ANNALS of Faculty Engineering Hunedoara
– International Journal of Engineering

Tome XII [2014] – Fascicule 2 [May]
ISSN: 1584-2665 [print]; ISSN: 1584-2673 [online]
a free-access multidisciplinary publication
of the Faculty of Engineering Hunedoara

[1.] Béla FODOR, [2.] László KALMÁR

FLUID AND HEAT ENGINEERING NUMERICAL ANALYSIS OF THE FLOW IN A TURBO BLOWER AGGREGATE

[1–2.]University of Miskolc, Faculty of Mechanical Engineering and Informatics, Department of Fluid and Heat Engineering, Miskolc, HUNGARY

Abstract: Numerical simulation of a radial turbo blower aggregate is to be represented in this paper, wherethe distributions of main flow parameters (e.g.: average temperatures and pressures referring to different flow cross sections) are determined inside the aggregate. During the simulationthe specific heat source produced by the aggregate's electromotor was taken into consideration in the calculation model of the blower aggregate. The calculated results are given a good insight for the flow structures inside the blower aggregate, which elements of the blower aggregate cause the nearly 50°C temperature increasing inside the aggregate during its operation. This temperature increasingcan lead to the growing of the power requirement and in case of long–term operation it can also lead to get damaged of the blower aggregate. One of the mains of this numerical investigation is to get information about the flow problems during the usage of the blower aggregate and by redesigning some parts of the aggregates to ovoid the development of operation problems mentioned above.
Keywords: CFD simulation, turbo blower aggregate

1. INTRODUCTION

Due to the increased public energy consumption, examinations trending to the reduction of power requirement of machines (used in the everyday life) come to the front significantly. For the determination of service parameters typical to the usage of these machineswe can apply measuring methods fixed in standard.In the last 10 years of industry significant demand came up for the application of so–called coupled numerical analyses beyond the many–sided utilization of computer aided design. Overall we name these examinations to simulations. Simulations start long (months or even years) before the prototype production of the actual equipment on behalf of manufacturing the product at the lowest cost and energy investment and in the most optimal construction.

Numerical analyses (have to be executed) put up significant computer demand towards the increase of accuracy and the reduction of calculating time. In point of calculating methods several mathematical description mode can be found in the relevant professional literature. In the classic CFD (Computational Fluid Dynamics) we simulate the real flow field with a simplified and in most cases 3D calculating range. Then we split the developed calculating range with the aid of a numerical lattice, and we search the numerical solution for the discretizated field by the application of so–called 'finite volumes' method. So we divide the full calculating field to finite number check volumes, where we take the calculating point into the central point of check volume [1]. In the calculating points we determine the discrete value of the basic equations of flow (used in the course of simulation) belonging to the initial and boundary conditions.Based on the received computed results we can create the most important characteristics of the examined range's time–

variable service condition and we get an exact picture about the flow relations evolving inside the machine.

In view of the mathematical model we spread out our analyses to the determination of a turbo blower aggregate's fluid mechanical and thermo–technical parameters, in the course of it, based on the flow relations of the aggregate we discovered the map, which helps the definition of the potential measuring points of the instrumental measurement. Besides we examined the location of heat forming during the operation of the aggregate and its spreading way.

The concerned tests are extensible to strength analyses or even acoustic analyses too, but we don't deal with it in the frame of this contribution.

2. CHARACTERISTICS OF THE TURBO BLOWER AGGREGATE'S RANGES

The examined turbo blower aggregate is such a fluid mechanical equipment group, which consists of two machines. The first machine unit is the turbo blower, which provides the pressure rise required to its operation.The other machine unitis the synchronous motor, which ensures the drive of the turbo blower (by their assembly on the same axis).The air delivered by the turbo blower provides the cooling of the drive motor besides the support of the aggregate's operational running.

Construction of the two machine units determines the ranges important from the aspect of numerical analysis. One of the ranges is the examined turbo blower's range, which is a unit equipped with one rotor, and furthermore with guide– and return guide wheels, of which the pressure ratio value between its suction– and pressure connection is π= 1,15. The other range is the engine space, which consists of a wound rotor and a stationary core. The rotor and the engine are coated. Figure 1.represents the structural construction and the above mentioned major parts of the turbo blower aggregate.

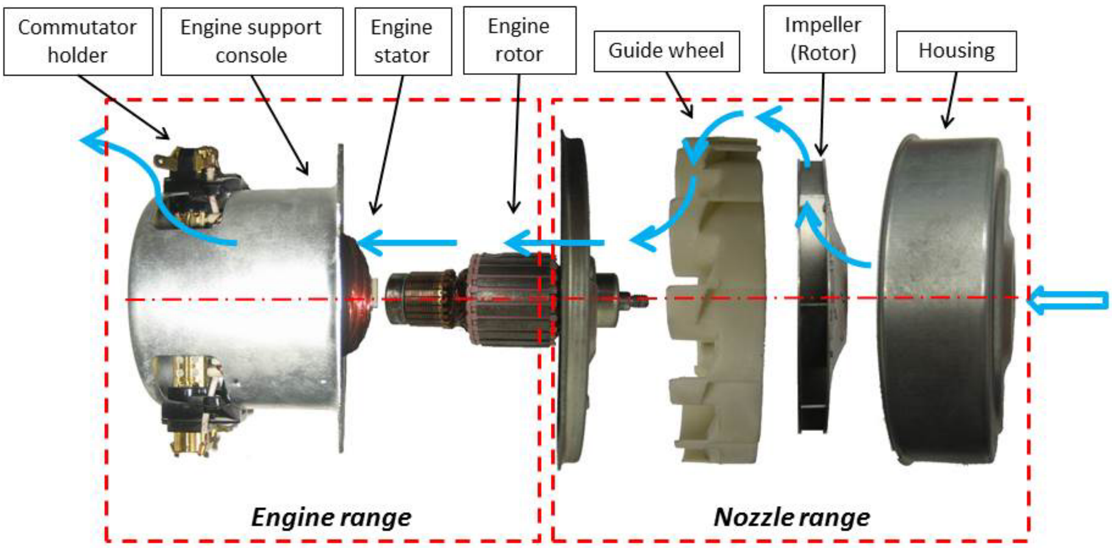


Figure 1.Structural construction of the aggregate in disassembly

Fitting to the structural design, first we divided the whole calculating range of the aggregate to subdomains. Due to the turbo blower's operation we can separate two flow fields with fundamentally differing properties. According to the Figure 2.a we named the range containing the rotor toROTOR and the stationary range to STATOR, which is bounded by the housing of turbo blower aggregate and the walls of guide wheel. The stationary and rotary units of the engine are located in the flow field called STATOR. During the simulation we considered these parts with the bounding surfaces.

By the application of the available geometrical typical and the graphical preliminary program of the ANSYS–FLUENT commercial software we created the total calculating range and numerical lattice ([2] és [3]) of the turbo blower aggregate for the preparation of numerical calculations.

3. BOUNDARY CONDITIONS AND CHARACTERISTICS OF NUMERICAL ANALYSIS

The primary purpose of numerical analysis is to calculate the service parameters of the mentioned turbo blower aggregate, furthermore the more detailed determination of values of the fluid mechanical and thermo–technical properties evolving inside the aggregate. During the operation of the aggregate the streaming air arrives to the rotor on the suction hose (marked with IN) of the turbo blower, which raises the energy of the liquid due to its rotation with external energy. After the air gets into the pressure chamber through the guide wheel on the rotor side, then it passes from the turbo blower unit on the pressure hose, streaming through the back side return guide wheel. Following it, medium gets into the engine space, where it traverses through the rotating coils and stationary core performing circumfluent movement, then it quits from the testing range through the two outlet ports of housing (marked with OUT1 and OUT2).

Inside the aggregate, one part of the heat evolves on the coils of driving engine, which is – considering measured and standard data – nearly 800W on the engine's rotor. That's why we regard the delivered medium as compressible. It can be established that significant heat evolves in the engine space of the driving engine during operation, however the heat development from the friction of medium is not irrelevant either.Based on the preliminary laboratory tests temperature growth rate can be even 50°C. Besides these, the aggregate does not require plus cooling, because medium (delivered by the nozzle) carries away the major part of originated heat.

In the course of performing numerical analyses we applied the following specifications to the boundary condition (henceforth Bc.), which we present hereinafter, using the markings of figure 2.a:

- in the suction side inlet cross section of turbo blower aggregate(IN) we specify the inlet mass flow rate $\dot{m}$ of the streaming air, (Bc.: „inletmassflow");
- in both outlet cross sections (OUT1 and OUT2) of the pressure side we specify the value of outlet pressure p_k, (Bc.:„outlet pressure");
- we separate well the inner bounding walls of the aggregate's nozzle part to type„wall";
- we define the calculating range of turbo blower rotor (ROTOR) with the neighbouring stationary range (STATOR) and their joint with a relevant „interface" surface;
- we change the rotor surface located in engine space as well to type „wall", where considering the size of rotor surface we specify 800W inlet heat power;
- we define further surfaces present in range to type„wall".

During the calculation process we calculate the actual average value of p_t absolute pressure evolving in the inlet cross section "IN" of the turbo blower aggregate's suction side, which we can monitor continuously in the course of time–variable simulation. In view of the outlet pressure p_k the pressure differences $\Delta p = p_k - p_t$ can be determined relating to the turbo blower aggregate [4]. As a result of simulation p_t pressure data has been determined, due to this value, parameters typical to the operation of the aggregate can be originated, like n speed, m mass flow and Δp pressure difference. We considered the following operating characteristics for the analysis: speed: 36480min^{-1}, volume flow rate: 28,2ℓ/s, rotor diameter: 104mm.

In accordance to the above mentioned we considered the compressibility of the streaming medium with the application of general gas law during the 3D analysis. We used the „SST k–ω"turbulence model for the numerical solution, furthermore we approximated variables conceived in the finite volumes in second grade.

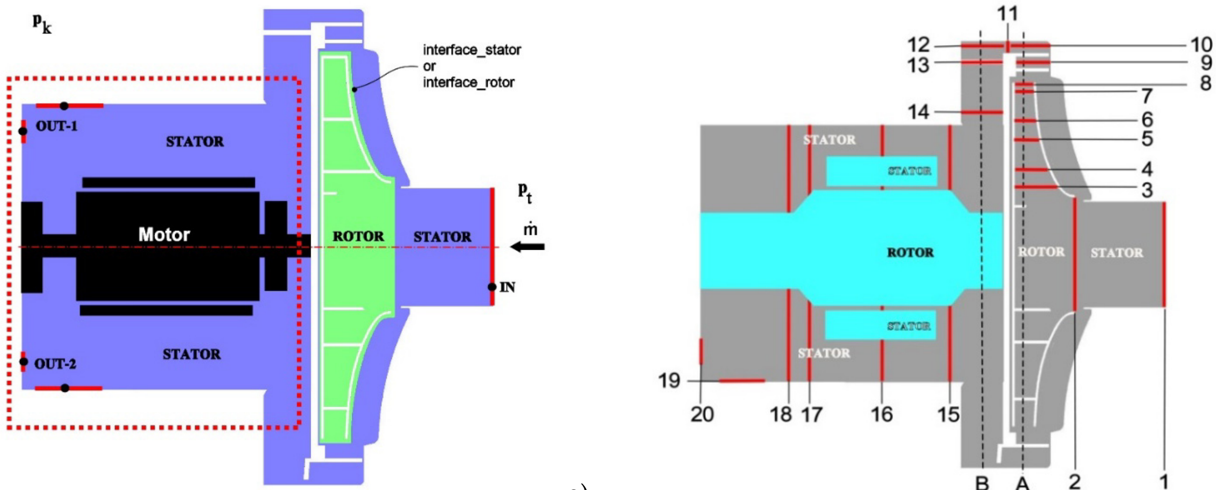
4. INTRODUCTION OF TEST RESULTS

Results received by the aid of simulation make possible the analysis of flow field evolving in the whole aggregate.

Data received during solution are available in the discrete points of the range, where on different – plain or curved – surfaces temperature field is determinable beside the flow parameters. For the exact mapping of the machine's operation we defined the most important parameters' average

ISSN: 1584-2665 [print]; ISSN: 1584-2673 [online]

value relating to the cross flow area in 20 pieces of different cross flow areas. With the aid of figure 2.b the main flow direction of the medium is easily perspicuous too along the increasing serial number of cross sections marked with 1–20. For the graphical display of the evolved temperature distribution we used further sections – located perpendicularly to the spin axis – marked with A, B, and also the so–called meridian section fitting to the axis of the aggregate.



a) b)

Figure 2. Drawing of the examined sections' arrangement

For the rotor's numerical analysis we need to analyse separately the regions in front of the rotor's front panel and behind the rotor's back panel. On behalf of it, at cylinder sections marked with 3–8 we divided the cross flow area to three subdomains: „Rotor inner cross–section", in front of the **"**Rotor front panel" and behind the "Rotor back panel".

In the above listed sections signed with number 1–20 we determine the average values of flow parameters (for example temperature, pressure, components of speed, etc.) relating to the cross flow area with molar averaging. We performed the averaging with the application of simulation values available in the nodal points of finite volumes located on the assigned cross flow area and in the nodal points of the surface.

Using the markings of figure 2.a and as a result of simulation we defined the average temperature distribution, the absolute – and dynamic average pressure distribution, furthermore the average density and mass flow of the medium evolving in the pointed sections. Because of extension reasons now we present changes only on the average temperature.

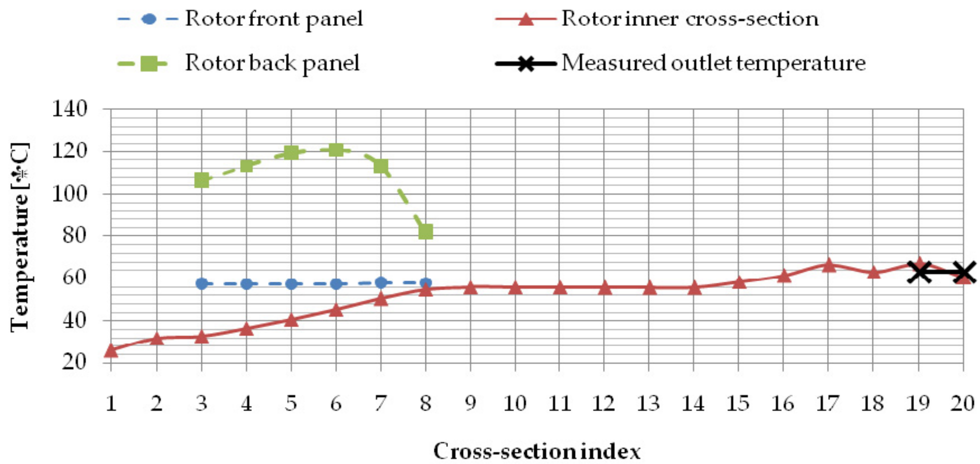


Figure 3.Calculated average temperature values [°C] are in cross–sections noted by 1–20 along the aggregate's flow channel and measured in the outlet cross–sections

We can find the average values belonging to cross flow areas of temperatures (evolved along the cross–sections 1–20) on Figure 3. We also represented here the average temperature of cross–sections in front of the rotor front panel and behind the rotor back panel in cross–sections 3–8. The air temperature in front of the rotor front panel is almost 60°C, which is the consequence of reverse flow – because of high leakage loss – evolving in the front space of rotor.

It can be seen well, that the local increase in temperature (evolving behind the rotor back panel) can be even 80°C, which forms out due to the significant dissipative loss evolved during the medium's very fast flow in a very narrow leak. In addition, the major part of the increase in temperature (occurring in the medium streaming further) generates to the effect of heat evolving on the coils of engine space and arriving into the flow space.

Figure 4 introduces the temperature distribution originated on the typical surfaces of the aggregate.On the figure we can see the nettedsurfaces of the rotor and the temperature distribution along the so–called meridian section (fitting onto the rotation axis) relating to the aggregate suction hose and the flow fieldin front of the rotor. On Figure 4 it can be also seen, that 20°C temperature medium arrives into the aggregate, then the air temperature – in the environment of the rotor's outlet cross–section – increases to 55–60°C.

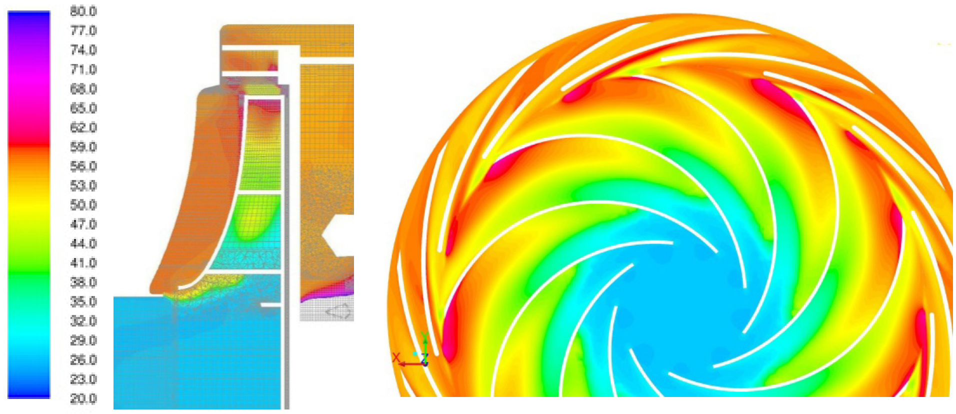


Figure 4.The aggregate's temperature distribution on a meridian direction (in left side of the figure) and an axial direction (on right side of the figure) – pointed in the rotor's blade channel – section marked "A"[°C]

5. SUMMARY

The introduced numerical simulation procedure is suitable for the execution of fluid and heat engineering numerical analysis in case of a turbo blower aggregate, with its aid the flow phenomenon inside the turbo blower aggregate can be analysed locally and globally too. Knowledge of the calculated results and those analysis forwards the further development of the turbo blower aggregate even in the phase of design.

By the analysis of the detailed results – received from the presented numerical simulation – we can find out which part of the aggregate should be modified on behalf of the service parameters' improvement (in our case the reduction of heat development evolving inside the equipment, etc.).

We compared the simulation results with the results performed with measurement, which showed good conformity in the outlet cross–sections (No. 19 and 20) according to the distribution represented on Figure 3 [4]. Due to extension reasons we don't deal with its introduction in this contribution.

One of the future goals of the numerical analysis is to make possible the performance of a numerical analysis comparing aggregates with similar construction, but other geometry per further development of the numerical method, by which, new and modern turbo blower aggregates can be developed.

Acknowledgements

Research work was carried out in the Mechanical Design Innovation and Technologies' Excellence Centre operating in the strategic research area of University of Miskolc, as a part of TÁMOP–4.2.1.B–10/2/KONV–2010–0001 project – in the frame of New Hungarian Development Plan –, with the support of European Union, by the co–financing of European Social Fund.

Furthermore we render thanks to Electrolux Kft. for the aid provided in the preparation of simulation.

Bibliography/References

[1.] ANSYS Inc., Fluent Theory Guide, Release 14.0, Canonsburg, PA, http://www.ansys.com

[2.] Fodor, B., Kalmár, L.: Turbófúvóaggregátbankialakulóáramlásnumerikusvizsgálata, 21th International Conference in Mechanical Engineering, Románia/Arad, 2013, pp. 118–121, (ISSN 2068–1267) Lang.: HUN

[3.] Kalmár, L., Janiga, G., Fodor, B., Varga, Z., Soltész, L.: Egy fokozatú turbófúvó numerikus modellezése, Numerical investigation of the flow in one–stage blower GÉP 2012/1. LXIII évfolyam, pp. 39–42., Miskolc (ISSN 0016–8572), Lang.: HUN

[4.] Kalmár, L., Szabó, Sz., Janiga, G., Fodor, B., Soltész, L.: Egy fokozatú turbófúvóaggregátokáramlási jellemzőinek numerikus és kísérleti meghatározása, Szivattyúk, kompresszorok, vákuumszivattyúk, XX. évfolyam, 2013, pp. 41–48., (ISSN 1219–1108), Lang.: HUN

ANNALS of Faculty Engineering Hunedoara – International Journal of Engineering