumption of the stand, series of field experiments was performed. The experiments were performed at 1/5 of the installation, discussed in the second chapter, for different numbers of revolutions in range of 2000-2450 RPM and screw blade installation angle of 11° -16°. Using pressure sensors were determined total and static pressure, and then calculated flow velocity. Graph of velocity dependence on radius for different screw installation angles at different revolutions is shown in Fig. 3.16. Numerical data are resulted in the tables 3.1., 3.2., 3.3.. The graph shows that close to the optimal installation angle of screw blade is 16° at a screw revolutions 2180 RPM. These data were used in computer calculations discussed in the second chapter.

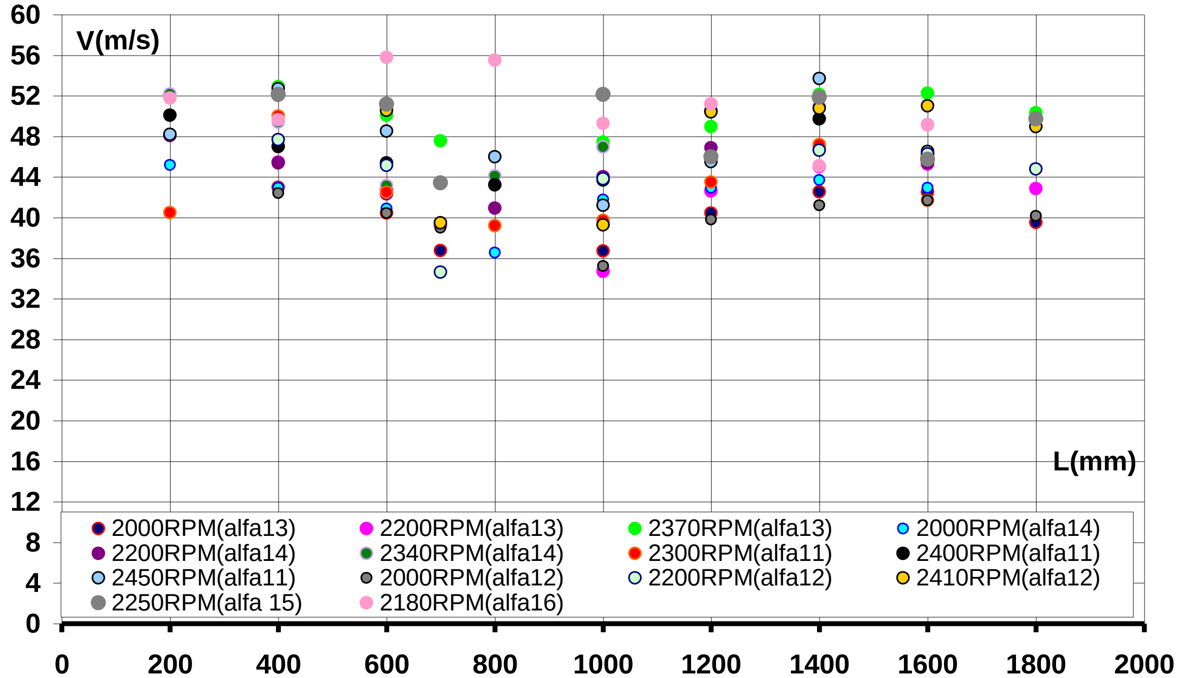


Fig. 3.16. Graph of velocity dependence on radius for different screw installation angles at different revolutions of screw

Tables 3.1.

	RPM2300	RPM2400	RPM2450	RPM2000	RPM2220	RPM2410
L mm	alfa 11	alfa 11	alfa 11	alfa 12	alfa 12	alfa 12
200	40.5	50.1	48.2	42.4	47.6	52.0
400	50	47	52.7	40.4	45.1	50.5
600	42.5	45.4	48.5	38.9	34.6	39.4
800	39.2	43.2	46			
1000	39.7	43.7	41.2	35.2	43.8	39.2
1200	43.5	50.5	45.5	39.8	46.0	50.3
1400	47.2	49.7	53.7	41.2	46.6	50.7
1600	41.7	46.2	46.5	41.6	46.2	51.0
1800				40.1	44.7	48.9
Average	**43.0**	**46.9**	**47.7**	**39.9**	**44.3**	**47.8**

Tables 3.2.

	RPM2000	RPM2200	RPM2370	RPM2000	RPM2200	RPM2340
L mm	alfa 13	alfa 13	alfa 13	alfa 14	alfa 14	alfa 14
200	42.9	45.4	52.8	45.1	48.0	52.1
400	40.4	42.7	50.0	42.9	45.3	49.4
600	36.7	39.2	47.5	40.8	42.3	43.1
1000	36.7	34.7	47.4	36.5	40.9	44.0
1200	40.4	42.6	48.9	41.7	44.0	46.9

1400	42.5	45.0	52.1	42.9	46.8	51.1
1600	42.4	45.2	52.2	43.6	47.0	50.9
1800	39.5	42.8	50.3	42.9	45.3	49.1
Average	**40.2**	**42.2**	**50.1**	**42.1**	**45.0**	**48.3**

Tables 3.3.

	RPM2250	**RPM2180**
L mm	**alfa 15**	**alfa 16**
200	52.1	51.7
400	51.1	49.6
600	43.4	55.7
1000	52.1	55.4
1200	45.9	49.2
1400	51.8	51.2
1600	45.7	45.1
1800	49.7	49.0
Average	**49.0**	**50.9**

Due to the fact that air jet generator operates at different speeds and uses different power, it was necessary to estimate energy consumption. A graph of power consumption dependence on screw revolutions at different screw installation angles is shown in Fig.3.17.

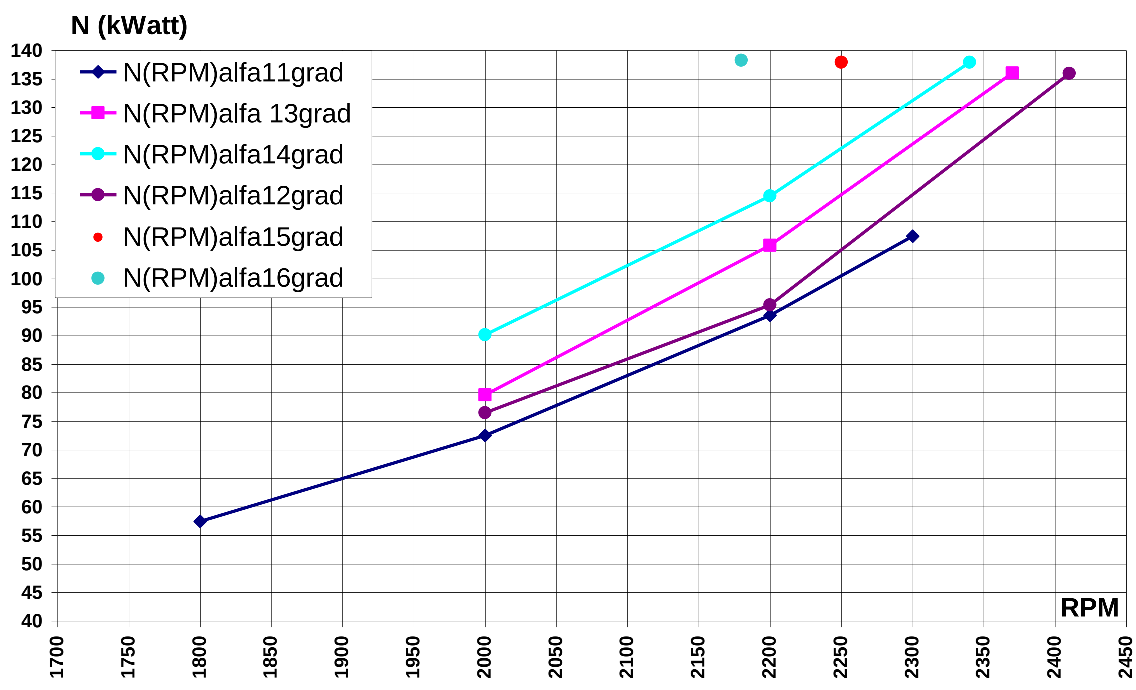


Fig.3.17. Graph of power consumption dependence on screw revolutions at different screw installation angles.

Natural experiment also allowed experimentally to determine influence of designed circular honeycomb on intensity of flow swirling in the outlet section the jet and to compare these data with results of computer simulations.

Fig. 3.18 presents results of experimental determination of vertical flow velocity Vy(r) in the outlet section of jet of experimental installation (without circular honeycomb - blue dots, with circular honeycomb - red dots). It should be noted that the graph of vertical flow velocity Vy(r) without rectifying device is not symmetric relative to the axis of jet. This is explained by the fact that in the experiment measuring rod was placed in the outlet section the jet as shown in Fig. 3.19. The right side of the rod is in one of zones of residual flow swirling with makrovortex that near to its axis created the failure of velocity. At same time the left side of the rod was located between two makrovortexes and there is no failure of velocity.

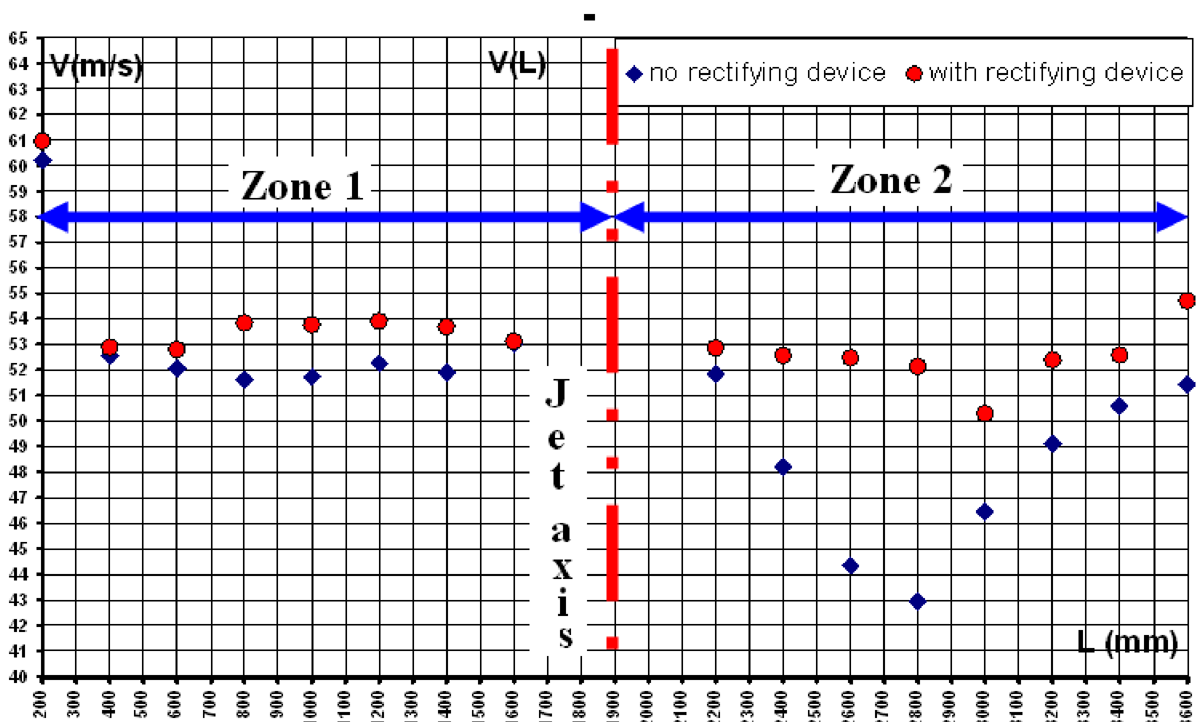


Fig.3.18. Results of experimental determination of vertical flow velocity Vy(r) in the jet in outlet section the experimental installation.

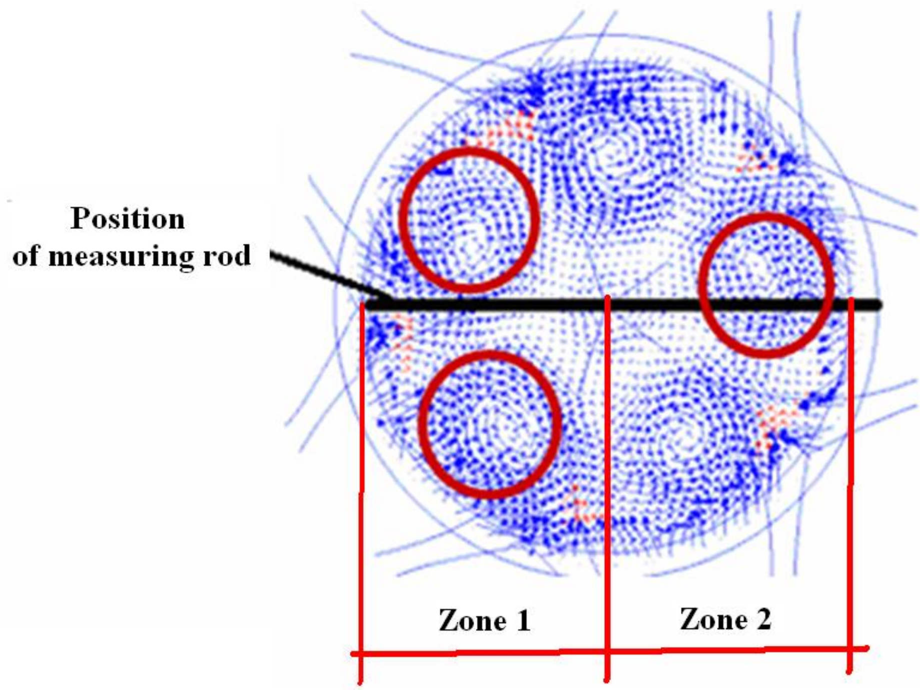


Fig. 3.19. Position of measuring rod

Results of comparison of vertical flow velocity Vy(r) in outlet section of the jet obtained from computer experiment are shown in Fig. 3.20.

From data in the figure follows:

- In a computer experiment Fig.3.20. , as well as in the full–scale test (natural experiment) Fig.3.18. at absence of a circular honeycomb ring is present zone of velocity failure (blue dots - natural experiment, pink dots - computer experiment).

- After installation circular honeycomb in natural experiment Fig.3.18. failure was successfully eliminated (red and blue dots), as confirmed by computer calculations Fig.3.20. (brown dots).

It should be noted received results of comparing computer and full–scale test give only quality resultate.

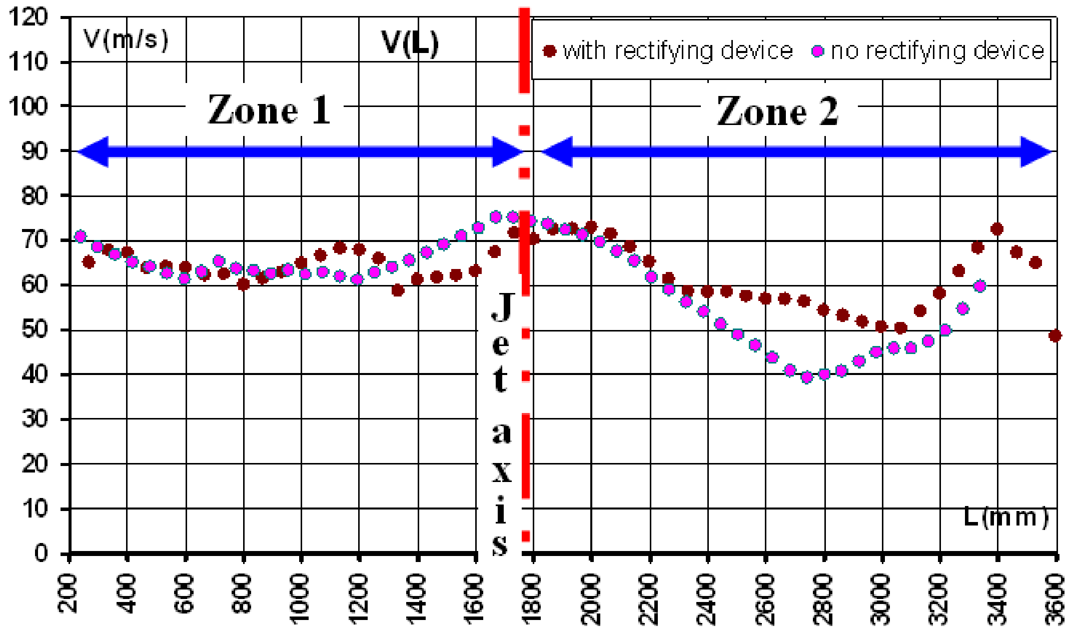


Fig. 3.20. Comparison of results numerical calculations and experimental determination of vertical flow velocity Vy(r) in the jet in outlet section of experimental installation

Thus, physical experiments satisfactory (quality) confirmed results of numerical calculations of developed method for optimized design of stand with five symmetrically located aerodynamic screws.

Fig. 3.21. shows full-scale installation which was designed, manufactured and installed in Jelgava (Latvia), in accordance with optimized computer model.

Fig. 3.21. Full-scale plant in Jelgava (Latvia).

MAIN CONCLUSIONS

1. Based on modern engineering CAD/CAE programs was formulated the method of computer simulation and calculation of the parameters of on ground located aerodynamic stands (or open wind tunnels), who are creating the large diameter vertical air jet of. Indicated stands are intended for the free manned flight in the vertical air flow.

2. Developed methods of computer simulation and numerical calculation of aerodynamic characteristics of refurbished aviation screws, allowing to obtain results of the calculations, which with accuracy of 5 - 10% coincide with known experimental data.

3. For the five-blade screw with the restored geometry, which was installed in the designed and manufactured module of experimental aerodynamic installation, results of executed natural experiments confirmed the satisfactory matching of the calculated and experimental data of the value of the vertical velocity of air jet in the outlet section of module.

4. Developed methods of computer simulation of aerodynamic characteristics submerged vertical air jet generated by system “the rotating aerodynamic screw in a ring (shrouded propeller) – rectifying device with profiled blades”. The given method provides possibility of change of frequency of rotation of the screw, the registration of influence of geometry of the screw on the form of blades of the rectifylng degree of an air flow in a working zone of jet.

5. Based on a series of calculations performed to optimize the geometry of the aerodynamic stands for generation of vertical air jet proposed universal design of air jet generator, which can be used as a separate module in stationary, sectional and mobile open type wind tunnels for the free flight of human in a vertical air jet.

6. Developed the method of computer simulation and analysis of the geometric and aerodynamic parameters of the complex aerodynamic installations, which generally contain the system of aerodynamic screws (or fans), gas-dynamic channels and rectifiers for the formation of large diameter free vertical jet with defined parameters.

7. Based on method (point 6) was designed, calculated and optimized aerodynamic stand with five symmetrically located five-blade screws, which create the submerged large diameter vertical air jet. Stand was manufactured, installed and experimentally tested in Jelgava (Latvia). Physical experiments satisfactorily confirmed the results of numerical calculations performed according to developed procedure.

LITERATURE

INTRODUCTION

1. *Радциг А.Н. Экспериментальная гидроаэромеханика. –Москва: МАИ, 2004 г.-296* **с.**

2. *Гошек И. Аэродинамика больших скоростей- Изд. Иностранной литературы., Москва: 1954 г. -521 с.*

3. *Горлин С.М., Слезингер И.И. Аэромеханические измерения. Методы и приборы. — Москва: Наука, 1964 г. -720 с.*

4. *Краснов Н.Ф., Кошевой В.Н., Данилов А.Н. и др. Прикладная аэродинамика. Изд. «Высшая школа», 1974г. - 732с*

5. *United States Patent No.: US 7,156,744 B2 // Recirculating vertical wind tunnel skydiving simulator.// Skyvcnturc, LLC, Austin, TX (US). Date of Patent:Jan. 2, 2007*

6. *International Patent PCI7US2005/027750 //Recirculating vertical wind tunnel skydiving simulator and reduced drag cablfe for use in wind tunnels and other locations.// Skyventure, llc [US/US]; 7751 Kingspointe Parkway, Suite 126, Orlando, FL 32819 (US). International Publication Date 2 February 2006 (02.02.2006)*

7. *United States Patent US006083110A //Vertical wind tunnel training device//. Sky Venture, Inc., Celebration, Fla. Publication Date Jul.4, 2000.*

8. *United States Patent 4,578,037 //Skydiving simulator// Alexander Macangus, 16 Chapel Street; Thomas Dickson, 4 Burgess Terrace, both of Edinburgh, Scotland. Publication Date Mar. 25, 1986.*

9. *Александров В.Л. Воздушные винты. Учебное пособие для авиационных ВУЗов Государственное издательство оборонной промышленности- Москва 1951.- 475 с.*

10. *Б.Н. Юрьев. Воздушные винты, вертолеты. Избранные труды. Том 1. Изд. Академии наук СССР Москва -1961.- 540 с.*

11. *Мхитарян А.М., Ушаков В.В., Баскакова А.П., Трубенок В.Д. Аэрогидромеханика. Москва: Машиностроение, 1984- 351 с.*

12. *Мхитарян А.М. Аэродинамика. Издание второе, переработанное и дополненное. Москва – Машиностроение, 1976 - 435 с.*

13. *Володко А.М. Основы аэродинамики и динамики полета вертолетов. Москва-Транспорт, 1988.*

14. Володко А.М. Основы летной эксплуатации вертолетов. Аэродинамика. Москва- Транспорт, 1984 - 256 стр.

15. Майкопар Г.И., Лепелкин А.М., Халезов Д.В. Аэродинамический расчет винтов по лопастной теории. Труды ЦАГИ, вып.529, - 1940.

16. Белоцерковский С.М., Васин В.А., Локтев Б.В. К построению нестационарной нелинейной теории воздушного винта. Изв. АН СССР, МЖГ, № 5, 1979.

17. Кравец А.С. Характеристики воздушных винтов. Государственное издательство оборонной промышленности. Москва - 1941г.

18. Кравец А.С. Характеристики авиационных профилей. Москва: Оборонгиз - 1939.

19. Плицкер Д.М., Турьян В.А. Аэромеханика. Государственное научно-техническое Издательство Оборонгиз, 1960- 276 с.

20. Barnes W. Mc Cormick. Aerodinamic, Aeronautics and Flight mechanics. John Wiley & Sons.inc., 1995-652.p.

21. Hartman E.P. and Bierman D. The aerodynamic characteristics of four full- scale propellers having different plane forms. NACA Report Nr. 643, 1938.

22. Курочкин Ф.П. Основы проектирования самолетов с вертикальным взлетом и посадкойю Москва - Машиностроение, 1970 г.

23. Ханжонков В.И. Аэродинамика аппаратов на воздушной подушке. Москва- Машиностроение, 1970 г.

24. Остославский И.В., Матвеев В.Н. О работе винта помещенного в кольце. Труды ЦАГИ Nr.248, 1935г.

25. Шайдаков В.И. Аэродинамические исследования систем «винт в кольце» на режиме висения. Труды МАИ Nr.111. Оборонгиз. 1959 г.

26. Ништа М.И. Аэродинамика летательных аппаратов и гидравлика их систем. Изд. ВВИА им. Н.Е.Жуковского. 1981 - 571с.

27. Идельчик И.Е. Справочник по гидравлическим сопротивлениям. — Москва: Машиностроение, 1992.

28. Горлин С.М., Слезингер И.И. Аэромеханические измерения. Методы и приборы. — Москва: Наука, 1964.

29. Румянцев А. Г., Силантьев В. А. //Расчётно-экспериментальное исследование обтекания механизированных профилей // Теплофизика и аэромеханика. 2010. Т. 17, № 2., 91–104 с.

30. Румянцев А. Г. //Обтекание механизированных профилей: расчёт и эксперимент// Авиация и космонавтика – 2009. VIII Международная конференция, Москва. 2009: Тез. докл. Москва: Изд. МАИ-ПРИНТ, 2009. 18–19 с.

31. Босняков С.М., Власенко В.В., Горбушин А. Р., Глазков С.А., Курсаков И.А., Михайлов С.В., J. Quest. *//Математическая модель Европейской аэродинамической трубы (ETW) и опыт её применения // ТРУДЫ МФТИ. — 2011. — Том 3, № 4*

32. Майкопар Г.И., Лепелкин А.М., Халезов Д.В. //Аэродинамический расчет винтов по лопастной теории.// Труды ЦАГИ, вып.529, 1940.

33. Лепелкин А.М. //Аэродинамический расчет тяжелонагруженного винта при нелинейной зависимости подъемной силы профиля от угла атаки.// Сборник работ по теории воздушных винтов, ЦАГИ, 1958.

34. Майкопар Г.И. //Приложения вихревой теории винта.// Труды ЦАГИ, вып.613, 1947.

35. Белоцерковский С.М., Васин В.А., Локтев Б.В. К построению нестационарной нелинейной теории воздушного винта. - АН СССР, МЖГ, № 5, 1979.

36. Ковалев Е.Д., Удовенко В.А. Метод расчета нестационарных аэродинамических характеристик одновинтового вертолета - Киев: " Технология и организация производства" , № 1, 1992.

37. Келдыш В.В. //Обтекание лопастей винта с отрывом потока.// Инженерный журнал. т.III, вып.1,1963.

38. Хэм Н.Д. //Аэродинамическая нагрузка на профиль в двумерном потоке при динамическом срыве.// Ракетная техника и космонавтика. № 10, 1968.

39. Ковалев Е.Д., Удовенко В.А. //Расчет аэродинамических характеристик воздушных винтов численными методами.// Авиация общего назначения - Харьков: №11, 1999.

40. Дербенев С.Г., Кальясов П.С., Любимов А.К. //Математическое моделирование взаимодействия маршевого винта с элементами аэродинамической компоновки судна на воздушной подушке (свп). Анализ аварий движительного комплекса свп пр. А-32 С.Г.// Вестник Нижегородского университета им. Н.И. Лобачевского, 2007, № 4, с. 92–97.

41. Ветчинкин В.П., Поляхов Н.Н. //Теория и расчет воздушного гребного винта.// Москва: Государственное издательство оборонной промышленности, 1940.

42. Г. Шлихтинг. Теория пограничного слоя. - «Наука». Ред. Физико-математическая Литература. Москва 1969.

43. Internet- http://www.cadreview.ru/files/u2/cadfem2011.pdf

44. Попов С.А., Игнатов Н.Е. //Проект вертикальной аэродинамической трубы ВТ-1.// МАИ. Авиационная техника 2009 г.

45.Wilcox D.C. Turbulence modeling for CFD. - DCW Industries, Inc 1994.

46.Патанкар С. Численные методы решения задач теплообмена и динамики жидкости. — Москва: Энергоатомиздат, 1984.

47. А.В. Печенюк. //Численное моделирование обтекания крыла конечного размаха с аэродинамическим профилем NACA-2406 потоком несжимаемой жидкости при малых числах Маха.// - ООО “Digital Marine Technology”.

48. Система моделирования движения жидкости и газа FlowVision. Руководство пользователя. - Москва, ООО «ТЕСИС», 2005.

49. Лукьянова И.Э., Шмелев В.В. Возможности FlowVision в построении моделей для исследования процессов удаления отложений в нефтяных резервуарах. - САПР и графика 2`2006

CHAPTER 1

50. Кунву Ли. Основы САПР CAD/CAM/CAE – СПб.: Питер- 2004,- 559 стр.

51.А.А. Алямовский и др., SolidWorks. Компьютерное моделирование в инженерной практике. - Санкт-Петербург, 2005, -800 стр.

52. А.Б. Каплун, Е.М. Морозов, М.А. Алферьева “ANSYS в руках инженера” Практическое руководство. - Изд. 2-е, испр. Москва: Едиториал УРСС, 2004,- 269 стр

53. Басов К.А. ANSYS справочник пользователя. - Москва 2001, - 637 стр

54. Titus Petrila, Damian Trif, Basics of fluid mechanics and introduction to computational fluid dynamics. - ©2005 Springer Science + Business Media, Inc., Print ©2005 Springer Science + Business Media, Inc. - Boston. 515 p.

55. Cfdesign technical reference. Upfront CFD Version 9.0. Copyright (C) Blue Ridge Numerics, Inc. 1992-2006

56. Robert W.Fox, Alan T. McDonald, Philip J. Prithard. Introduction to fluid mechanics.- Sixth Edition., John Wiley &Sons, INC. -787 p.

57. Martin O. L. Hansen. Aerodynamics of Wind Turbines. - Second Edition. Copyright © Martin O. L. Hansen, 2008. London • Sterling, VA - 192 p.

58. Лойцанский Л.Г. Механика жидкости и газа.- 7-е издание., испр.- М.Дрофа, 2003.- 840 с.

59. Dirba A., Uiska J., Zars V. Hidraulika un hidrauliskās mašīnas - Riga, Zvaigzne, 1980.g.

60. Черный С.Г., Чирков Д.В., Лапин В.Н., Скороспелов В.А., Шаров С.В. Численное моделирование течений в турбомашинах – Новосибирск: Наука – 2006 – 202с.

61. Cherny S.G., Chirkov D.V., Lapin V.N., Skorospelov V.A., Turuk P.A., Numerical simulation of a turbulent flow in Francis hydroturbine // Russ. J. Numer. Anal. Math. Modeling – 2006 – V. 21, № 5, – P. 425-446.

62. Cherny S. Chirkov D., Lapin V., Lobareva I., Sharov S., Skorospelov V. //3D Euler flow simulation in hydro turbines: unsteady analysis and automatic design // Springer Series: Notes on Numerical Fluid Mechanics and Multidisciplinary Design – 2006 – V. 93, – P. 33-51.

63. Ву Мань Хиву, Попов С.А. //Проблемы моделирования течения в осевых вентиляторах аэродинамических труб//- Московская молодёжная научно-практическая конференция «Инновации в авиации к космонавтике - 2012». 17-20 апреля 2012 года. Москва. Сборник тезисов докладов. - М.: ООО «Принт-салон». Тираж 600 экз. «Инновации в авиации и космонавтике - 2012» Сборник тезисов докладов- Москва 17-20 апреля 2012

64. V. Ushakov, N. Sidenko, G. Filipsons. //Численный анализ особенностей аэродинамики и теплообмена цилиндра в вязком осциллирующем потоке//, - Starptautiski citejama anonimi recenzejama žurnala Научно-теоретический журнал Nr.6, 2011.g. “Автоматика и вычислительная техника”, Изд. - Институт электроники и вычислительной техники Латвийского университета 58-76с.

V. Ushakov, N. Sidenko, G. Filipsons, //Computer Analysis of the Aerodynamics and Heat Exchange of a Cylinder in a Viscous Oscillating Flow// - Automatic control and computer sciences,- Allerton press, inc a Division of Pleiades Publishing, Inc. ISSN 0132-4160 , http://elibrary.ru - Springer, SpringerLink, Secaucus, New Jersey, USA, - Ulrich's International Periodicals Directory, New Providence, USA. - VINITI (Россия) 346-360p.

65. Кутуков А.С. //Моделирование и расчет насосов при помощи программного обеспечения cfdesign//. - ООО «НИП-Информатика». Nr.4, 2009.

66. V.Ušakovs, N.Sidenko. //Žāvējamās mašīnas nestacionāras skaitliskās analīzes metodika.// - Mašinzinātne un transports, Intelektuālais Transporta sistēmas 18. sējums, 2004.g. 11-13. oktobri, Latvija, Rīga.- Rīga: Izd. "RTU" 2005.g., 102-112. lpp.

67. David Baraff. An Introduction to Physically Based Modeling: Rigid Body Simulation I—Unconstrained Rigid Body Dynamics.- Robotics Institute Carnegie Mellon University. This document is ©1997 by David Baraff. Siggraph '97 course notes physically based modeling. http://www.cs.cmu.edu/~baraff/sigcourse/notesd1.pdf.

68. Кондранин Т.В., Ткаченко Б.К., Березникова М.В., Евдокимов А.В., Зуев А.П. Применение пакетов прикладных программ при изучении курсов механики жидкости и газа.- Учебное пособие — М.: МФТИ, 2005. — 104 с.

69. Седов Л.И. Механика сплошной среды. - Том 1. Изд: «Наука». Ред. Физико-математическая Литература. - Москва 1970.- 492 с.

70. David Murray. Iside SolidWorks. - Second Edition Copyright (C) 2001 by on World Press is an imprint of Thomson Learning.

71. Шам Тику. Эффективная работа с SolidWorks 2004.- Изд.: Питер 2004

72. Прохоренко В. SolidWorks. Практическое руководство.- Изд.: ООО "Бином-Пресс" 2004.

73. Дударева Наталья, Загайко Сергей. SolidWorks 2007.- Изд.: БХВ-Петербург, 2007

74.. Касторский В.Е, Курочкин Ф.П. Практические работы по курсу воздушных винтов.- Изд.: Инженерная академия им. Жуковского. 1948 г.

75. Internet: http://www.profili2.com/

76. Lukaszewicz A. //Method for propeller reconstruction using reverce engineering// - 3th International Conference On Scientific Aspects Of Unmanned Aerial Vehicle. 2008.g. May 7-9, Poland, Kielce. - Rīga: Izd. "RTU" 2008.g., 412-421p.

77. Internet: http://www.tesis.com.ru/equip/kreon/

78. CFdesign v.9, User's guide. - Copyright (C) Blue Ridge Numerics, Inc. 1992-2007

79. Larry J. Segerlind. Applied Finite Element Analysis.- John Wiley and Sons, Inc. New York/London/Sydney/Toronto, 1976.

80. Zienkiewicz O.C. The finite element Method in Engineering Science.- McGraw-Hill company, Inc. London, 1971.

81. J.J. Connor, C.A.Brebbia. Finite element techniques for fluid flow. - Newness-buterworths, London-Boston, 1977- 264 p.

82. IZM projekts Nr. R7320 RTU TMF AI //Universālas datormodelēšanas sistēmas nestacionāru aerogāztermodinamisku uzdevumu risināšanai// 2007. gads.

83. RTU projekts Nr. R7330 TMF AI // Attiecībā pret plūsmu kustīgu ķermeņu nestacionāras aerohidrodinamikas un siltumapmaiņas īpatnību izpēte// 2008. gads.

CHAPTER 2

84. V. Ushakov, N.Sidenko //Численный анализ аэродинамических характеристик восстанавливаемого винта//- 4 th International Conference on the Scientific Aspects of Unmanned AerialVehicles-2010, Poland, Kielce 5-7 May 2010, - Izd.: Kielce University Of Technology Faculty of Mechatronics and Machine Building Chair of Information Technology and Armament Al. Tysiąclecia P.P. 7, 25-314 Kielce, Poland, 590-601 p. ISBN 978-83-88592-70-6.

85. V. Ushakov, N.Sidenko. //Aerodynamic characteristics of the vertical air jet behind the directing vanes of the aerodynamic propeller, installed on the ground.// The Third World Congress "Aviation in the XXI-st centery''- Safety in Aviation and Space Technology 22-24 September 2008.g. Ukraine, Kiev – Kiev: Izd. "National Academy of Sciences of Ukraine National Aviation University" 2008.g., 15.85-15.92p.

86. Projekts SIA „AERODIUM" //Vertikālās gaisa strūklas aerodinamiskie aprēķini un „gaisa plūsmas taisnotāja" skiču projekta izstrāde.// Rīga, Latvija, 2007 gads.

87. V. Ushakov, N. Sidenko, G. Filipsons. //Computer simulation and optimization of parameters of device, which creates a large diameter vertical air jet by means of symmetric system of fans.// - 2-nd International Specialized Symposium "Space & Global Security of Humanity" (SGS 2010). 5-9 July 2010.g., Riga, Latvia.- Riga: Izd. "Transport and Telecommunication Institute, 2010.g.", 96p., ISBN 978-9984-818-29-0. viperson.ru, http://spacesystems.tsi.lv.

88. Ushakov V., Sidenko N. //Development of computer simulation method and analysis of parameters of the vertical airstream generator for free flight of human.// - The 19TH

International Scientific and Technical Conference on Transport, Road-Building, Agricultural, Hoisting & Hauling and Military Technics and Technologies. Jule 1-4, 2011 Bulgaria ISBN 1310-3946, http://elibrary.ru.

CHAPTER 3

89. Latvijas Patents Nr. P-09-105 //Vertikālas gaisa strūklas ģenerators cilvēka brīvajam lidojumam.//, V.Ušakovs, N.Sidenko, G.Filipsons. Rīga, Latvija, 2009 gads.

90. V. Ushakov, N.Sidenko. //Оптимизация аэродинамического генератора вертикальной воздушной струи для свободного полета человека.//- IX International Scientific and Technical Conference «The Improvement of The Quality, Reliability and Long Usage of Technical Systems and Technological Processes». Conference Dedicated 80 Years CIAM, December-12-19, 2010 Sharm El Sheikh (Egypt). 81-85 p. ISBN 978-966-330-106-8.

91. Sidenko N., Filipsons G. //Simulating system nonsteady problems of Aerogasthermodynamics with immovable and movable boundaries.// - The 20TH International Scientific and Technical Conference on Transport, Road-Building, Agricultural, Hoisting & Hauling and Military Technics and Technologies. Jule 1-4, 2012 Bulgaria

92. Брусиловский И.В. Аэродинамика осевых вентиляторов. — М.: Машиностроение,1984- 240 с.

93. Ушаков К.А., Брусиловский И.В., Бушель А.Р. Аэродинамика осевых вентиляторов и элементы их конструкций.- М.: Госгортехиздат, 1960 г. - 422 с.

94. Дейч М.Е. «Техническая газодинамика» издание 2 М-Л: Госэнергоидат, 1961, 664 с.

95. Лаврентьев М.А., Шабат Б.В. Проблемы гидродинамики и их математические модели.- М.: Наука, 1977 г.- 408 с.

96. Мингалеева Г.Р. //Механизм движения жидкости и газа по спирали на участках крутого поворота тракта.// - Письма в ЖТФ, 2002, том 28 вып. 15. 79-85с.