*Conference on Modelling Fluid Flow (CMFF'22)*
***The 18th International Conference on Fluid Flow Technologies***
***Budapest, Hungary, August 30 -September 2, 2022***

# IDENTIFICATION OF LOW FREQUENCY FLUCTUATION IN CENTRIFUGAL FAN

Magdalena STANIK[1], Dominika JASKÓŁA[1], Dominik DEDA[1]

[1] Department of Power Engineering and Turbomachinery, Silesian University of Technology, Konarskiego 18, 44-100 Gliwice, Poland
* magdalenastanik1@gmail.com

## ABSTRACT

Centrifugal fans are the most common turbomachines used in technical applications. Many efforts have been made to find the most efficient numerical method of modelling flow in such turbomachines. However, only a few publications focus on the unsteadiness that may have an impact on device efficiency and noise generation. The main goal of this study was to develop an experimental method to investigate or investigate the frequency of oscillation inside the radial fan that can affect the noise. The focus was on measuring pressure changes inside the fan casing, which defined the non-stationary fluctuations in the airflow. The geometry of the fan and the volute were modelled together. The geometric model allowed for the generation of the fluid domain, discretization of space and the construction of a mathematical model describing flow and non-stationary phenomena occurring inside the fan machine. The geometrical model was created using the Solid Works software and the numerical investigation was done using the ANSYS Fluent solver. Computer calculations were compared with the results of experimental research carried out on a 3D printed model. Further, the numerical analysis was validated against the experimental results. Graphs of the dependence of the amplitude (pressure) as a function of time and frequency were obtained. Characteristic frequencies for the largest pressure amplitudes and other relationships describing acoustic phenomena in the centrifugal fan were obtained. The above-mentioned results are the results of a preliminary banking study, which lay a solid basis for subsequent research that is to be taken.

**Keywords: Acoustics, Centrifugal fan; CFD; Fourier analysis.**

## NOMENCLATURE

| | | |
|---|---|---|
| $\dot{V}$ | $m^3/s$ | volumetric flow rate |
| $\Delta p$ | Pa | pressure increase |
| $t$ | °C | temperature |
| $n$ | rpm | rotational speed |
| $b$ | - | blades number minute |
| $d$ | m | diameter |
| $f$ | Hz | frequency |
| $\dot{m}$ | kg/s | mass flow rate |

## INTRODUCTION

Fans, blowers, and flow compressors are working machines designed to transport or compress the gas, one of these activities may dominate the other. Blowers, unlike compressors, compress the medium to relatively low pressure, the contractual limit is circa 0.29 MPa. In many cases, overcoming the resistance is the main task of the fan, not the delivery of compressed gas. External work put on the rotor shaft forces the gas, at the same time giving it an increase in kinetic energy. As the energy transfer process takes place continuously during the flow through the rotating fan channels, it is classified as a continuous flow machine. The name of the radial fan tested in the project comes from the radial outflow of the medium from the rotor. It takes place approximately in a plane perpendicular to the axis of rotation, and the meridional components have radial velocities or are slightly deviated from this direction. The centrifugal fan folds up from the rotor mounted on the motor drive shaft. In the centrifugal fan, the rotors are made of two discs (supporting and covering), between which the blades are fixed [1]. The main source of sound in the ventilation system is the fan. Fan noise has different origins. The primary factor is the fan design itself, the number of blades and their shape takes a significant role. Performance, pressure, air velocity, size, and shape of the casing are other significant factors. A general breakdown of noise can be made into noise due to aerodynamic and mechanical factors. The aerodynamic noise is directly related to airflow through the vane system and the accompanying changes in the surface pressure in the blade-to-blade channel. The sources of mechanical noise are influenced by: imbalance of the rotor, incorrect operation of the bearings, operation of the electric motor, and possible mechanical vibrations of components caused by stiffness of the structure. The area, where acoustic vibrations occur is called the acoustic field. This field describes the propagation of sound in an idealized space in which the influence of the bonding surfaces and the items in it. The distribution of this field is negligible. Under certain conditions, when the influence of the surfaces limiting the movement of the acoustic wave is insignificant, noise measurements can be made assuming that the acoustic field has the characteristics of a free field. A diffuse sound field is an area where a sound wave reflects so many times that it moves in all directions with the same amplitude and probability. The most frequently measured size of the acoustic field is the acoustic pressure, which is the difference between the environmental pressure at a given moment and static pressure. On its basis, it is determined, by sound intensity or sound power. Fan tests on fans can be carried out by two main methods, i.e. quantitative (balance) and qualitative (structure test of flow). Quantitative research includes mass and energy balance, and qualitative research includes invasive and non-invasive measurements. The numerical research (CFD) mentioned in the article belongs to both groups simultaneously. [7] Significant advances in gas flow design methods using numerical fluid mechanics and increases in the computational efficiency of computers have enabled the development of research perspectives in the field of fluid flows. This method gives the possibility of imaging taking into account the flow and fluctuations in the design process of the radial fans. [9] Thanks to the numerical method it is possible to consider complex flow phenomena occurring in rotating machinery. From the results obtained employing the CFD tools, a frequency analysis was performed, and the Furrier transformation made it possible to change the signal into an amplitude spectrum[8]. Hydraulic losses and pressure drops are influenced by many factors, such as the housing structure or the impeller itself[10]. As it is known from the research, even such trivial aspects as the gap between the inlet and the rotor influence pressure losses and machine efficiency [10]. Poor design can lead to unwanted flow interruptions and the formation of turbulence that will adversely affect the performance of the entire machine. [11]

## CENTRIFUGAL FAN MODELS

The studies were conducted simultaneously at the Department of Power Engineering and Turbomachinery (DPET) at the Silesian University of Technology (SUT) and the Institute of Thermal Turbomachinery (ITT) at Karlsruhe Institute of Technology (KIT). The models were designed for a similar design point, however, they differ in dimensions. Nevertheless, they can be compared according to fan similarity theory. Real radial fans were built, the design of which was based on a series of calculations carried out by A. Get [2]. The number of rotor blades was calculated, and the length, central as well as outlet angle and the skeleton radius of the blade were calculated according to mentioned reference. Further, they were designed in Solid Works and manufactured employing Nouvel fast prototyping tools, i.e. 3D printing. The table presents the design parameters for which the system elements were designed. Figure 1 shows the SUT fan geometry.

**Table 1. Fans design parameters**

| Variable | KIT | SUT |
|---|---|---|
| $V$, m$^3$ /s | 0,05 | 0,05 |
| $\Delta p$, Pa | 63 | 50 |
| $t$, ° | 20 | 20 |
| $n$, rpm | 600 | 1000 |
| $b$, - | 9 | 8 |
| $d$, m | 0.325 | 0.25 |

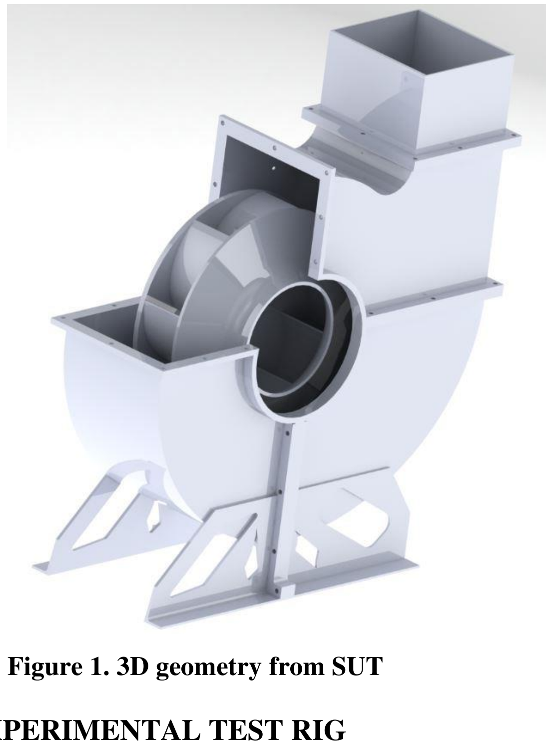

**Figure 1. 3D geometry from SUT**

## EXPERIMENTAL TEST RIG

Both experimental stands have been created to conduct acoustic tests of the fan model. Advanced printing methods were used to create an experimental model. The test rig in Karlsruhe is illustrated in Figure 2.

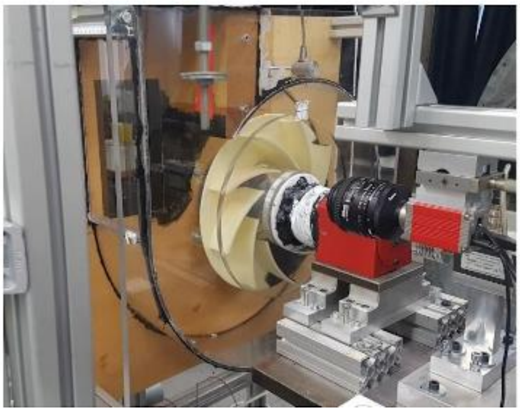

**Figure 2. The centrifugal fan test rig of KIT**

The hub, the spiral casing as well as the sidewall of the housing hub are made of acrylic glass for optical accessibility. To record the performance curve of the centrifugal fan, pressure values are taken upstream and downstream of the fan what allowed for determining the total pressure increase. Pressure measurement in the straight pipe gives the volume flow rate according to DIN EN ISO 5167. Additionally, the temperature sensor at the inlet as well as the outlet pressure and temperature sensor were mounted. The working point can be shifted along the performance curve by either closing the throttle or powering the auxiliary fan. The inlet section of the test fan was designed to allow determining boundary conditions for the numerical simulations.

The SUT test rig was designed to meet similar flow parameters of the KIT fan. The rig was manufactured employing 3D printing. An electric motor powered by 24 V DC with a 10:1 gear ratio was selected. The stand was also equipped with basic measuring elements: a tachometer and an anemometer. The acoustic measurements were carried out using a probe by PCB PIEZOTRONICS [3]. It is a compact sound pressure measuring device that allows measuring the pressure fluctuation in the flow field that is a source of the noise. The probe is constructed with a detachable stainless steel tub, whose task is to provide an acoustic signal into the microphone inside the probe housing. The microphone with the ICP probe, model 377B26, was placed in the two measuring points shown in the figure below. Measurements were carried out in the area of the air outlet of the impeller as the highest velocity and pressure values are expected there, see Figure 3. Point 1 is halfway between the impeller and the casing, point 2 is offset from the impeller by the same distance. Point 2 is also near the volute tongue, where the greatest turbulent movements happen.

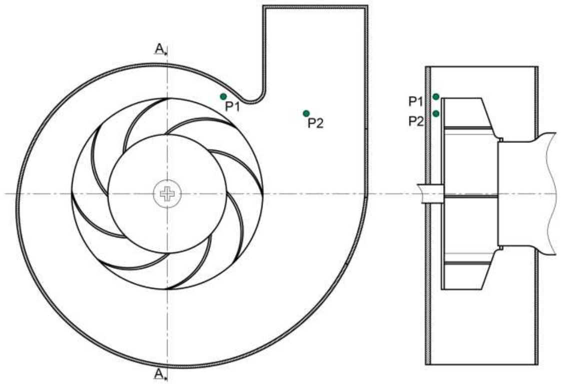


**Figure 3. Measuring points in a model of SUT.**

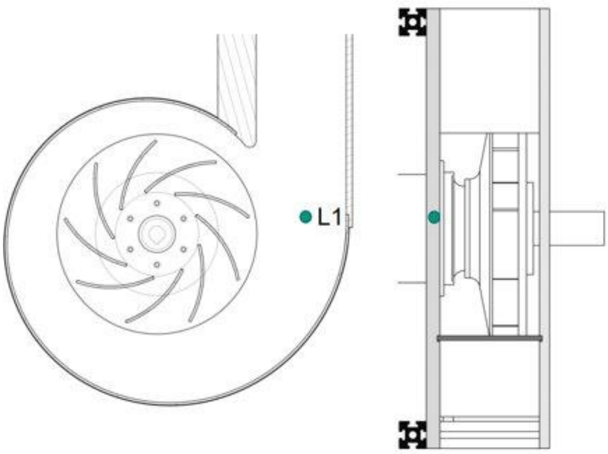


**Figure 4 Measuring points in a model of KIT.**

## NUMERICAL EXPERIMENT

For both models, the simulations were carried out in an unsteady state. Direct numerical simulation (DNS) methods solve the Navier-Stokes equation without any simplifications and predict the unsteady flow including all fluctuations spectra in space and time, and thereby the acoustic field. However, such methods are very demanding in terms of computational time and are not available for technical applications at the moment. Similarly, large eddy simulation (LES) methods are not practical for these applications either, because the requirement regarding the numerical mesh resolution is extremely difficult to meet. Therefore, it was decided that the appropriate method will be unsteady Reynolds-averaged Navier-Stokes (URANS) complemented with Metner shear stress transport (k-ω SST) and the scale adaptive simulation (SAS-SST) turbulence models will be used and compared. The computation domain is split into two parts, the flow field in the volume containing the fan is computed in a rotating frame, and the inlet and the discharge duct are set as stationary. It is important to note that the frequency range is also limited by the mesh spacing. The boundary conditions for the KIT model were selected for the fan nominal load, i.e. the volume flow rate of 0.0637 kg/s. This value was set up at the outlet, whereas at the inlet total parameters were assumed, i.e. total pressure of 1 bar and total temperature of 300 K. The rotor rotational speed is 600 rpm. The simulations are conducted on a coarse and a fine mesh. The numerical mesh for the case under analysis contains about $0.65\times10^6$ elements for the rotor domain and $0.8\times10^6$ elements for the stator domain, which gives almost $1.5\times10^6$ elements in total. The numerical mesh is of a tetrahedral type with prisms in boundary layers. The $y^+$ value ranged from 1 for the rotor blades to 3 for the outlet port of the volute.

Figure 5 shows the fan characteristics as the relationships between the pressure rise, efficiency, and the volume flow rate for the rotor constant speed (600 rpm). Results were obtained experimentally and from steady-state CFD simulations with the SST turbulence model on the coarse mesh. It is noticeable that there is a very good convergence between the experimental and numerical results. The adopted numerical method well captures the operating conditions of the centrifugal fan under consideration.

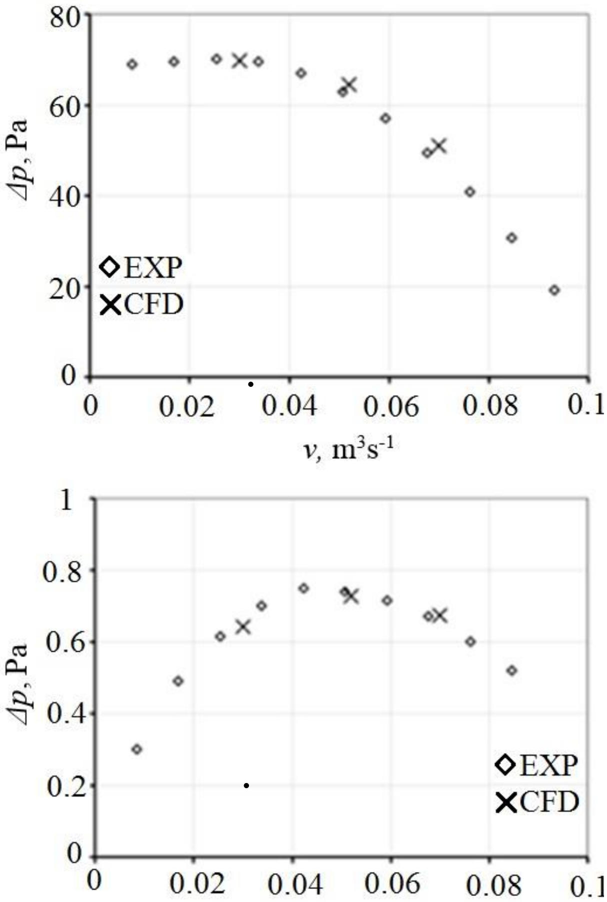


**Figure 5. Fan characteristic – comparison of the experiment with CFD modelling**

The document presents the results of the transient state from SUT. The experiment was based on the pressure difference between the inlet and outlet of the domain. The fan is designed for a pressure difference of 50 Pa. The design pressure value was also used as a boundary condition for the calculation. The rotor rotational speed is 1000 rpm. The domain of the chasing and rotor consists of $2.1\times10^6$ elements in total. Results of CFD analysis do not show full agreement with the experiment. This may be due to insufficient mesh quality. However, a more likely reason is the rotor design itself. The applied computational model allowed for an approximate construction of the rotor but did not include all the exact parameters allowing for a sufficient accuracy of the representation of the blades. A coarse mesh sensitivity analysis was conducted to identify if the under-densified mesh is the source of the failure to achieve the required accuracy. Each of the prepared meshes was characterized by the different number of elements or approaches to element size distribution. A feature of the coexistence of all used numerical grids was the application of cells of specific shapes inside the computational domain. It showed that the further increment of the mesh elements gives no meaningful change in occurring flow phenomena. The time step is set to equal 1° displacement of the rotor.

The numerical analysis allowed us to determine the speed inside the fan housing. It is visible that the highest speeds occur in the vicinity of the vanes runoff and at the impeller outlet, see Figure 7. Undesirable air movement is observed which is returned from the volute to the impeller, see Figure 7. This is due to the gap between the rotor and the outlet. Regions of increased velocity coincide with regions of increased pressure

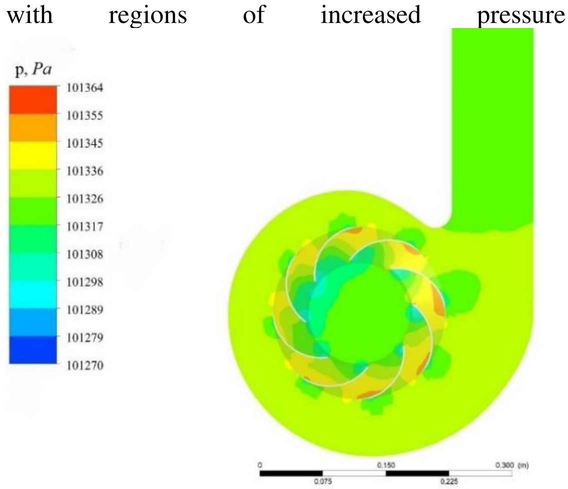


**Figure 6 Total pressure contours in SUT fan.**

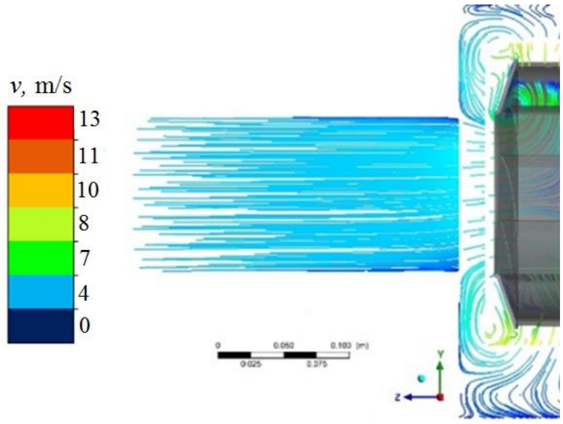


**Figure 7. Velocity contours for model form SUT.**

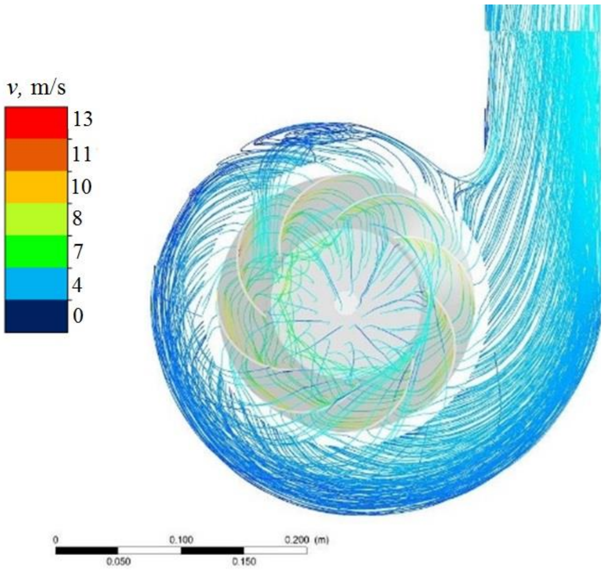


**Figure 8 Velocity counts for the model from SUT.**

## FREQUENCY ANALYSIS OF PRESSURE FLUCTUATIONS.

Measurement conducted in KIT, and SUT allowed for obtaining several different waveforms, being the dependence of the voltage signal (mV) on time (ms). The obtained voltage signal was converted into a pressure value according to the conversion factor included in the device manual further, the Fourier analysis was performed for selected 1024 time samples. Graphs of the dependence of the amplitude (pressure) as a function of time and frequency were obtained. Figures 9 and 10 show the obtained spectrum of pressure fluctuation based on the experiment whereas. Figures 11 and Figure 12 depict the pressure fluctuation spectrum based on the numerical experiment. As it can be seen in the experiment, there appears a tonal noise with a frequency lower than Blade Passing Frequency (BPF).

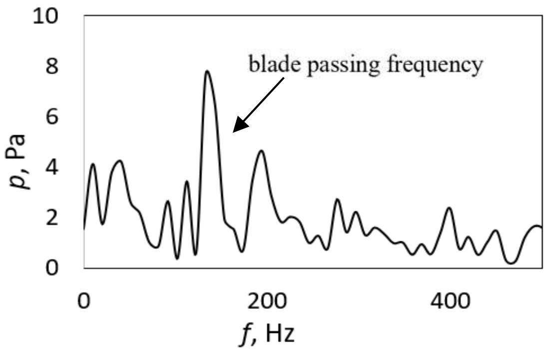


**Figure 9. Pressure fluctuation spectrum based on the experiment, point 1, SUT**

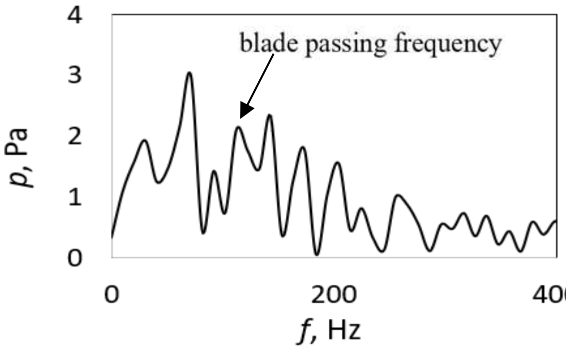


**Figure 10. Pressure fluctuation spectrum based on the experiment, point 1, SUT**

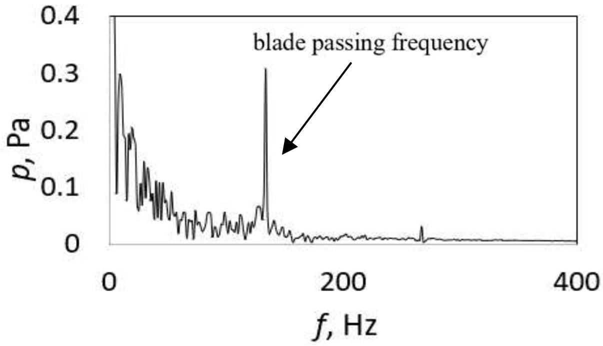


**Figure 11 Pressure fluctuation spectrum based on the CFD, point 1, SUT**

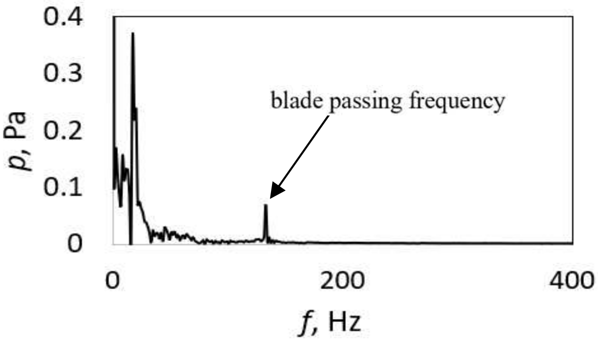


**Figure 12 Pressure fluctuation spectrum based on the CFD, point 2, SUT**

Presented FFT analysis from SUT shows that for both experiment and CFD analysis it is possible to distinguish blade passage frequency. However, this phenomenon is much more noticeable in point 1 located closer to the rotor. It can be seen that in point 2, the frequency of the passing blades is less significant, while other pressure peaks become more pronounced and rach the value of blade passing frequency at this part of the model. Given the location of the point, it can be concluded that the observed fluctuations are associated with increased tidal turbulence. This may be due to interactions with the casing, which would confirm the study.

Figure 13 depicts the pressure fluctuations obtained numerically. The BPF obtained from the URANS simulations with the SST and the SAS turbulence model totals 97 Hz, which seems to be a bit too high. In the result of the simulation based on the SAS turbulence model, there is a frequency that is half the BPF value. Additionally, significant differences appear between the two models for higher frequencies, exceeding 1.5 kHz. The maximum amplitude of the pressure fluctuations is about 0.7 Pa. It corresponds to the blade passing frequency (BPF) and shows harmonic features.

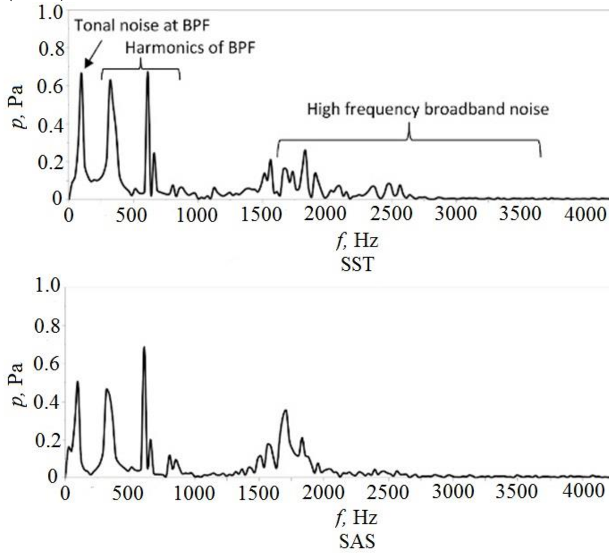


**Figure 13. Pressure fluctuation spectrum based on CFD, KIT [4]**

## CONCLUSIONS

The main objective of the presented research is to find a research workshop. The results of collaboration recently started between the Department of Power Engineering and Turbomachinery of the Silesian University of Technology and the Institute of Thermal Turbomachinery of Karlsruhe Institute of Technology to investigate the flow field with a focus on the noise emission within a centrifugal fan is presented in this paper. The research is both experimental and numerical. Although a complete method for identifying noise sources in flow machines has not been fully developed, the results seem promising. The numerical model has the potential for deep investigation of noise emission due to the pressure fluctuation in centrifugal fans. Studies conducted by the two universities, despite their differences, show considerable similarity in FFT analysis. In both, the KIT and SUT experiments it can be seen that not only blade passing frequency but also the low frequency fluctuation is present. However, its sources, due to different ranges of values, have not been found.

The presented centrifugal fans have been designed for nominal load and operating conditions. Taking into account the similar results of both party studies, the opportunities for improvement are identified which may lead to a deeper understanding of flow phenomena in the most popular turbomachinery. It is also worth remembering that due to the interaction with the piping system, temporary drops in the flow rate may occur. The main aim of that paper was to extend numerical calculations so that they illustrate low frequency flow fluctuations. Achieved results show that numerical methods were correctly chosen and allowed the presentation of phenomena which are the topic of the paper. It has to be emphasized that this was a primary experiment to prove the research gap that has to be studied to provide more sustainable machines.

## REFERENCES

[1] S. Kuczewski, Wentylatory Promieniowe, Warszawa, 1966.

[2]A. Gjeta, L. Malka “Outlet Surface Area Influence in Spiral Casing Design on Centrifugal Fan Performance,” European Journal of Engineering Research and Science, January 2020.

[3] Probe Microphone ICP®, Installation and Operating Manual

[4] B. Pritz et al. Identification of unsteady effects in the flow through a centrifugal fan using CFD/CAA methods.

[5] S. Dykas et al., “Numerical method for modelling acoustic waves propagation.”.Arch.Acoust.35(2010),.

[6] S. Fortuna, K. Sobczak; “Numerical and experimental investigations of the flow in the radial fan.”.Mechanics 27(2008), 4,138–143.

[7] T.Siwek,”Badania przepływów w wentylatorze z wirnikiem promieniowym o zabudowie osiowej”, Kraków 2017

[8] G.Szwoch „Analiza częstotliwościowa na procesach sygnałowych”, Politechnika Gdańska

[9] S.Boroń, P.Kubica, „Zastosowanie numerycznej mechaniki płynów CFD do modelowania zabezpieczania pomieszczeń stałymi urządzeniami gaśniczymi gazowymi”, CNBOP-PIB 2016

[10] A.Gejta, „Effect of Clearance Gap in Spiral Casing Design of a Centrifugal Fan with Optimized Impellers”, European Journal of Engineering Research and Science · September 2019
[11] Yu-Tai Lee, „Impact of fan gap flow to the centrifugal impeller aerodynamics”, Glasgow 2012