EDITORIAL AND PUBLISHING OFFICE

Wydawnictwa IMP (IFFM Publishers), The Szewalski Institute of Fluid-Flow Machinery, ul. Fiszera 14, 80-231 Gdańsk, Poland, Tel.: (+48) 58-52-25-141, Fax: (+48) 58-34-16-144, E-mail: redakcja@imp.gda.pl; jfrk@imp.gda.pl http://www.imp.gda.pl/transactions-of-iffm/

© Copyright by the Szewalski Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Gdańsk 2016

Typeset in \LaTeX

Printed and bounded by Fabryka Druku Sp. z o.o.
ul. Zgrupowania AK Kampinos 6, 01-943 Warszawa

Terms of subscription
Subscription order and payment should be directly sent to the Publishing Office (The Szewalski Institute of Fluid-Flow Machinery, ul. Fiszera 14, 80-231 Gdańsk, Poland; Attn. Mrs. Aleksandra Nowaczewska, e-mail: now@imp.gda.pl). Osiągalne są również wydania archiwalne.

Warunki prenumeraty w Polsce
Wydawnictwo ukazuje się cztery razy w roku. Cena numeru wynosi 25,- zł. Zamówki z określeniem okresu prenumeraty, nazwiskiem i adresem odbiorcy należy kierować bezpośrednio do Wydawcy (Wydawnictwa IMP, Instytut Maszyn Przepływowych im. Roberta Szewalskiego PAN, ul. Fiszera 14, 80-231 Gdańsk; e-mail: now@imp.gda.pl). Osiągalne są również wydania archiwalne.

Articles in Transactions of the Institute of Fluid-Flow Machinery are abstracted and indexed within:
INSPEC Database,
Energy Citations Database,
Applied Mechanics Reviews,
Abstract Journal of the All-Russian Inst. of Sci. and Tech. Inf. VINITI,
BazTech,

ISSN 0079-3205
Artur Andrearczyk\textsuperscript{a, b}, Grzegorz Żywica\textsuperscript{a}

A concept of a test stand for the investigation of a 3D printed turbochargers and selected fluid-flow machinery

\textsuperscript{a} Department of Turbine Dynamics and Diagnostics, Institute of Fluid Flow Machinery, Polish Academy of Sciences, 80-231 Gdańsk, Fiszera 14, Poland  
\textsuperscript{b} Gdańsk University of Technology, Narutowicza 11/12, 80-233 Gdańsk, Poland

Abstract

The paper describes the general concept of using the rapid prototyping methods and their application to manufacturing of selected components of rotating machinery. Chosen rapid prototyping technology (multijet printing) allows precise manufacture of rotor models with complex geometry, without the need for their further processing. The research is planned to be performed on designed test rig, which is described in the paper. 3D printed elements will be pretested using air from a high performance compressor. Basic preliminary testing and own experience has shown that this technology is faster, cheaper and more accurate than conventional manufacturing. This fact can significantly contribute to the development of production and research of prototypical fluid-flow machines. The tensile strength testing of the two materials which are used in a 3D printer in different printing directions has been also discussed.

Keywords: Rapid prototyping; Polymeric resin; Fluid-flow machines

1 Introduction

Rapid prototyping technologies are currently undergoing dynamic development. These technologies have a wide spectrum of application which ranges from forming the enclosure for a device (also as a decorative element) and replacing worn
or damaged machine components, to structural testing of upper and lower limb prostheses [1–3]. Nowadays rapid prototyping is the most commonly used in biomedicine and dentistry [4,5]. Rapid prototyping is also used to manufacture parts of aircraft engines by the GE Aviation. The Department of Turbine Dynamics and Diagnostics at the Institute of Fluid-Flow Machinery of the Polish Academy of Sciences (IFFM PAS) is fully engaged in the research and development works concerning the prototypical fluid-flow machinery, including sophisticated microturbines. After preliminary analysis of manufacturing costs and duration of prototypical turbines by means of conventional methods, it was decided to employ a 3D print technology called MJP (multijet printing), which was not previously used for production of microturbine components. This is an inkjet 3D printing process used in the case of printouts that require highly precise production and complicated shapes as well as serial production. The multijet printing is one of the top ranked technologies for this type of production. During the printing process, either photocurable plastic resin or casting wax material (supporting material) is deposited layer by layer. Printing materials are available in different varieties, each one being defined by its specific physical and chemical characteristics. The choice of this technology arises from a need to provide high repeatability of details for printed elements. The ProJet 3500 HDMax printer – which uses multijet printing technology – was employed as a tool to manufacture the components to be examined. It can works in four modes: high definition (HD), high speed (HS), ultra high definition (UHD) and extreme high definition (XHD). In each mode the print speed and are very accuracy different. The differences between the four operating modes can be summarized as follows:

- HD – single layer thickness 32 µm, printing speed 3 mm/h;
- HS – single layer thickness 32 µm, printing speed 4 mm/h;
- UHD – single layer thickness 29 µm, printing speed 2 mm/h;
- XHD – single layer thickness 16 µm, printing speed 1.5 mm/h.

Each operating mode allows to fully utilize the workspace with the dimensions of 298 mm × 185 mm × 203 mm, which provides printing of machine components that are slightly larger than microturbines’ components. Cartridges installed in the printer contain approximately 2 kg of polymeric resin (building material) or 2 kg of the supporting material. The printing device described is the basic piece of equipment of the Rapid Prototyping Laboratory at the IFFM PAS (shown in Fig. 1.) The major advantage of printing machine components using the MJP technology is the excellent precision of representation of complex surfaces and edges. In the case of XHD mode, the printhead deposits printing material layer by layer (with layer thicknesses as low as 16 µm), until the finished product is
A concept of a test stand for the investigation... built. Thanks to the positioning of material depositing nozzles over the entire printhead width and the use of a supporting material (wax), this 3D printing machine is able to cope with elements of any geometry, even the most complicated one. Ultraviolet-curable liquid photopolymer (plastic resin) is used as a building material. This material and its UV-curing method are currently tested for adhesion strength and rheology [6,7]. One of the two available kinds of this material has been tested to estimate its tensile strength in two directions [8]. Depending on the printout direction the tensile strength was 47 MPa and 41 MPa. The specification annexed to the material states that the maximum operating temperature is 56°C. It is supposed that the tensile strength of the second stronger kind of the photopolymer will be around 65 MPa, where its maximum operating temperature is 88°C according to the technical specification [9]. The above information on the building materials provides assurance that the elements printed by means of the rapid prototyping will be able to perform their function during research on prototypical turbines.
2 Research concept

The experimental research on elements manufactured by the multijet printing technology has been undertaken as part of the work on the development of rapid prototyping techniques and the verification of a selected technology. During the first step, the correct performing of the elements created by means of the adopted printing technology will be tested. If the preliminary tests will pass successfully, the prototypes made of the photopolymer will be tested under increasingly more demanding operating conditions. The subject of this research will be an automotive turbocharger made by Renault. The turbocharger shown in Fig. 2a has been dismantled, and all individual structural components were measured. Based on this technical specifications of a three-dimensional turbocharger model was created, using the Autodesk Inventor Professional 2015 software, presented in Fig. 2b.

![Turbocharger used in the studies: (a) – photograph, (b) – 3D model.](image)

The first step of further research will be to determine the basic operating parameters of the original turbocharger. Then the two main turbocharger components, namely the compressor wheel and the turbine wheel, will be printed using the MJP technology. Both wheels will be fixed on a steel shaft, which will be mounted inside the turbocharger body. The 3D models of these components were obtained by means of a structure sensor 3D scanner and a 3D laser scanner. Figure 3 shows a three-dimensional model of the turbocharger’s turbine wheel.
The research will be performed to demonstrate whether the elements created by means of the rapid prototyping techniques can be used with similar effect on the target machine (i.e., the basic operating parameters of the turbocharger have similar values to those obtained during the operation with original parts). The main emphasis will be placed on the verification of flow characteristics and overall performance of the machine containing components made of a plastic material. The dynamic performance tests of the two rotating systems created using different methods will be also carried out. Ultimately, rapid prototyping processes are to be applied during the first testing phase of new constructional solutions for fluid-flow machinery such as microturbines, positive displacement expanders or compressors. The ability to manufacture elements with complex shapes (e.g., turbine wheels equipped with vanes of varying cross-sections) in a fast and cost-efficient manner allows to significantly speed up some of the time-consuming processes such as construction optimization and numerical model verification. Manufacturing of the most complex turbomachines’ components by means of 3D printers situated at in-house laboratories allows production of small elements to be carried out independently of specialized external providers (i.e., companies that offer machining services), moreover, the time needed for researching designing, and building is considerably reduced, thus enabling quick implementations. The photograph of a 3D printing device that will be used for production of machine components required for the experimental research described above is shown in Fig. 1.

3 Test rig

The test rig has been designed to conduct research on the turbocharger. Its schematic diagram is shown in Fig. 4. The test rig’s frame is drafted in such
a way as to enable simple and trouble-free disassembling of the turbocharger (e.g., when replacing its rotor or the complete device).

![Figure 4: Schematic diagram of the test rig for research and development of turbines: 1 – turbocharger, 2 – compressor, 3 – buffer tank, 4 – oil tank, 5 – oil pump, V – throttling valve.](image)

As one can see in Fig. 1, the turbocharger is supplied with compressed air. For this a compressor with a maximum working pressure of 1 MPa was used. A buffer tank acting as a heat exchanger allows to reduce the air temperature directly before the compressor inlet. Forced air circulation takes place along the circuit denoted by a dotted line. A pump circulates oil in a closed circuit. The oil in the storage tank before the pump provides correct lubrication of the turbocharger’s bearings, in order to prevent damage to the machine due to dry-running bearings. The turbocharger has an oil outlet, adjacent to the bearing supply system, which allows maintaining continuity of lubrication. The oil circulation is indicated by a broken line in the schematic diagram shown in Fig. 4. An indicative arrangement of the measuring sensors is also illustrated in this figure. All elements of
the test rig are installed on a rigid steel baseplate equipped in rubber feet. The turbocharger will be mounted on slide rails allowing to locate it in any position, and in the future to change a research facility.

To be able to determine the operating characteristics of turbocharger the measurement system will comprise 9 measuring points to automatically measure selected parameters such as temperature, pressure, rotational speed and air flow rate in the circulation of the installation. In the later stages of the research, the vibrodiagnostics of the running machine will be performed using acceleration sensors installed at the measuring points. This is to investigate the proper operation of the machine. The presence of a flow regulating valve (the throttling valve) installed at the turbine outlet allows varying the rotational speed. Temperature and pressure measurements will be implemented by K-type thermocouples and pressure transducers (providing absolute pressure range from 0 to 1.6 MPa), respectively. The air flow rate at the turbocharger outlet will be measured by means of a Coriolis flow meter. Each measuring point is connected to a data acquisition module (National Instrument model NI-6210). The schematic diagram of the measuring system is shown in Fig. 5.

![Schematic diagram of the measuring system for the test rig used for testing turbines.](image)

As indicated in the schematic diagram, the sensors installed on the test rig are connected to the measurement module. Thermocouple signals processed by programmable temperature transducers (TMD-20 model, manufactured by Czaki...
Thermo-Product) – denoted by the symbol ‘tt’ – are being sent to analogue inputs (AI) of the measurement module. The pressure transducers and flow meter output current signals, so it is necessary to connect them to the measurement module by means of the resistor that has a resistance of 240 Ω. The rotational speed sensor is connected to the impulse counter, which is one of the digital inputs (DI) available on NI-6210 measurement module. The measurement module is connected to a laptop via the USB cable. It is planned to create a computer application for integrating the entire measurement process, i.e., acquisition of data from the sensors, their processing and storing as well as remote monitoring of the selected operating parameters of the turbine (online access).

The test rig is versatile enough to enable running tests on other fluid-flow machinery with similar dimensions and power capacities, such as vapour microturbines or scroll expanders intended for operation in ORC (organic Rankine cycle) systems. An additional strong point of this test rig is also the possibility of integrating it with other test rigs used in the Department of Turbine Dynamics and Diagnostics (e.g., test rigs for carrying out research on ORC cycles, heat exchangers and pumps).

4 Summary and conclusions

The paper presents the concept of research aiming to develop prototypical fluid-flow machinery incorporating parts manufactured by means of rapid prototyping techniques. The test rig equipped with a measurement system has been designed and built. The turbocharger has been prepared for experimental tests and is ready for replacement of its elements with 3D printed elements. In recent years, the 3D printing technology described has contributed significantly to the development of many fields of science. Following the research carried out on the building materials and also on the elements printed using this technology it can be presumed that the results of the planned studies will facilitate and speed up the entire development and production process of prototypical machinery, particularly in the light of high accuracy of printouts and their good tensile strength. The manufacturing of elements needed for particular tests has permitted an accurate study of the multijet printing technology and its inbuilt benefits and drawbacks. An additional asset of this technology is quick printing of small elements of any shape which can be used as interchangeable parts in various machines (e.g., in microturbines).

Received in July 2016

ISSN 0079-3205 Transactions IFFM 133(2016) 3-11
A concept of a test stand for the investigation...

References


[9] Data sheet for the material called VisiJet m3 Crystal:  
http://www.3dsystems.com/ru/materials/visijet-m3-crystal
Mariusz Banaszkiewicz\textsuperscript{a,b,*}, Krzysztof Dominiczak\textsuperscript{a,b}

A methodology for online rotor stress monitoring using equivalent Green’s function and steam temperature model

\textsuperscript{a} GE Power sp. z o.o., Stoczniova 2, 82-300 Elblag, Poland
\textsuperscript{b} Institute of Fluid Flow Machinery, Polish Academy of Sciences, 80-231 Gdańsk, Fiszera 14, Poland

Abstract
The requirement of high operational flexibility of utility power plants creates a need of using online systems for monitoring and control of damage of critical components, e.g., steam turbine rotors. Such systems make use of different measurements and mathematical models enabling calculation of thermal stresses and their continuous control. The paper presents key elements of the proposed system and discusses their use from the point of view of thermodynamics and heat transfer. Thermodynamic relationships, well proven in design calculations, were applied to calculate online the steam temperature at critical locations using standard turbine measurements as input signals. The model predictions were compared with operational data from a real power plant during a warm start-up and show reasonably good accuracy. The effect of variable heat transfer coefficient and material properties on thermal stresses was investigated numerically by finite element method (FEM) on a cylinder model, and a concept of equivalent Green’s function was introduced to account for this variability in thermal stress model based on Duhamel’s integral. This approach was shown to produce accurate results for more complicated geometries by comparing thermal stresses at rotor blade groove computed using FEM and Duhamel’s integral.

Keywords: Steam turbine; Thermal stress; Green’s functions; Duhamel’s integral

*Corresponding Author. Email adress: mariusz.banaszkiewicz@ge.com

ISSN 0079-3205 Transactions IFFM 133(2016) 13–37
Nomenclature

\(c_0\) – isentropic velocity, m/s
\(c_p\) – specific heat, J/kg K
\(E\) – Young modulus, MPa
\(G\) – Green’s function for stress, MPa/°C
\(h\) – enthalpy, kJ/kg
\(m\) – mass flow rate, kg/s
\(n\) – rotational speed, rev/min
\(p\) – pressure, MPa
\(r\) – radial coordinate, m
\(T\) – temperature, °C
\(t\) – time, s
\(u\) – circumferential velocity, m/s
\(v\) – specific volume, m³/kg
\(X\) – Green’s function for temperature, °C/°C
\(y\) – valve opening, %

Greek symbols

\(\alpha\) – heat transfer coefficient, W/m²K
\(\beta\) – thermal expansion coefficient, 1/K
\(\eta\) – efficiency
\(\lambda\) – thermal conductivity, W/m K
\(\nu\) – Poisson number
\(\sigma\) – stress, MPa

Subscripts and superscripts

\(CS\) – control stage
\(N\) – nozzle
\(eq\) – equivalent
\(T\) – throat
\(m\) – metal
\(perm\) – permissible
\(s\) – steam
\(surf\) – surface
\(0\) – upstream stop valves
\(1\) – downstream stop valves
\(2\) – downstream control valves
\(V_i\) – valve \((i = 1, 2, 3, 4)\)

1 Introduction

Modern energy markets put a requirement of high operational flexibility of power plant units [1,2]. Due to this, new designed units, including steam turbines, have to fulfill a series of requirements regarding the number and rate of change of specified operation states. With regard to steam turbines, the requirements concern
the number and time of start-ups from different thermal conditions and the rate of load change. The above operation states generate elevated loads and stresses in turbine components leading to material damage due to thermomechanical fatigue [3].

The problem of thermal stress monitoring and control in steam turbines was considered already at mid-sixties [4]. In the first thermal stress control systems, steam turbine rotors were supervised using start-up probes which were thermo-physical models of turbine rotors [5–7]. Thermal stresses were calculated based on measured temperature difference between the probe surface and its integral-averaged temperature. A better accuracy of stress calculation was achieved by replacing the measured average temperature with a mathematical model [8–10]. Further development of thermal stress supervision systems consisted in complete resignation from the temperature probe and modeling thermal stresses based on the standard process measurements of power units, which were a basis for calculating the characteristic temperature difference surface-mean [11]. Rusin et al. [12] proposed a new method of thermal stress modeling in turbine rotors employing Green’s functions and Duhamel’s integral, and steam temperature measurement at critical location. The Green’s functions and Duhamel’s integral have been for many years used for fatigue life monitoring of nuclear plants equipment by such companies like EPRI [13–15], GE [16] or EDF [17–18]. Such an approach has also been adopted for monitoring of power boilers operation by Taler et al. [19], and as shown by Lee et al. [20] can also be used for calculating stress intensity factors at transient thermal loads. Recent developments in the field are focused on the use of artificial neural networks for predicting boiler wall temperature [21] and turbine thermal stresses [22–23].

The major issue in using Green’s function and Duhamel’s integral method in modeling transient thermal stresses in steam turbines components is time-dependence of material physical properties and heat transfer coefficients affecting proper evaluation of Green’s functions. There are known approaches assuming determination of the influence functions at constant values of these quantities [24]. The inclusion of temperature dependent physical properties proposed by Koo et al. [25] relies on determining the weight functions for steady-state and transient operating conditions. A more important, in most cases, variation of heat transfer coefficient can be taken into account by calculating the surface temperature using a reduced heat transfer model and employing Green’s function to calculate a stress response to the step change of metal surface temperature [26]. However, numerical solution of a multidimensional heat transfer model is complicated and time-consuming, and due to this cannot be used in online calculations. A full in-
clusion of time variability of the physical properties and heat transfer coefficients proposed by Zhang et al. [27] relies on the solution of nonlinear heat conduction problem by using artificial parameter method and superposition rule and replacing the time-dependent heat transfer coefficient with a constant value together with a modified fluid temperature. The effectiveness of the method has been proved by an example of a three-dimensional model of a pressure vessel of nuclear reactor.

The present paper deals with the problems of modeling steam temperature and thermal stresses for online supervision of low-cycle fatigue life. A control system is proposed and its main elements are:

- thermodynamic model enabling fast calculation of steam temperature at critical location,
- thermal stress model for online calculation of rotor thermal stresses.

The models have been validated by comparing the results of numerical calculations with real turbine measurements and more accurate 3D simulations.

2 Thermal stress control system

Thermal stress control systems of steam turbine rotors are currently a commonly used measure of protecting the design fatigue life of high- and intermediate pressure turbine rotors. These systems are very important elements of turbine control and protection systems and operate in closed-loop control. A schematic diagram of turbine control system including a module responsible for thermal stress control is shown in Fig. 1. Based on turbine measurements (e.g., temperature, pressure, speed), the stress controller calculates stresses and load fraction and outputs to the turbine controller a signal of stress margin which reduces the set values of speed or load rates. The turbine controller positions, with the help of actuator, the control valve head controlling in this way the steam flow rate and temperature upstream the turbine blading. These two parameters have impact on the rotor temperature and the resulting thermal stresses.

The thermal stress controller calculates stresses at rotor critical locations on the basis of measurement signals and compares them with the permissible stresses. On the basis of these two stresses, a load fraction, $LF$, is calculated using the formula

$$LF = \frac{\sigma_{eq}}{\sigma_{perm}}.$$  

The equivalent stress, $\sigma_{eq}$, is computed based on the measured temperature, pressure, rotational speed and using a mathematical model of temperature (if it is not
Figure 1: Schematic diagram of turbine control system with thermal stress control.

measured directly) and thermal stresses. The permissible stress, $\sigma_{\text{perm}}$, is derived from the material fatigue characteristics knowing the required number of cycles and assuming a material model.

Thus, the key elements of the considered thermal stress control system are:

- thermodynamic model allowing for steam temperature calculation at critical regions,
- thermal stress model for online stress calculation,
- relationship between elastic stress and total strain allowing for the calculation of permissible stresses for a given number of start-ups.

3 Thermodynamic model

3.1 Model formulation

In order to calculate thermal stresses at rotor critical locations, it is necessary to know the steam temperature at these locations, which is an input signal to the stress calculation algorithm. Steam temperature can be evaluated in two ways, namely:

- direct measurement of temperature at critical location using a thermocouple,
- calculation of steam temperature at critical location based on measurements at different regions and using a mathematical model.

Direct temperature measurement requires installing a measurement sensor at critical location, which is not always possible due to design restrictions. Moreover, a problem of dynamic temperature measurement error occurs due to the thermal resistance of thermocouple and thermowell, which has to be robust for mechanical integrity reasons [28]. Also heat flux variations during transients and
high-frequency unsteadiness occurring at every rotor passage with varying amplitudes originating from complex blade row interactions make the heat transfer time-dependent and accurate measurements become essential for correct thermal assessment of steam turbines [29].

A more universal way is steam temperature calculation at critical location based on a measurement at a different suitable point. For this purpose, a mathematical model is required describing steam thermodynamic process from the measurement point to the critical location. In online monitoring and control systems, the thermodynamic model should be fast, accurate and reliable, to ensure the capability of its implementation in the turbine control system and the required quality of computations. For this reason, only zero- or one-dimensional models with a minimum number of iteration loops can be considered. Modeling steam temperature is the only way of its evaluation in the areas not accessible for measurement or in turbines not equipped with the required temperature measurement.

As experience shows, the most critical region from the point of view of thermal stress is the control stage. That is why for modeling purposes we assume a steam admission system with the control stage, as shown in Fig. 2, which is typical for nozzle control.

![Figure 2: Steam admission system with nozzle control.](image)

For online steam temperature calculation it is proposed to use a mathematical model employing the measurement of current opening of the control valves. The model enables online calculation of two quantities which are very important from the viewpoint of thermal loads and stresses:
• steam temperature in control stage chamber and temperatures downstream each control valve and nozzle sector for current openings,
• mass flow rates through each valve and total mass flow through the turbine.

For the considered steam admission system consisting of two stop valves and four control valves, the following measurements are used as input signals to the steam temperature model:
• live steam temperature – $T_0$,
• live steam pressure – $p_0$,
• steam pressure downstream control stage – $p_{CS}$,
• control valves openings – $y_{V1}$, $y_{V2}$, $y_{V3}$ and $y_{V4}$,
• turboset rotational speed $n$.

The steam temperature model is based on the relations known from steam turbines theory and used in design calculations. In the proposed mathematical model these relationships are given in a consistent way with necessary simplifications and additional elements resulting from the specificity of online calculations carried out in the entire range of turbine operating conditions. In the first step, steam pressures downstream control valves and mass flow rates through each nozzle sector are calculated. One control valve cooperates in series with a nozzle sector. For such a system, the condition of equal mass flow rates can be written

$$\dot{m}_2 = \dot{m}_D.$$  \hspace{1cm} (2)

The valve mass flow rate $\dot{m}_2$ is given by the relationship [30]:

$$\dot{m}_2 = 0.667 q A_T \sqrt{\frac{p_1}{v_1}},$$  \hspace{1cm} (3)

where $q$ is a relative mass flow rate determined from the valve characteristics in a function of opening and pressure ratio, $A_T$ – section area of valve throat, while $p_1$ and $v_1$ denote steam pressure and specific volume upstream the valve. The nozzle sector mass flow rate $\dot{m}_D$ is described by equation [31]

$$\dot{m}_N = 1.42 p_2 (p_0 v_0)^{-0.5} A_N \sqrt{-0.09 + 1.09 p_{CS}/p_2 - (p_{CS}/p_2)^2},$$  \hspace{1cm} (4)

where $A_N$ is exit section area of nozzle sector.

A nonlinear equation with unknown $p_2$ is obtained from Eqs. (2)–(4) and is solved iteratively using the bisection method. Knowing the pressure $p_2$, we can calculate the mass flow rate from Eq. (3) or (4). Performing similar calculations for each valve, we find the appropriate pressures and mass flow rates for all valves and nozzle sectors.
In the control valves, isenthalpic process of steam throttling from upstream pressure, \( p_1 \), to downstream pressure, \( p_2 \), takes place. Assuming additionally that the enthalpy upstream the control valve is equal to the steam enthalpy upstream the stop valve, we can express steam temperature downstream the control valves as a function of steam temperature and pressures directly measured on the turbine:

\[
T_2 = f(p_0, T_0, p_2).
\] (5)

Knowing the steam temperature and pressure downstream the control valves and steam pressure downstream the control stage, we can calculate steam enthalpy using the formula known from the theory of steam turbines [31]

\[
h_{CS} = \frac{h_{CS}^1 \dot{m}_2^V_1 + h_{CS}^2 \dot{m}_2^V_2 + h_{CS}^3 \dot{m}_2^V_3 + h_{CS}^4 \dot{m}_2^V_4}{\dot{m}_2^V_1 + \dot{m}_2^V_2 + \dot{m}_2^V_3 + \dot{m}_2^V_4},
\] (6)

where: \( h_{CS}^1, h_{CS}^2, h_{CS}^3, h_{CS}^4 \) – steam enthalpies behind each sector of the control stage.

These enthalpies depend on the circumferential efficiency, \( \eta_0 \), of each sector which depends on the current opening of the control valve feeding this sector. In this way, in the proposed calculation model a relationship between the control stage efficiency and current valve openings is considered. It provides a capability to take into account in online calculations the influence of valve timing on steam temperature in the control stage.

The circumferential efficiency is a function of velocity ratio \( u/c_0 \) and for online calculations, an individual efficiency characteristics prepared on the basis of control stage calculations using design tool is used. An example of stage efficiency characteristics is shown in Fig. 3. The results of calculations performed with the design code are presented by squares, while the continuous line represents an approximation with the function

\[
\eta_o = A + B \left( \frac{u}{c_0} \right) + C \left( \frac{u}{c_0} \right) \ln \left( \frac{u}{c_0} \right) + D \left( \frac{u}{c_0} \right)^2 + E \left( \frac{u}{c_0} \right)^2 \ln \left( \frac{u}{c_0} \right) + F \left( \frac{u}{c_0} \right)^{2.5} + G \exp \left( -\frac{u}{c_0} \right)
\] (7)

in which \( A, B, C \ldots G \) are the coefficients of the approximation function. In this figure the values of stage efficiency are related to its maximum value.

The calculated steam enthalpy, \( h_{CS} \), and measured pressure in the control stage, \( p_{CS} \), allow to determine steam temperature from a thermodynamic function:

\[
T_{CS} = f(p_{CS}, h_{CS}).
\] (8)
3.2 Comparison with measurements

To validate the steam temperature model, measurements were carried out on a 370 MW steam turbine. A steam temperature measurement consisting of a thermowell with inserted thermocouple was installed in the control stage chamber. The temperature measurement was taken continuously and information about steam temperature was available at any instant of turbine operation. The measured temperatures were compared with the results of calculations performed using the mathematical model at various operating conditions (start-ups, shut-downs, load changes, etc.). An example of measurements and calculations is shown in Fig. 4. It presents the measured values of live steam temperature (dash line), control stage temperature (long dash line) and casing temperature (solid line) taken in the vicinity of steam temperature measurement. For the purpose of comparison also calculated steam temperature in the control stage chamber (dot line) is shown, which as it is seen from the plot, corresponds very well with the measured temperature in the entire time period. For the presented start-up the maximum temperature deviation is 17°C, and hence the maximum relative error of calculations does not exceed 5%. Typical measurement error for the industrial temperature measurement circuit of the type used here is ±2.5°C. Based on this it can be said that a good accuracy was achieved with this simple online model. The model calculates online steam enthalpy in the control stage and its mass flow rate, thus enabling determination of the control stage power at on-load operation.
Figure 4: Measured and calculated temperature variations during warm start-up.

Figure 5: Mass flow rate, power and enthalpy drop variations during warm start-up.
Figure 5 presents a variation of the mass flow rate, turbine and control stage power and its enthalpy drop calculated for the analyzed warm start-up. Enthalpy drop was computed using both the calculated and measured steam temperature in the control stage. Both predictions agree very well with each other, thus the control stage power calculated based on the enthalpy drop during turbine loading can be considered as realistic.

4 Thermal stress modeling

4.1 Green’s function method

It is assumed that the thermal stress model used in online calculations is based on Green’s function and Duhamel’s integral method. This method allows for fast calculation of thermal stresses at supervised areas for any changes of fluid temperature causing heating-up or cooling-down of an element. In steam turbine rotors a reliable and continuous measurement of the rotor surface temperature has not been possible so far, thus for online monitoring purposes the steam temperature measurement is used as a leading signal. Consequently, in heat transfer model Fourier’s boundary condition is employed [32]:

$$\lambda (r) \frac{\partial T (r, t)}{\partial n} = -\alpha (r) (T_s (t) - T (r, t)) , \; r = R ,$$

(9)

where $\lambda (r)$ is metal thermal conductivity, $T (r, t)$ is surface temperature, $T_s (t)$ denotes steam temperature, $n$ is normal direction, $\alpha (r)$ – heat transfer coefficient, and $R$ is the outer radius.

Assuming that steam temperature is described by Heaviside’s function $H(t)$, the boundary condition Eq. (9) can be written in the form [25]

$$\lambda (r) \frac{\partial X (r, t)}{\partial n} = -\alpha (r) (H (t) - X (r, t)) ,$$

(10)

where $X (r, t)$ is a Green’s function for temperature.

Using the Green’s function $X (r, t)$ we can calculate the metal temperature for arbitrary variation of steam temperature employing Duhamel’s theorem [32]:

$$T (r, t) = \int_0^t X (r, t - \tau) \frac{\partial T_s (\tau)}{\partial \tau} d\tau .$$

(11)

Considering elastic stresses only it can be assumed, according to the thermo-elasticity theory, that stress distribution in an elastic body is a unique function
of temperature distribution, and in this connection the thermal stresses can be calculated using Duhamel’s integral as

$$\sigma_{ij}(r,t) = \int_0^t G_{ij}(r,t-\tau) \frac{\partial T_s(\tau)}{\partial \tau} d\tau ,$$  \hspace{1cm} (12)

where $G_{ij}(r, t)$ is a Green’s function for thermal stress component $ij$.

For simple geometries (plate, cylinder, sphere) Green’s functions for temperature and stress can be determined analytically by solving one-dimensional transient heat conduction problem. In case of more complicated shapes it is necessary to use numerical methods, e.g., finite element method. Also in our case numerical integration of Eq. (12) is necessary, it is thus transformed from continuous into discrete form

$$\sigma_{ij}(r,t) = G_{s,ij}(r) T_s(t) + \sum_{t-t_d}^t G_{t,ij}(r,t-\tau) \Delta T_s(\tau) ,$$  \hspace{1cm} (13)

where $t_d$ is a cut-off time, $G_{s,ij}(r)$ is a value of the influence function in steady state, while $G_{t,ij}(r)$ is a transient part of the influence function.

Green’s functions are determined individually for each supervised region, and in case of thermal stresses these functions must be determined for each component of the stress tensor $\sigma_{ij}$.

An example of Green’s function for temperature and stress components at step increase of fluid temperature by 1°C heating the external surface of a long cylinder is shown in Fig. 6. The calculations were performed with constant heat transfer coefficient $\alpha = 1000$ W/m²/K and constant material physical properties evaluated at temperature 350°C. The temperature (Fig. 6a) and stress (Fig. 6b) responses are presented for the cylinder axis and its outer surface heated by steam.

The Green’s function for surface temperature, $X_{surf}$, initially rapidly increases and reaches nearly 1 after approx. $10^4$ s. The temperature at cylinder axis, $X_{cent}$, shows some inercy and a delay time of temperature response equal $500$ s is seen. After this time the temperature increases at approximately constant rate and after about $4000$ s the rate of variation visibly starts to decrease. The cylinder surface response is typical for surface-type sensor, while Green’s function at cylinder axis is a response of middle-type sensor. The stress response at both areas is a result of temperature variations: axial and circumferential stresses at cylinder surface are equal to each other and rapidly grow reaching a minimum (compressive stress) and subsequently tend slowly to zero; the stresses at cylinder axis grow more slowly, their maxima are lower than at surface (axial stress is two times higher than circumferential stress) and are tensile stresses (sign +).
Figure 6: Green’s functions for temperature $X$ (a) and stress $G$ (b) for externally heated cylinder.
All stress components within the cylinder tend to zero and a stress-free state is reached after approx. $1.4 \times 10^4$ s, which corresponds to the instant when temperature within the whole cylinder is uniform.

4.2 Green’s function for temperature dependent material properties and time dependent heat transfer coefficient

The presented in the previous section Green’s functions for temperature $X$ and thermal stresses $G$ were determined with constant physical properties and constant heat transfer coefficient $\alpha$. The physical properties of steel influencing temperature (specific heat, $c_p$, thermal conductivity, $\lambda$) and stress (Young’s modulus, $E$, Poisson’s number, $\nu$, thermal expansion coefficient, $\beta$) distribution strongly depend on temperature, while heat transfer coefficient $\alpha$ varies with temperature and flow velocity (via the Reynolds number Re). The dependence of these coefficients on temperature and velocity for typical operating conditions of 1CrMoV rotor is shown in Figs. 7 and 8. The heat transfer coefficient in labyrinth seal was calculated using Kapinos et al. correlation [33]:

$$\frac{\alpha \cdot d}{\lambda} = 1.125 \text{Re}^{0.65} \left( \frac{\delta}{h} \right)^{0.35} \left( \frac{b}{s} \right)^{0.1} \left( \frac{h}{s} \right)^{0.32},$$

(14)

where $d$ is the shaft diameter, $\delta$ is seal radial clearance, $h$ is seal channel height, $s$ is the distance between teeth, and $b$ is the teeth width.

Among the considered physical properties the most varying is the specific heat which in the analysed temperature range increases by more than 50%. The largest drop is observed for the Young’s modulus which decreases by more than 30%. Heat transfer coefficient depends most on the flow velocity. From the variation of labyrinth seal, $a$, shown in Fig. 8 it is seen that its value can change even by an order of magnitude. The influence of temperature is smaller and more distinct in the range of higher velocities.

A consequence of the above shown variation of physical properties and heat transfer coefficient is a dependence of Green’s function, $G$, on these parameters. The dependence is illustrated in Fig. 9 showing Green’s function variations for axial stress at cylinder surface calculated by finite element method at different temperatures and heat transfer coefficients. Each individual curve corresponds to stress variation determined at constant temperature, $T = \text{const}$, and constant heat transfer coefficient, $\alpha = \text{const}$. The variation range of these parameters was assumed so as to cover typical operating conditions of 1CrMoV rotors. As it is seen from the plots, the largest effect on the Green’s function $G$ is observed for the heat transfer coefficient, which significantly affects the stress maxima, time of
A methodology for online rotor stress monitoring using equivalent...

Figure 7: Dependence of typical physical properties of 1CrMoV rotor steel on temperature.

Figure 8: Dependence of heat transfer coefficient on temperature and velocity, accor. Eq.(14).
their occurrence and decay rate. The influence of temperature variation is much smaller and most visible from the instant of maximum stress to the moment of their vanishing.

Figure 9: Green’s function, $G$, for axial stress at cylinder surface at various temperatures and heat transfer coefficients.

As a result of so high sensitivity of Green’s function, $G$, to the conditions at which it was determined, a scatter in stress variation calculated using Eq. (13) with different Green’s function is observed. Figure 10 presents axial stress variation with time for cylinder surface calculated using the functions presented in Fig. 9. The calculations were performed for the following conditions:

- initial conditions: metal temperature $T_m(0) = 100^\circ$C, steam temperature $T_s(0) = 200^\circ$C
- boundary condition: steam temperature rate $dT_s/dt = 2^\circ$C/min

For comparison, also the axial stress variation calculated in three dimensional heat transfer model with the same initial and boundary conditions but with heat transfer coefficient, $a$, linearly changing with time from 100 to 10000 W/m$^2$/K is shown. The heat transfer coefficient was varied until $t = 9000$ s and from this instant remained constant.
A methodology for online rotor stress monitoring using equivalent...

As it is seen from the presented stress variations, the assumption of constant physical properties or constant heat transfer coefficient leads to large errors in calculating thermal stresses at large variations of these parameters. In order to improve the accuracy of thermal stress calculations using Duhamel’s integral, Eq. (13), it was proposed to take into account the influence of temperature and heat transfer coefficient on the shape of Green’s functions. It was realized by assuming variation in time of the heat transfer coefficient, $\alpha$, and calculating the equivalent Green’s function, $G_{eq}$, by imposing steam temperature step change from a minimum to a maximum temperature of the element. By imposing such a step change we took into account the variation of material physical properties in the predicted range of operating temperatures. Green’s functions for cylinder axial and circumferential stresses calculated in such a way are shown in Fig. 11. The functions were evaluated numerically by assuming, as before, linear variation of the heat transfer coefficient, $\alpha$, from 100 W/m$^2$/K to 10000 W/m$^2$/K in time $t = 9000$ s and step temperature change from 100°C to 500°C. The physical properties of steel were assumed as temperature-dependent according to the curves shown in Fig. 7. The Green’s functions calculated in such a way are not qualitatively different compared with those determined at temperature $T = 350$°C and heat transfer coefficient $\alpha = 10000$ W/m$^2$/K. Due to different parameters at

Figure 10: Axial stresses at cylinder surface during heating-up with a rate of 2°C/min for Green’s functions determined at different conditions.
which both sets of functions were determined, small differences in stress maxima and times of their occurrence are seen.

Using the Green’s functions shown in Fig. 11, thermal stress calculations were performed for cylinder heating-up at the same conditions as in the previous example and the results are presented in Fig. 12. It is clearly seen that not only qualitative but also very good quantitative agreement between the predictions of Green’s function method and nonlinear three dimensional model was achieved.

4.3 Model verification for turbine rotor

The proposed concept of equivalent Green’s function, $G_{eq}$, was validated on an example of a high pressure steam turbine rotor. A cold start-up lasting 240 min was analyzed. The selected example is characterized by a much more complicated geometry and high variability and nonuniformity of thermal boundary conditions resulting in generation of highly non-uniform temperature and stress fields within the rotor. A geometric model of the rotor with a zoomed first blade groove being a critical location is shown in Fig. 13. A variation in time of the turbine mass flow rate, steam temperature and heat transfer coefficient is shown in Fig. 14. All
quantities are related to its values at nominal conditions.

Temperature and stress calculations were performed by means of a 3D model of heat and momentum transport using the rotor axisymmetry. In the simulations, temperature-dependent material physical properties and time- and space-dependent (in axial direction) heat transfer coefficients on the rotor outer surfaces were used. The rotor blades were not included in the model, and internal surfaces of the blade grooves were assumed as adiabatic. The stress calculations were performed using linear elastic material model. The rotor initial temperature was 100 °C.
Figure 14: Variation in time of the mass flow rate, steam temperature and heat transfer coefficient.

Figure 15: Temperature (a) and equivalent Huber-Mises-Hencky stress (b) distribution in first blade groove after 1500 s from the beginning of start-up.

Based on the variation in time of the rotor stress field it was found that the largest stresses are generated at the bottom of the first blade groove. Figure 15 presents the temperature (a) and equivalent Huber-Mises-Hencky stress fields in the region of blade groove after 1500 s from the beginning of start-up. At this instant, a global stress maximum in the rotor during the whole start-up occurred. As it is seen from the temperature distribution, a high radial gradient of tempera-

ISSN 0079-3205 Transactions IFFM 133(2016) 13–37
ture in the groove occurred and temperature difference between the rotor surface and blade groove bottom reached 30% of the surface temperature. In the bottom left corner of the groove, very high stresses were generated at this instant and they were highly concentrated at his point. The stress field away from this region was more uniform, and the stress values much lower than in the concentration area – on the level of 30% of maximum stress.

Next, the equivalent Green’s functions for the point where stress maximum occurred were determined. The form of these functions for individual stress components is shown in Fig. 16. It is clearly seen that all Green’s functions are negative, which means that with temperature increase all components of the thermal stress tensor will be negative, so at the groove bottom a state of multiaxial compression will persist.

![Figure 16: Equivalent Green’s functions for stresses at blade groove bottom (red point).](image)

Employing the equivalent Green’s functions presented in Fig. 16, stress calculations were performed for the considered start-up using Duhamel’s integral, Eq. (13), and the results were compared with the predictions of 3D model. Comparison of the variation of radial and circumferential stress components as well as Huber-Mises-Hencky stress calculated by two methods is shown in Fig. 17. It is seen from the stress curves that a very good accuracy was achieved both for the stress components and the equivalent stress. The stress curves coincide with each
Figure 17: Variation of the groove stresses during cold start-up calculated using equivalent Green’s functions (lines) and 3D model (points).

other both during stress increase and decrease; also the local stress maxima and minima occurring after 1500 and 10500 s correspond very well.

The achieved accuracy of blade groove stress predictions for this particular geometry and operating parameters provides a basis for expecting a general capability of equivalent Green’s functions and Duhamel’s integral to accurately predict thermal stress evolution in steam turbine rotors. Further investigations would be required in order to confirm a potential general applicability of this approach to different designs and operating conditions. The use of equivalent Green’s functions in Duhamel’s integral which were evaluated by solving a problem with nonlinear boundary conditions and variable physical properties can provide an accurate solution for thermal stresses, despite the fact that the integral was derived for linear problems (i.e., constant heat transfer coefficient and physical properties).

For online monitoring and control of thermal stresses in steam turbine rotors during start-ups, it is necessary to use Green’s functions determined for thermo-flow conditions typical for cold, warm and hot starts and use in the stress model different Green’s functions, depending on the start-up type.
5 Conclusions

The paper presented a methodology of online thermal stress calculation and control in steam turbine rotors. The main elements of the proposed methodology are: thermodynamic model allowing for calculating steam temperature at critical locations and rotor thermal stress model.

For calculating thermal stresses, steam temperature is used as a leading input signal which is calculated on the basis of thermodynamic model of the turbine inlet part. In such a case, it is not necessary to install a steam temperature measurement as the model is based on the turbo-set standard measurements. A model of the control stage with four nozzle sectors capable of calculating transient steam temperature downstream the stage for arbitrary openings of the control valves is studied. Steam temperature measurements carried out during turbine transient operating conditions and presented here for a warm start-up showed a potential accuracy of the investigated model.

In the rotor thermal stress model, Green’s function and Duhamel’s integral are used. A crucial effect of the variation of material physical properties and heat transfer coefficient on the shape of Green’s function and thermal stress evolution was shown on the example of a simple cylinder representing turbine rotor heated at constant rate. In order to take into account these variations the so-called equivalent Green’s function was proposed, which provides a potential for accurate calculation of thermal stresses both for simple geometries and temperature changes (heating-up of cylinder), and for more complicated shapes of real steam turbine rotors (e.g., blade grooves). However, its general applicability to steam turbine rotors should be confirmed by further investigations. The equivalent Green’s functions are valid for given in advance thermo-flow conditions and their applicability to turbines’ start-ups is possible thanks to the exact definition at design phase of start-up types and conditions at which they occur.

Received in July 2016

References


ISSN 0070-3205 Transactions IFFM 133(2016) 13–37


A methodology for online rotor stress monitoring using equivalent... 37


Łukasz Brenkacz, Grzegorz Zywica

An experimental investigation conducted in order to determine bearing dynamic coefficients of two hydrodynamic bearings using impulse responses

Department of Turbine Dynamics and Diagnostics, Institute of Fluid Flow Machinery, Polish Academy of Sciences, 80-231 Gdańsk, Fiszera 14, Poland

Abstract
The paper presents the experimental investigation carried out in order to identify bearing dynamic coefficients of two hydrodynamic bearings from impulse responses. In this method the stiffness, damping and mass coefficients of two hydrodynamic bearings were calculated in a single algorithm. For each rotational speed, were obtained 12 dynamic coefficients which enable to fully describe the dynamic state of the rotor. Exciting force signals, applied using an impact hammer were shown. Displacements of the shaft were measured by eddy current sensors. The measurements were carried out at various rotational speeds, including resonance speeds. The vibration amplitude in a resonance case increases significantly with time after the excitation was induced by an impact hammer. Excitations of the rotor with an impact hammer were recorded using a high-speed camera. These unique recordings and simultaneous analysis of the trajectory of the bearing journals depict the contact phenomena that occur during impulse excitation of the rotor.

Keywords: Hydrodynamic bearings; Bearing dynamic coefficients; Experimental study

Nomenclature

\[ d \] \quad \text{rotor diameter}

\[ D \] \quad \text{disc diameter}

*Corresponding Author. Email adress: lbrenkacz@imp.gda.pl

ISSN 0079-3205 Transactions IFFM 133(2016) 39–54
1 Introduction

Values of stiffness and damping coefficients of hydrodynamic bearings are very important for vibration analyses carried out for fluid-flow machines [1,2]. Following difficulties found in the computational identification of such coefficients, many experimental methods for determination of their values have been proposed [3,4].

Experimental investigations were conducted with different types of bearings, in particular, with bearings that have a complex construction. The paper [5] presents the identification of dynamic coefficients of a hybrid gas bearing that has a sophisticated and robust construction with a complex structure of the foils. The bearing supports load both on a hydrostatic and hydrodynamic film. The Monte Carlo method was employed for the identification of dynamic parameters of a rotor supported by magnetorheological damping devices [6]. Literature studies showed that an experimental determination of stiffness and damping parameters of the bearings is conducted not only for radial bearings but also for thrust ones. The experimental identification of stiffness and damping characteristics for an axial foil bearing has been dealt with in detail in the article [7].

The method for the determination of bearing dynamic coefficients applied by the authors of this paper operates in frequency domain. Its first version was presented by Nordmann and Schoellhorn [8]. The basic algorithm was extended by adding the possibility to compute sixteen stiffness and damping coefficients [9].

The authors of the paper [10] modified the algorithm for the computation of stiffness and damping coefficients introduced by Qiu and Tieu in such a manner that it also allows calculating eight mass coefficients. The determination of stiffness, damping and mass coefficients by means of a single algorithm permits verification of the results already in a preliminary stage of an experimental investigation. The identified dynamic coefficients of a bearing can be verified on the basis of mass coefficients since a shaft mass is usually known in advance. This approach allows determination of all dynamical parameters of a rotor – bearings system by means of an experimental research.

The properly processed signal of a system response and an excitation force
signal are used to determine the values of bearings’ dynamic coefficients in linear form. They are determined at specific operating conditions of the rotating system. The article [10] provides the example of calculations based on data from a numerical model. The part of a signal corresponding to the stable operation should be subtracted from the signal registered during the experimental testing [11]. The sensitivity of the method for identification of dynamic coefficients of hydrodynamic bearings based on a signal obtained from a numerical model is presented in [12]. The issues discussed in this paper included, inter alia, the impact of a nonuniform distribution of the force exciting rotor vibration, rotor unbalance, positioning of the measurement sensors, etc. It turned out that there are parameters which have a very significant impact on the results of experimental investigations, but some of them can be easily corrected.

The work described in this paper is a continuation of previously conducted research [10–13] in which the measurement method and its precision were described in detail. The article discusses the manner of performing experimental research. The construction of the test stand, the test apparatus used and the signals measured during experimental tests are covered. The video recordings taken by a high-speed camera demonstrate the impact hammer’s slip over the surface of a rotating rotor shaft. The results of such testing have not yet been presented by any other researcher.

2 Basic technical characteristics of the test stand

The test stand for testing small rotors was built in order to conduct experimental research on rotor – bearings systems and to analyse defects, such as bearing damage, rotor unbalance, misalignment, etc. The photo of the test stand is presented in Fig. 1.

Figure 2 shows the diagram of the test stand on which the most important components are marked along with their dimensions. The length of the test stand is 1.25 m, and its width and height are 0.36 m and 0.65 m, respectively. The axes of the coordinate system used during the experimental investigation are shown in the top left-hand corner of this figure. The test stand rests on a 13 mm steel plate with attached two channel bars equipped with rubber feet that allow for height adjustment and leveling of the plate. Its weight – without the supporting structure – was approximately 60 kg. The rotor shaft was supported by two bearings. The system was driven by a three-phase motor with a maximum speed of 3450 rpm. The motor rotations were adjusted by means of a frequency inverter with a capacity of 1.5 kW. The motor was mounted to a gear that increases
the speed with a gear ratio 3.5:1. The presence of the inverter allows to vary the motor speed up to 12,000 rpm. The gear is connected to the rotor shaft by means of a permanent coupling. The coupling diameter is 50 mm and its length is 60 mm. The oil lubricated bearing system was equipped with the pump. During the experimental tests, the oil pressure was 0.16 MPa.

The tested rotor had a length of $L_{\text{shaft}} = 920$ mm. The distance between the coupling and the first bearing support was 170 mm. The rotor was mounted in two bearing supports. The distance between the supports (i.e., $l_1 + l_2$) was
An experimental investigation conducted in order to determine bearing.

580 mm. The bearing located closer to the motor is marked with the number 1, while the bearing located on the other end of the rotor has the number 2. The rotor disc was precisely centered between the bearing supports, so the distances \( l_1 \) and \( l_2 \) equal each other (see Fig. 2). The rotor diameter was \( d = 19.02 \) mm and the rotor disc – \( D = 152.4 \) mm. The excitations were performed using an impact hammer at the point \( P_e \), at a distance of \( l_e = 30 \) mm from the rotor disc’s midpoint. For safety reasons, the rotor – bearings system was equipped with a lockable casing made of hard transparent plastic.

The rotor was supported by two hydrodynamic bearings with the same geometries. The radial bearing clearance was \( 76 \) \( \mu \)m and the bearing length was \( L = 12.6 \) mm. Every bearing had two supply ports situated on both sides of the shaft. The supply ports have a diameter of 2.54 mm. The viscosity grade of the lubricating oil was ISO 13.

3 Experimental testing procedure

The calculation diagram of the experimental determination of bearing dynamic coefficients is shown in Fig. 3. In the first step, the necessary research was carried out. The selected point \( P_e \) located on the rotor shaft surface (shown in Fig. 2) was hit in a horizontal direction (X) with an impact hammer when the rotor was operating at constant speed. This action was repeated around a dozen times within 40 s. Then, the measurement was repeated but hitting the shaft was realized in a vertical direction (Y). In the second step, the reference signal (corresponding to the stable operation of the rotor) was subtracted from the signal registered after the excitation. This operation was carried out using the computer program called ‘Signal’ [14], created for this purpose. The third step was zeroing values of the signal corresponding to an excitation force, omitting the values related to the main component. The signals thus obtained were subjected to a FFT (fast Fourier transform) analysis [15]. The spectral components received during the zeroing process are analyzed in a frequency domain. The matrixes \( \mathbf{A} \), \( \mathbf{Z} \) and \( \mathbf{I} \) were then created, the detailed description of which is given in the paper [10]. The dynamic coefficients of hydrodynamic bearings were calculated by the least squares method.
4 Test apparatus

In fluid-flow machinery, unbalance and misalignment are common causes of elevated vibration levels [16]. Before starting the measurements, the shaft alignment was conducted by means of the OPTALIGN Smart RS device (manufactured by Prüftechnik) and two wireless laser sensors. The rotor was balanced using the Diamond 401 device, produced by MBJ Electronics s.c.

A SCADAS Mobile system, manufactured by LMS International, was used for data acquisition purposes. It was connected – via the Ethernet interface – to a laptop equipped with the Test.Lab 11B software. Four eddy current sensors (model CWY-DO-501A) were employed for measurements of the rotor displacement. The uniaxial displacement sensors were placed on the bearing supports. The eddy current sensors were positioned at the points \( P_1 \) and \( P_2 \) equidistant from the bearings \( (l_{1p} = l_{2p} = 25 \text{ mm}) \). They were mounted at an angle of 90° to each other and at an angle of 45° to the axes of the global coordinate system. These sensors were connected to a SCADAS Mobile analyzer by means of a demodulator. Their maximum measurement range was 1 mm. Four uniaxial accelerometers (model 608A11) were implemented for vibration velocity measurements. The accelerometers were mounted in the bearing supports in two directions that were mutually perpendicular and also perpendicular to the rotational axis of the rotor. An impact hammer, 21.6 cm long and weights 0.16 kg – made by PCB Piezotronics – was employed to realize excitations. During the measurements, the hammer was fitted with a hard impact cap (ST STL), which implied that the
maximum impact force that could be obtained was 360 N. The rotational speed of the rotor was measured by using a 152 G7 laser tachometer, manufactured by Optel Thevon.

The analysis of a hammer impact in slow motion was also carried out within the framework of this research. A high-speed Phantom v2512 camera (manufactured by Vision Research) was employed for this purpose. The camera is equipped with a high-performance CMOS sensor and has a throughput of 25 Gpx/s which enables recording frames at 25,600 fps (frames per second) and 1280×800 pixels resolution. It enables recording frames at 1 million fps with the lowest resolutions and at 25,600 fps with the maximum resolution of 1280×800. For signal analysis, the software programs called PCC 2512 (Phantom Camera Control) and TEMA Motion were used, which were made by Vision Research and Image Systems, respectively. Two lenses were used, namely Zeiss Planar T* 50mm F/1.4 ZF.2 and Nikon 200 mm F4.0. ED-IF AF Micro-Nikkor.

5 Signal processing

In order to use the results from experimental research in further analysis, the signals must be processed in an appropriate way. The eddy current sensors are positioned at an angle of 45° to the axes of the global coordinate system. The appropriate processing of the signal registered for the first bearing was implemented by means of Eqs. (1) and (2). Analogical operations were carried out for the second bearing. The response recorded at the first bearing in the X direction after the excitation in the Y direction is denoted by 1XY. A transformed signal is denoted by 1XY'.

\[
\begin{bmatrix}
1XX' \\
1YX'
\end{bmatrix} = \begin{bmatrix}
\cos \emptyset & \sin \emptyset \\
-sin \emptyset & \cos \emptyset
\end{bmatrix} \begin{bmatrix}
1XX \\
1YX
\end{bmatrix},
\]

(1)

\[
\begin{bmatrix}
1XY' \\
1YY'
\end{bmatrix} = \begin{bmatrix}
\cos \emptyset & \sin \emptyset \\
-sin \emptyset & \cos \emptyset
\end{bmatrix} \begin{bmatrix}
1XY \\
1YY
\end{bmatrix}.
\]

(2)

The signals registered by eddy current sensors were corrected using Eqs. (3) and (4). This correction was due to the fact that the sensors were situated next to the bearing supports, rather than at the bearing midpoints. Based on the sensitivity analysis described in article [12], it can be stated that the failure to take into account the above correction could have resulted in differences of several percentage points for the calculated values of bearing dynamic coefficients.

\[
\begin{bmatrix}
XX_1 \ YY_1 \\
XX_1 \ YY_2
\end{bmatrix} = \frac{1}{l_w + l_2} \begin{bmatrix}
l_w + l_1 & l_w - l_1 \\
l_w - l_2 & l_w + l_2
\end{bmatrix} \begin{bmatrix}
XX_{1p} \ YY_{1p} \\
XX_{2p} \ YY_{2p}
\end{bmatrix},
\]

ISSN 0079-3205 Transactions IFFM 133 (2016) 39–54
\[
\begin{bmatrix}
  XY_1 & YX_1 \\
  XY_1 & YX_2
\end{bmatrix} = \frac{1}{l_e + l_2} \begin{bmatrix}
  l_e + l_1 & l_w - l_1 \\
  l_e - l_2 & l_e + l_2
\end{bmatrix} \begin{bmatrix}
  XY_{1p} & YX_{1p} \\
  XY_{2p} & YX_{2p}
\end{bmatrix}.
\]

The calculations also reflect the fact that the point \( P_w \) (the point in which the excitation forces are acting) is not equidistant to both bearing supports, but has a distance of \( l_w = 30 \text{ mm} \) to the rotor disc’s midpoint. This fact has been taken into account by applying the necessary corrections using the formulas (5), (6) and (7).

\[
l_{1f} = l_1 + l_e = 290 + 30 = 320 \text{ [mm]},
\]

(5)

\[
l_{2f} = l_2 - l_e = 290 - 30 = 260 \text{ [mm]},
\]

(6)

\[
F_{1shifted} = F_1 \frac{l_{2f}}{2l_1}, \quad F_{2shifted} = F_2 \frac{l_{1f}}{2l_2}.
\]

(7)

6 Measurement results

If the shaft displacement signal at a bearing support is measured and recorded for 10 s (as shown in Fig. 4) it is possible to observe the ‘rotor flow’, besides the displacements related to the shaft revolutions that are changing 3250 times per minute. Such changes are specific to hydrodynamic bearings when the rotor shaft, apart from its rotational motion, also performs the motion in relation to the bearing bush. The vibration analysis is a very complex task in terms of spectral analysis, even for properly operating machines. In the vibration spectrum of a signal both harmonic (1X, 2X, 3X, \ldots) and sub-harmonic (X/2, X3, \ldots) components and also the components related to vibration of a supporting structure can be present.

In order to obtain a set of data required to determine bearing dynamic coefficients for a single rotational speed, a two-step analysis was necessary. In the first step the excitation force was applied in the Y direction (at the point \( P_w \), see Fig. 2) by means of an impact hammer. In the second step, the measurement was conducted at the same rotational speed with the difference that hitting the shaft was realized in a horizontal direction (X). The exemplary signal of the excitation force acting in the Y direction was shown in Fig. 5a. Figure 5b presents the signal measured in the Y direction by an eddy current sensor positioned close to the bearing no. 2, when the rotor shaft was rotating at a constant speed of 4500 rpm. The graph shows a stable operation of the rotor together with periodic amplitude increases, caused by a hammer hit.

The values of the excitation force were around 80 N. The signal of the excitation force, shown for a shorter time period, is presented in Fig. 6. Most of the
excitations lasted from 0.1 ms to 0.2 ms. The values of the excitation force were zeroed, besides the values of the main peak. The exemplary results before and after zeroing these values can be seen in Figs. 6a and 6b, respectively.

Figure 7 contains the signal measured in the X and Y directions – Figs. 7a and 7b, respectively – during stable operation of the bearing no. 2. The next parts of this figure present the signals measured in the X and Y directions after applying the excitation in the X and Y directions – parts c) and d), respectively – by means of an impact hammer (the signal corresponding to this excitation force is shown in Fig. 6). It can be observed that the vibration amplitudes changed rapidly and then stabilized – the rotor returned to its stable operation. The amplitudes were increasing or decreasing (for a short period of time), depending on the present location of the rotor.

The computer program ‘Signal’ served to choose the relevant parts from the excitation forces and reference signals (i.e., the signals registered after the excitations). Choosing the appropriate signal ranges was the most time-consuming part of bearing dynamic coefficients calculation. The signals presented in Fig. 8 are the results of subtraction of the signals situated in Figs. 7a and 7b from the signals visible in Figs. 7c and 7d, respectively. The left graph in Fig. 8 demonstrates the response signals corresponding to the bearing no. 1 after the excitation applied.
Figure 5: a) Excitation force vs. time. b) The signal measured in the Y direction close to the bearing no. 2 at 4500 rpm after the excitation in the Y direction was applied by an impact hammer.

Figure 6: Excitation force in the X and Y directions vs. time. Excitations were applied at the speed of 4500 rpm: a) signal without zeroing values outside the range of the main component, b) signal with zeroing.

in Y (lines marked with 1YY and 1XY) and X (lines marked with 1YY and 1XY) directions. Similarly, the right graph in Fig. 8 shows the response signals which relate to the bearing no. 2. The first letter after the bearing number indicates the direction of displacement measurement and it is followed by the direction of
An experimental investigation conducted in order to determine bearing...

Figure 7: Stable operation of the bearing no. 2 at 4500 rpm (a and b) and the signal registered after applying the excitation (c and d). The graphs present the displacements in the X (a and c) and Y (b and d) directions.

excitation force. For instance, the system response signal registered in the X direction for the second bearing after the excitation force had been applied in the Y direction is indicated in Fig. 8b by means of the symbol 2XY.

Figure 8: Vibration amplitude of the rotor after the excitation in X and Y directions vs. time:

a) bearing no. 1, b) bearing no. 2 (the reference signal was subtracted from the signal obtained after excitation).

At some rotational speeds, vibration amplitudes have reached worryingly high levels that can be linked to the resonance phenomenon. Figure 9 demonstrates the signal registered at the second hydrodynamic bearing during its operation at a constant speed of 4000 rpm after the excitation in the Y direction was applied.
The signal registered before the excitation was subtracted from the signal recorded after the hammer strike was performed. It can be observed that the vibration amplitude rises from about 0 to above 30 $\mu$m within 0.3 s. Such behaviour is typical for systems functioning under unstable operating conditions and its occurrence can lead to machine damage.

Figure 9: The vibration amplitude (that grows over time) registered on the bearing no. 2 at 4000 rpm after the excitation in the Y direction was applied by an impact hammer.

7 Analysis of the impact hammer slip

The way the impact hammer hits the rotor shaft is an important element of the research from the point of view of the accuracy of testing results. During numerical calculations, the effect of the hammer’s slip over the surface of a rotating rotor shaft is extremely hard to implement in a computer program. Also, the possibility to observe this phenomenon during experimental tests is very difficult to achieve. The shaft displacements were registered (in two directions that were mutually perpendicular and also perpendicular to the rotational axis of the shaft) using the eddy current sensors situated next to the bearing supports. On this basis, it could not be concluded that the hammer strike was carried out as intended, i.e., in the specified direction of impact and without slipping. Therefore, it was decided to carry out observations using a high-speed camera.

Figure 10a shows one frame from a video recording in slow motion, showing the impact hammer’s strike. The aluminium disc is visible on the left-hand side of this figure. A black-and-white striped ribbon was stuck on the rotor shaft in order to track its movement as precisely as possible. The impact hammer’s cap is in the upper part of the figure. Figure 10b shows a zoomed-in shaft part in which
the excitations were applied by means of an impact hammer.

The duration of action of the exciting force (modal hammer contact with the shaft) is from 0.1 to 0.2 ms. The rotor diameter is 19.02 mm. The surface speed is 4489 mm/s at a shaft speed of 4500 rpm. The impact hammer surface can be regarded as a flat surface, the rotor surface is curved. After many hours of experimental research, there are visible slight traces of hits on the rotor surface. Their size does not exceed 0.5 mm. The duration of an impact is short enough that the friction force created after exciting through a modal hammer may be omitted. This fact was also confirmed by the observations made using a high-speed camera.

The description of the hammer movement recorded by means of a high-speed camera is quite cumbersome. However, the experimental investigation has shown that the hammer slip is small and may be neglected. Forces that act on the rotor may be treated as excitations applied in the directions corresponding to a direction of one of the axes (X or Y). This information is valuable and confirms the relevance of initial assumptions made.

8 Summary and conclusions

The article presents the experimental research aiming at the determination of stiffness, damping and mass coefficients of two hydrodynamic bearings. The associated methodology employed has been described in detail, including measured signals and their processing required for carrying out the calculation process.
Moreover, the tests were also performed by means of a high-speed camera.

The paper available in the scientific literature describe various algorithms for estimation of bearing dynamic coefficients. It is very rare to encounter paper giving a detailed description of signals measured experimentally. However, there are many paper describing the impulse method for determining the parameters of bearings, seals and dampers [17,18]. The lack of thorough analyses concerning the excitation response signals and their processing in the current literature were the motive for writing this article. Another novelty in this paper is the consideration given on the impact hammer slip.

The experimental studies involved the measurement of the forty seconds periods of rotor operation (around a dozen times) during which the rotor vibrations were excited using an impact hammer hitting the shaft separately in two perpendicular directions. Only the signal fragments beginning with the moment of impact and ending after the time the rotor returns to the normal operation were used as the basis for calculations. The computer program called ‘Signal’ has proven to be an effective tool for selecting signals, their processing and storing in a suitable form.

The signal preparation procedure included the subtraction of the reference signal (i.e., the signal registered just before the excitation occurred) from the signal measured immediately after the excitation took place. Such a procedure was carried out for two bearings in a single operation and for every excitation signal that was measured by eddy current sensors. In general, it may be said that finding the reference signals was relatively easy for the entire speed range. However, in the case of some speeds near the resonance speed (around the speed of 4000 rpm), this task was slightly more difficult, mainly due to the fact that the registered signals were phase-shifted – it was detected after comparison of the frequency of a sinusoidal signal measured by the eddy current sensor situated next to the bearing support no. 1 with the frequency of the signal measured in the same direction by the sensor positioned close to the bearing support no. 2.

It seemed that the impact hammer’s slip (and the related unequal distribution of exciting forces on the rotor shaft surface) occurring during the experimental tests is a factor which may have an impact on calculation results. The contact of the hammer with a fast-rotating shaft surface may result in the situation that the excitation force is not perpendicular to the axis of the shaft. No information is available on this point in the scientific literature. The numerical computations performed also did not allow for verification of whether a slip of the impact hammer can have a considerable impact on the results obtained. The authors of the article came up with the idea of using a high-speed camera in order to study this
An experimental investigation conducted in order to determine bearing... phenomenon in a slow motion. The observation of the impact hammer’s strike (using the 1/250th slow motion playback mode) has led to the conclusion that the contact time is sufficiently short to assume that the direction of force applied by the hammer is fully in line with the intended direction.

Acknowledgement The research is financed by the Polish National Science Centre as a research project no. 2015/17/N/ST8/01825.

Received in July 2016

References


David Garcia\textsuperscript{a*}, Grzegorz Liśkiewicz\textsuperscript{b}

**Stable or not stable? Recognizing surge based on the pressure signal**

\textsuperscript{a} Mechanical and Aerospace Engineering, University of Strathclyde, 75 Montrose Street Glasgow G1 1XJ, UK
\textsuperscript{b} Institute of Turbomachinery, Lodz University of Technology, 219/223 Wólczańska, 90-924 Łódź, Poland

**Abstract**

The surge protection may be worth millions of dollars. This is typical price of a centrifugal compressor repair combined with additional cost of nonfunctionality of an industry employing it. This threat is normally secured by application of an antisurge systems. Typically they are activated at predefined working conditions when compressor mass flow rate approaches region affected by the surge. As a result those systems are vastly limiting its operational range usually by a desirable region where compressor attains large pressure ratio. Therefore, a modern anti-surge systems are aiming at diminishing this tradeoff by reacting to the real pressure signal gathered at high frequency. This paper presents one of those methods employing singular spectrum analysis. This algorithm has not been widely used for this application, while it was shown herein that it may bring clear distinction between stable and nonstable working condition, even at presurge conditions. Hence in further perspective it may bring anti-surge protection quality, that was not met with another methods.

**Keywords:** Singular spectrum analysis; Nonlinear dynamics; Statistical pattern recognition; Compressor; Surge

\textsuperscript{*Corresponding Author. Email address: david.garcia@strath.ac.uk

ISSN 0079-3205 Transactions IFFM 133(2016) 55–68
1 Introduction

1.1 Unstable phenomena in centrifugal compressors

Unstable phenomena in centrifugal compressors were first identified by Emmons in 1950s [1]. In 1976 Greitzer introduced a mathematical model describing a development of surge [2], which was verified by an experiment [3]. Till now, the model was analysed and developed by numerous researchers including Hansen et al. [4], Elder and Gill [5], Fink et al. [6], Gravdahl and Egeland [7], Willems [8], Meuleman [9], Helvoirt [10] and more recently Yoon et al. [11]. There are also examples of another attempts of simulating surge. Macdougall and Elder [12] used relations for a polytropic compression process in combination with principles of conservation of mass, momentum and energy. This method was followed by Badus et al. [13] and Botros [14] including variation of the impeller speed. These three models can be applied to both axial and centrifugal compression systems, nevertheless, they are quite complex.

Figure 1: Compressor performance curve, system resistance curve and the surge region.

In above mentioned works a concept of surge limit was introduced. Figure 1 shows a performance curve of a compressor. Area on the left side of the surge limit is beyond the capabilities of the device due to the fact that there is a high risk of unstable work conditions appearance. On the other hand, the compressor operating in the vicinity of the surge limit achieves the highest pressure ratio and high efficiency. Therefore, this is a very attractive region from the point of view of the user. That is why a surge phenomenon, as well as local unstable structures
Stable or not stable? Recognizing surge based on... (rotating stall [15–16] and inlet recirculation [17]) remain a subject of interest for a wide range of scientists trying to develop modern antisurge devices [18].

Figure 2: Classification of anti-surge systems by their moment of reaction: 1 – prevention, 2 – fast detection, 3 – active control [18].

Figure 2 shows a classification of antisurge control systems by their moment of activation. The simplest ones (no. 1) reduce the compressor operational range to a certain region, which is separated from the surge limit by a predefined surge margin. This is the cheapest solution, yet, not always effective and limiting the system operational range. Another class of antisurge systems is based on a rapid instability detection (no. 2). Their effectiveness depends on the algorithm applied. Active antisurge control units are also considered (no. 3), where the oscillations are damped enabling the compressor to safely operate in a region which is potentially subjected to a surge phenomenon. The biggest disadvantages of the solutions no. 2 and no. 3 include their operational costs and at the same time their lack of versatility. That is why, researchers constantly seek a better design in the field of antisurge control units, which would reconcile two diverging aims – guarantee of safety and operation in close vicinity of the surge.

Figure 2 visualizes the fact of high potential of an active antisurge systems. Therefore it is not surprising that many researchers examined this type of surge protection schemes. It could be generalized that any protection system of that type is based on an input signal and uses a controller that steers some adjustable
elements. In most cases it was an adjustable valve [19–22], while Williams and Huang [23] and Jungowski et al. [10] considered speaker, Gysling et al. [7] moving wall in plenum, and Gravadhl et al. [13] variable shaft speed. Regardless of the chosen solution, there are some common problems of the active antisurge systems. This includes mainly a large level of complexity, high operational costs as well as lack of 100% guarantee of their reliability.

Antisurge systems based on the detection of the phenomena prior to surge are free from those problems and still provide operation in close vicinity of the surge margin. Those systems react on first surge indicators such as the mass flow decrease [24] or local flow separation [25]. It could be also based on simple pressure measurement subject to specific algorithm. One of them is singular spectrum analysis (SSA) which is an extension of principal component analysis (PCA) for nonindependent signals [26,27]. The SSA method allows to limit the signal analysis to a few first components that possess the greatest impact on the result. The method is widely used for the signal compression, trends prediction, structural damage detection [28]. The number of attempts to apply this method to surge detection is limited, yet provided promising conclusions [29,30].

Results of [31] have shown that after implementing SSA the compressing system had limited dimension and could be represented with just a few first principal components. Usually using three is enough because they contain the majority of the variance in the principal components decomposition. Moreover, it was shown that the first principal component was reproducing the value of the average pressure. Most promising for surge detection were components number two and three. In two dimensional space constructed from them the phase portrait of a signal changed into limit cycle with strong oscillations at inception of unstable flow structures. Moreover, this method has proven to be sensitive to all kinds of flow instabilities: local (like inlet recirculation) or global (like surge).

The aim of this paper is to continue this effort and analyze the effect of clustering of signals subject to SSA. In other words clustering appears when points from stable and unstable operation group in different spaces after SSA procedure. If this happens it is very easy to distinguish them in the same way in which SSA is applied to distinguish healthy from unhealthy mechanical structures [31].
2 Method

2.1 Experimental rig

Experiment was carried out on a single stage centrifugal blower DP1.12 [32]. The inlet pipe had diameter $D_{in} = 300$ mm and was followed by the Witoszynski nozzle and the impeller with 23 blades, vaneless diffuser and circular volute. The rotor inlet diameter at the hub and the inlet span equaled $D_{hub} = 86.3$ mm and $b_1 = 38.9$ mm, respectively. At the outlet those values changed to $D_2 = 330$ mm and $b_2 = 14.5$ mm. The diffuser outlet diameter was equal to $D_3 = 476$ mm. The volute radius was gradually increasing streamwise from the volute tongue gap of 5 mm towards the outlet pipe of diameter $D_{out} = 150$ mm. A throttling valve was mounted at the end of the outlet pipe. The rotor was driven by an asynchronous alternating current (AC) motor with the rotational speed of 100 Hz that corresponded to a flow rate of 0.75 kg/s and pressure ratio $PR = 1.08$. Rotational speed yielded the impeller tip speed equal to 103 m/s.

![Cross section of the experimental rig used in this study with positions of the pressure gauges.](image)

The test stand was equipped with 2 dynamic subminiature Kulite transducers (XCQ-093-5D, natural frequency 150 kHz) connected to an Iotech Wavebook 516/E data acquisition system. Their position is presented on the stand cross-section in Fig. 3. Transducers were mounted flush to the walls to measure the static pressure at the rotor inlet and at the volute outlet. Similarly to previous studies performed on this stand the position of the throttling valve was described by the dimensionless throttle opening area parameter referred to as $TOA$. 

ISSN 0079-3205 Transactions IFFM 133(2016) 55–68
TOA = 100% corresponds to fully opened valve, while TOA = 0% corresponds to fully closed valve. Each signal contained $2^{21}$ samples gathered with the frequency of 100 kHz. The temperature was controlled and remained constant during the entire experiment at 26 °C.

2.2 Singular spectrum analysis approach

The methodology used here is based on PCA which is a statistical procedure that uses an orthogonal transformation to convert a set of multivariate observations into a set of linearly correlated variables. However, PCA generally assumes that the data components are independent, but in the case of time series, the values are generally non-independent, and thus an extension of PCA called singular spectrum analysis (SSA) provides a better alternative [27]. SSA is PCA applied to lag versions of a single time series variable. It follows the following four steps:

**Data collection:** The variables measured, in this particular case the pressure, are arranged in vectors with length $N$ as $\mathbf{x} = (x_1, x_2, \ldots, x_j, \ldots, x_N)$ where $j = 1, 2, \ldots, N$.

**Embedding:** Given a window with the time series by $W$ points ($1 < W \leq \frac{N}{2}$) the $W$-time lagged vectors arranged in columns are used to define the trajectory matrix $\tilde{\mathbf{X}}$. These vectors are padded with zeros to keep the same vector length.

$$
\tilde{\mathbf{X}} = \begin{pmatrix}
x_1 & x_2 & x_3 & \ldots & x_w & \ldots & x_W \\
x_2 & x_3 & x_4 & \ldots & x_{w+1} & \ldots & x_{W+1} \\
x_3 & x_4 & x_5 & \ldots & x_{w+2} & \ldots & x_{W+2} \\
x_4 & x_5 & x_6 & \ldots & x_{w+3} & \ldots & \vdots \\
x_5 & x_6 & \vdots & \ldots & \vdots & \ldots & x_N \\
x_6 & \vdots & \vdots & \ldots & x_N & \ldots & 0 \\
\vdots & \vdots & \vdots & \ldots & x_N & \ldots & 0 \\
\vdots & \vdots & \vdots & \ldots & 0 & \ldots & 0 \\
x_N & 0 & \ldots & 0 & \ldots & 0 & 0 \\
0 & 0 & \ldots & 0 & \ldots & 0 & 0
\end{pmatrix}
$$

The embedding matrix $\tilde{\mathbf{X}}$ is the representation of the system in a succession of overlapping vectors of the time series by $W$-points.
**Decomposition:** The empirical orthogonal functions (EOFs), which represent the principal directions of the system, are calculated by the eigenvalue and eigenvector decomposition of $\mathbf{C}_X = \mathbf{\tilde{X}}'\mathbf{\tilde{X}}$ (where $\mathbf{\tilde{X}}'$ denotes the transpose matrix of $\mathbf{\tilde{X}}$) by solving the eigenvalue problem defined

$$\mathbf{C}_X \rho_k = \lambda_k \rho_k .$$

(2)

The decomposition into eigenvalues yields $k$-eigenvalues and $k$ – eigenvectors which define orthonormal basis of the decomposition of $\mathbf{\tilde{X}}$. The eigenvalues $\lambda_k$ are ordered in decreasing and the eigenvectors $\rho_k$ in the same order that their corresponding eigenvalues. Each eigenvalue defines the partial variance in the direction of its corresponding eigenvector, therefore the sum of all eigenvalues gives the total variance of $\mathbf{X}$.

The decomposition into a certain number of principal components (PCs) of the source signal provides the distribution of the autocorrelated variance among these components. Projecting the measured data $\mathbf{\tilde{X}}$ onto the EOFs matrix yields the corresponding PC matrix $\mathbf{A}$ which contains the variance information distributed among these PCs:

$$A^k_n = \sum_{w=1}^{W} x_{n+w-1} \rho^k_n .$$

(3)

**Reconstruction:** A certain number of reconstructive components (RCs) are reconstructed by using the part of a time series that is associated with a single EOF or several by combining the associated PCs

$$R^k_n = \frac{1}{W} \sum_{w=1}^{W} A^k_{n-w+1} \rho^k_n ,$$

(4)

where $k$ – eigenvectors give the $k^{th}$ RC at $n$ – time between $n = 1, \ldots, N$ which was embedded in a $w$ – lagged vectors with the maximum $W$ – length.

The variance is distributed into the RCs in a decreasing order from the first components until the last ones. Therefore, the first RCs contain much more descriptive information of the dynamical system than next ones. By the use of a certain number of RCs, an approximated reconstructed source signal was developed.

### 2.3 Clusterisation

This section introduces the effect of clusterisation in order to be used as a metric for distinguishing between stable and not stable phenomena. The RCs obtained
in the signal decomposition are used as reference state, which describes an stable working condition of the turbocompressor. The detection whether the system is stable or not stable is based on the comparison of an observation signal to the reference state. A new observation signal is multiplied with the RCs (inner product between two vectors) and hence projected in the reference space constructed by the RCs

\[ T = x_s R, \]

where \( T \) is the transformation of the observation signal \( x_s \) onto the reference space \( R \). The dimension of the reference space is \( p \leq W \). Therefore, when two observation signals are compared onto the reference space, it is expected that if the two observation signals are similar, the distance between them is minimum, however if they are different, the distance between them will increase.

### 3 Results

#### 3.1 Regions of operation of the machine

Figures 4 and 5 present pressure values as a function of the throttle opening area obtained based on the outlet and inlet signals respectively. Grey area represents the region between minimal and maximal values obtained in each valve position throughout the measurement. Bold line connects average values registered therein. Each of 146 measurements presented in the plot contained \( 2^{21} \) samples gathered with the frequency of 100 kHz. This means that measurement lasted over 20 s and corresponded to over 2000 impeller rotations. A similar phenomenon to the represented in these plots was observed in all the measurements during the experiment.

#### 3.2 Base constructed from first two reconstructed components (RC1 and RC2)

Figures 6 and 7 represent families of signals gathered at \( TOA < 65\% \). All of them are projected into two-dimensional space constructed in a way described in Sec. 2.3. This space is spanned by two principal components named as RC1 and RC2 henceforth. These are two initial components obtained in the SSA method conducted on signal gathered in the blower nominal point \( (TOA = 30\%) \). The value of throttle of each projected signal is marked with an adequate color corresponding to a colorbar.
Figure 4: Blower outlet min-to-max and average pressure in a function of the throttle opening area (Copyright ©2014 Elsevier).

Figure 5: Blower inlet min-to-max and average pressure in a function of the throttle opening area (Copyright ©2014 Elsevier).

4 Discussion

4.1 Three flow conditions

Based on Figs. 4 and 5 one can distinguish 3 phases of operation of the machine in analyzed control points. At $TOA > 20\%$ it was working in a stable manner with constant min-to-max span caused by a local pressure fluctuations and inevitable signal noise. The average pressure in inlet was close to zero at all circumstances. The average outlet pressure was rising and dropping by following the blower performance curve. At $9\% < TOA < 20\%$ the amplitude of pressure oscillation
Figure 6: Points representing outlet pressure signals in a space constructed from the principal base (RC1, RC2) used in this study.

Figure 7: Points representing inlet pressure signals in a space constructed from principal base RC1, RC2 used in this study.

significantly raised in both locations indicating appearance of flow instability. It can be, however, regarded as moderate as the amplitude rise was not bigger than twice in size. Nevertheless, the observed phenomenon can be regarded as global for the machine, as it was observed in both control points. According to widely
accepted nomenclature [3,4,7,32,33] this state can be regarded as mild surge or transient phase before inception of the deep surge. At \( TOA < 9\% \) the amplitude of oscillations rose significantly. In inlet it equaled 5 kPa, while in the outlet around 10 kPa. This difference clearly corresponds to a Greitzer model, where oscillations are strongest in the plenum [6,7] and hence, the surge was observed.

4.2 Representation of the signals in reference space RC1, RC2

Based on Figs 6 and 7, one can distinguish the different working conditions of the machine operation phases. The observation signals are projected onto the reference space which is able to identify alterations within the dynamical system [28,34,35]. In both plots points representing pressure signal have a tendency to be projected along a straight line.

Figure 6 represents the projection of the observation signals onto the reference state created by the signals measured in the outlet. Results suggest that this analysis can be also used to identify the nominal working conditions of the machine similarly to a correlation integral applied by Gu et. al [34]. All points are aligned until the machine goes to deep surge condition \( (TOA < 10\% \)\). In this case, the points move away from the straight line, which indicates the unstable phenomena. It can be also observed that the optimum working conditions are around at \( TOA = 30\% \) where all points concentrates in a cloud of point. This indicates that the system nominal working conditions are attained at this \( TOA \).

Figure 7 represents the projection of the observation signals onto the reference state of signal measured in the inlet. It can be observed that for observations corresponding to \( 30\% > TOA > 20\% \) the spacing between points increases and hence it alerts that deep surge is about to occurs. This is very important, that the indication appears so quickly, because this measurement point is located further upstream of the impeller compared to studies, where the inlet recirculation was normally observed [32,36,37]. This phenomenon is known to strongly influence the variations of the signal in the phase space, and hence would be strongly influencing position of the points in given projection [31]. Nevertheless, it could be observed, that even in other control points analysis of pressure signal could give indication of incoming surge inception. However in this range, the points are still proportionally aligned by the contribution of RC1 and RC2. When the \( TOA < 10\% \), the machine enters deep surge condition (global effect). This phenomena could be clearly distinguished because their projection points move away from the line and the spacing becomes enormous. This gives again very clear indication between stable and unstable signals.
5 Summary

In presented study the signals were gathered in inlet and outlet zones of the centrifugal blower and subject to dynamical study by means of the singular spectrum analysis. Signal was decomposed into a principal components that were used to reconstruct the reference space by means of the reconstructive components. The projection of the observation signals onto the reference state was studying in order to identify different working conditions of the centrifugal blower. The analysis was implemented in signals measured in the inlet and outlet by different throttle opening areas that defines different working conditions. The main conclusions of this study are listed below:

- Method allows to distinguish between stable and unstable working conditions by considering points spacing and positioning in relation to another points. The results obtained clearly align with the results obtained in a previous study by the authors [31]. However this analysis is able to characterise dynamical responses in multidimensional vectors that make easier the visualization of the surge effect.
- The projection of the signals onto the reference state provides information of global effects such as deep surge but also local effects.
- Apart from detecting unstable flows method could be also applied for specifying nominal working conditions of the blower.
- The use of two dimensions of the reference space (RC1 and RC2) is enough to detect the phenomena described by different working conditions at different TOA.

Received in July 2016

References


ISSN 0079-3205 Transactions IFFM 133(2016) 55–68
Stable or not stable? Recognizing surge based on... 67


ISSN 0079-3205 Transactions IFFM 133(2016) 55–68


Filip Grapow*, Grzegorz Liśkiewicz

Compressor modeling using Greitzer model validated by pressure oscillations

Institute of Turbomachinery, Lodz University of Technology,
219/223 Wólczańska, 90-924 Łódź, Poland

Abstract

Nowadays compressors are used in almost every industry. Compressor failure can suspend production of the whole system so the importance of preventing from failures is obvious and essential. One of the most dangerous flow instabilities which is capable of destroying machine in few seconds is surge, which occurs in conditions of low mass flow rate. Greitzer model, apart from its long history, is still most common mathematical model describing surge. It is widely used to predict the surge onset and pressure oscillations during it. However, it is based on parameters that are not directly related to real machine and their choice is not always obvious. Therefore, the calculations may be inaccurate which results in wrong surge prediction. The other approach to Greitzer model is presented in this paper, which in some cases can assure that the process of compressor modeling is more accurate. The applicability of Greitzer surge model for real machines has been analyzed. Method of implementation is based on experimental pressure signal gathered during unstable work of compressor. Presented method is based on experimental compressor characteristic and outlet pressure signal from unstable work of compressor. From that data it is possible to determinate the value of Greitzer model’s parameters for selected operational point. Thanks to this method this model could be applied for reliable antisurge protection.

Keywords: Centrifugal compressor; Surge; Greitzer model; Compressor modeling

*Corresponding Author. Email adress: filip.grapow@p.lodz.pl
Nomenclature

\( A \) – pressure dimensionless amplitude
\( A_c \) – compressor pipe area, \( \text{m}^2 \)
\( A_t \) – throttle pipe area, \( \text{m}^2 \)
\( A_{in} \) – cross-section area of the passage at the impeller inflow, \( \text{m}^2 \)
\( A_{out} \) – cross-section area of the passage at the impeller outflow, \( \text{m}^2 \)
\( B \) – Greitzer model stability parameter
\( b_1 \) – blade height at the leading edge, \( \text{m} \)
\( b_2 \) – blade height at the trailing edge, \( \text{m} \)
\( D_2 \) – diameter of the impeller at the trailing edge, \( \text{m} \)
\( D_3 \) – diameter of the diuser outlet, \( \text{m} \)
\( D_{hub} \) – diameter of the hub at the leading edge, \( \text{m} \)
\( D_{in} \) – diameter of the inlet pipe, \( \text{m} \)
\( D_{out} \) – diameter of the outlet pipe, \( \text{m} \)
\( f_{nom} \) – nominal frequency, \( \text{Hz} \)
\( f_{rot} \) – experimental rotational frequency, \( \text{Hz} \)
\( G \) – Greitzer model parameter
\( L_c \) – compressor pipe length, \( \text{m} \)
\( L_t \) – throttle pipe length, \( \text{m} \)
\( L_p \) – blade passage length, \( \text{m} \)
\( m \) – mass flow, \( \text{kg/s} \)
\( \dot{m}_c \) – mass flow in compressor pipe, \( \text{kg/s} \)
\( \dot{m}_t \) – mass flow in throttle pipe, \( \text{kg/s} \)
\( \dot{m}_{nom} \) – nominal mass flow, \( \text{kg/s} \)
\( p_c \) – pressure of compressor pipe inlet, \( \text{Pa} \)
\( p_t \) – pressure of throttle pipe inlet, \( \text{Pa} \)
\( p_p \) – plenum pressure, \( \text{Pa} \)
\( p_{s-in} \) – signal representing static pressure at the blower inlet, \( \text{Pa} \)
\( p_{s-out} \) – signal representing static pressure at the blower outlet, \( \text{Pa} \)
\( R_\theta \) – volute radius, \( \text{m} \)
\( TOA \) – throttle opening area, \( \% \)
\( t \) – time, \( \text{s} \)
\( t \) – dimensionless time
\( U_t \) – blade tip speed, \( \text{m/s} \)
\( V_p \) – plenum volume, \( \text{m}^3 \)
\( Z \) – number of blades
\( \Delta A \) – amplitude absolute error
\( \Delta P \) – pressure difference, \( \text{Pa} \)

Greek symbols

\( \Delta \omega \) – frequency absolute error, \( \text{Hz} \)
\( \pi \) – pressure ratio
\( \pi_{nom} \) – nominal pressure ratio
\( \delta \) – blade tip clearance, \( \text{m} \)
\( \Phi \) – dimensionless mass flow

ISSN 0079-3205 Transactions IFFM 133(2016) 69–89
1 Introduction

Centrifugal compressors are turbomachines used in almost all kinds of industry like chemical, petrochemical, aircraft, power, etc. Very often compressor is crucial part of installation and its failure can paralyze whole production and cause severe losses. This is why stable and safe working conditions are so important and require special attention. The existence of flow instabilities decreases safe working range and thus the machine is not working with maximum performance. This also can be counted as loss because machine does not use its full potential. Two most dangerous instabilities are rotating stall and surge and they are often reason of compressor failure. Rotating stall is less destructive phenomenon than surge and it is said to be frequent reason of surge onset, whereas surge can completely destroy machine in few seconds [1–3]. It is very dangerous phenomenon which is still not fully understood in centrifugal compressors and is avoided mainly by provision of large surge margin that strongly limits matching operational range.

1.1 Surge

Surge is flow instability occurring when mass flow is decreased below its critical value. Critical mass flow is different for all compressor rotational speeds and points placed on compressors performance for different speeds creates surge line (Fig. 1).

Surge appears as a pressure and mass flow fluctuations in axial direction. It is one-dimensional instability which affects compressor with pipeline system and it can be visualized as limit cycle oscillations on compressor map [5,6]. Typical deep surge limit cycle and pressure oscillation during surge are presented in Fig. 2 using dimensionless parameters. In this instability annulus-averaged mass flow is unsteady but circumferentially uniform. Sometimes oscillations are so big that reversed mass flow occurs. This is very dangerous because this oscillations can induce significant mechanical and thermal loads. Also, the frequency of oscillations can be close to resonance frequency of machine component. A distinction between...
different kinds of surge depending on fluctuations amplitude is often defined [7]:

1. Mild surge – no reverse flows occurs and the oscillations have frequency slightly less than Helmholtz’s frequency [4,5,8,9].
2. Classic surge – no reverse flows occurs, oscillations are bigger than in mild surge but frequency is lower.
3. Modified surge – rotating stall and surge mixed resulting in nonaxisymmetric, unsteady flow [9,10].
4. Deep surge – reverse flow are possible and has biggest fluctuations. Most dangerous of all types.

It is very important to predict surge onset for purpose of preventing centrifugal compressor from crossing surge line and potential failure. All compressors have defined surge margin which is located at a safe distance before surge line. If we can define surge onset more precisely then this margin can be closer to surge line and
as a result machine loses less of its operating range. Surge line can be determined by mathematical models. Since the surge discovery made by Emmons et al. [12] in 1955 many mathematical models have been developed. Some of them are presented in Tab. 1 [9].

<table>
<thead>
<tr>
<th>Model</th>
<th>Flow type</th>
<th>Compressor type</th>
<th>Flow instability</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hansen [16]</td>
<td>1-D Incom.</td>
<td>C</td>
<td>S</td>
</tr>
<tr>
<td>Fink [17]</td>
<td>1-D Incom.</td>
<td>C</td>
<td>S</td>
</tr>
<tr>
<td>Moore-Greitzer [18]</td>
<td>2-D Incom.</td>
<td>A</td>
<td>S, RS</td>
</tr>
<tr>
<td>Haynes [19]</td>
<td>2-D Incom.</td>
<td>A</td>
<td>S, RS</td>
</tr>
<tr>
<td>Com. – compressible</td>
<td>A – axial</td>
<td></td>
<td>S – surge</td>
</tr>
<tr>
<td>Incom – Incompressible</td>
<td>C – centrifugal</td>
<td></td>
<td>RS – rotating stall</td>
</tr>
</tbody>
</table>

1.2 Greitzer model

Most commonly used surge model is Greitzer one [15] developed in 1976 which has been proven by experiment the same year [20]. Despite the fact it has been developed for axial compressors it can be successfully used also in centrifugal compressors what has been proved in [11,16,17,21]. Still many antisurge systems are based on that model and it may be used to define surge margin.

This model is based on compressor scheme which is presented in Fig. 3. It consists of three parts: compressor, plenum, and throttle, and all lower subscripts \( c \), \( p \) and \( t \) corresponds to this parts, respectively. The compressing part of compressor installation is defined as a pipe with area \( A_c \) and length \( L_c \). Compression is accomplished by virtual piston generating pressure rise. Throttle is defined similarly but without piston – it is just pipe which can be closed on its end by decreasing outlet rea. In both mentioned about cases there is significant kinetic energy hence areas and lengths are selected in a way which guarantees dynamics similar to compressor and valve respectively. These pipes are connected by plenum which is big reservoir in which kinetic energy is neglected but has variable pressure \( p_p \). It is defined by its volume \( V_p \). Translating this scheme to real machine, compressor is rotor and diffuser, plenum is everything between diffuser
and throttle (so pipeline, volute, etc.).

Greitzer model also has its assumptions which are pointed below [7]:
1. incompressible flow within pipes, inviscid and one-dimensional, hence compressor mass flow $m_c$ and throttle mass flow $m_t$ are constant along the ducts;
2. isentropic compression in the plenum;
3. constant temperature in the whole compression system;
4. uniform pressure in the whole plenum;
5. valve behavior is quasi-static;
6. gravitational force is neglected.

Original Greitzer model consists of four equations but fourth is used only for axial machines so it will not be considered in this paper. The other three equations can be derived from basic momentum and energy conservation equations. Model operates with dimensionless parameters presented below:

mass flow
$$\Phi = \frac{\dot{m}}{\rho U_{tip} A_c},$$  

(1)

pressure rise
$$\Psi = \frac{\Delta p}{\frac{1}{2}U_{tip}^2},$$  

(2)

time
$$\dot{\hat{t}} = t \omega_h,$$  

(3)

where: $A_c$ - compressor pipe area, $\Delta p$ – pressure difference; $t$ – time, $U_{tip}$ – blade tip speed, $\rho$ – density, $\omega_h$ – Helmholtz frequency, $\dot{m}$ – mass flow.
Greitzer model equations defines:

mass flow in compressor which depends on pressure difference between compressor and plenum

\[ \frac{d\Phi_c}{dt} = B (\Psi_c - \Psi_p) , \]  

(4)

throttle mass flow

\[ \frac{d\Phi_t}{dt} = \frac{B}{G} (\Psi_p - \Psi_t) , \]  

(5)

and pressure rise in plenum depending on difference of mass flows from compressor and throttle

\[ \frac{d\Psi_p}{dt} = \frac{1}{B} (\Phi_c - \Phi_t) . \]  

(6)

Greitzer model has also two parameters: \( B \) and \( G \). \( B \) (parameter) is called stability parameter because its value determines if system is stable or not. Its physical interpretation is slope of compressor performance curve. Bigger \( B \) corresponds to bigger slope and thus less stable system

\[ B = \frac{U_{tip}}{2L_c\omega_h} , \]  

(7)

\[ G = \frac{A_c}{L_t A_t L_c} , \]  

(8)

where: \( A_t \) – throttle pipe area, \( L_c \) – compressor pipe length, \( L_t \) – throttle pipe length.

It is generally known that due to negligible throttle dynamics (which appears as very low value of parameter \( G \) because of very low throttle length \( L_t \)) Eq. (5) can be neglected. If parameter \( G \) is very low then according to this equation even very low pressure gradient induce big mass flow. This means that throttle follows system changes very. This is commonly used manner done for example in [6,11,22–27]. Therefore, because of lack of dynamic behavior in throttle it is assumed that pressure in plenum, \( \Psi_p \), is equal to throttle pressure, \( \Psi_t \).

1.3 Greitzer model drawbacks

It is clear that the Greitzer model is very simplified comparing to real machine. Some of assumptions are very inaccurate. For example neglecting kinetic energy in plenum is obviously significant misstatement. That is why some models enhance it by coping with mentioned misstatements. They mostly improve pipeline dynamics like in Yoon et al. model [24], Goyne and Allarie [26], Helvoirt and de Jager [22] or Helvoirt et al. [28].
Another significant drawback is ambiguous way of specifying Greitzer model parameters. It is hard to determinate $L_c$ and $A_c$ in way guarantying constant dynamics. Area of compressor can be set for example as outlet passage area and length is sometime approximated with length of blade or blade passage. According to [28] most commonly used method of determining this parameter is to iterate it and choose the one best suits to machine. But it is clear that the model depends on the whole system including pipeline, throttle etc. The method proposed by Helvoirt in [29] using approximate realizations is accurate but complicated.

1.4 Aim of study

The aim of this study is to present a simple and universal method of matching the Greitzer model simulation with a given compressor based on pressure signals. This solution provides extensive range of antisurge system adjustments that will make it efficient and reliable. This paper is not presenting new experimental results but is using one presented in [30–33].

2 Method

In this paper the results of experimental investigations of single stage centrifugal blower has been used to simulate centrifugal blower by two equation Greitzer model. Pressure signals of surge from centrifugal blower have been collected and analyzed in previous studies, and research results obtained on this machine considering surge onset was published in [30–33].

2.1 Experimental setup

Experimental investigations were performed on single stage centrifugal blower DP1.12 [30–33] which cross section is shown in Fig. 4.

Air entered blower by inlet pipe, A, of diameter $D_{in} = 300$ mm and then was accelerated in Witoszyński nozzle, B, [34] in purpose of aligning velocity profile on rotor, C, inlet. Downstream of rotor air entered parallel, vaneless diffuser, D. After diffuser air flew through circular volute, E, of variable radius $R_\theta$ changing from 5 mm to outlet pipe diameter $D_{out} = 150$ mm with radius increasing streamwise. Outlet pipe consisted of two straight elements of lengths 250 mm and 3750 mm and was connected by elbow. At the end of pipe the throttle was placed.

Motor which was driving blower was asynchronous AC motor (400 V/15 kVA). Blower design point was attained at nominal rotation frequency $f_{nom} = 120$ Hz, nominal mass flow $\dot{m}_{nom} = 0.8$ and nominal pressure ratio $\pi_{nom} = 1.12$ and was
Compressor modeling using Greitzer model validated by pressure.

Figure 4: Cross section of centrifugal blower DP1.12.

designed for ambient inlet conditions. In this experiment blower worked at lower conditions of rotational frequency \( f_{rot} = 100 \text{ Hz} \), mass flow \( \dot{m} = 0.75 \text{ kg/s} \) and pressure ratio \( \pi = 1.08 \) due to damage risk. At surge rotational speed yielded the impeller tip speed equal to \( U_{tip} = 103 \text{ m/s} \). All parameters presented in this section are summarized in Tab. 2.

Table 2: Parameters of the centrifugal blower DP1.12.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( D_{in} )</td>
<td>300 mm</td>
</tr>
<tr>
<td>( Z )</td>
<td>23</td>
</tr>
<tr>
<td>( D_0 )</td>
<td>476 mm</td>
</tr>
<tr>
<td>( \pi )</td>
<td>1.08</td>
</tr>
<tr>
<td>( D_{hub} )</td>
<td>86.3 mm</td>
</tr>
<tr>
<td>( \delta )</td>
<td>0.8 mm</td>
</tr>
<tr>
<td>( D_{out} )</td>
<td>150 mm</td>
</tr>
<tr>
<td>( U_{tip} )</td>
<td>103 m/s</td>
</tr>
<tr>
<td>( b_1 )</td>
<td>38.9 mm</td>
</tr>
<tr>
<td>( L_p )</td>
<td>154 mm</td>
</tr>
<tr>
<td>( R_b )</td>
<td>5 - 150 mm</td>
</tr>
<tr>
<td>( f_{nom} )</td>
<td>120 Hz</td>
</tr>
<tr>
<td>( \dot{m}_{nom} )</td>
<td>0.8</td>
</tr>
<tr>
<td>( b_2 )</td>
<td>14.5 mm</td>
</tr>
<tr>
<td>( A_{in} )</td>
<td>626 mm²</td>
</tr>
<tr>
<td>( f_{rot} )</td>
<td>100 Hz</td>
</tr>
<tr>
<td>( \dot{m} )</td>
<td>0.75 kg/s</td>
</tr>
<tr>
<td>( \pi_{nom} )</td>
<td>1.12</td>
</tr>
</tbody>
</table>

2.2 Signal gathering and analysis

Static pressure measurements were made at inlet, \( p_{s-in} \), and outlet, \( p_{s-out} \), of the compressor (subscript \( s \) corresponds to static pressure) with two pressure dynamic subminiature Kulite transducers, placed as it was shown in Fig. 5. A 100 kHz sampling rate was employed for data acquisition with a total acquisition time
20.97 s. An Iotech Wavebook 516/E acquisition system was used to capture the data. Every measurements consists 2097152 collected signal samples. hence total acquisition time – signal time corresponds to above 2000 rotor full revolutions.

![Figure 5: Cross section of centrifugal blower DP1.12 with pressure gauges positions.](image)

The pressure signal was gathered for different operation points both stable and unstable. It was achieved by decreasing the outlet area which was expressed in TOA (throttle opening area) parameter, varying between 0 and 100% (fully closed and fully open respectively). This parameter is proposed in [33] where also whole mass flow calculation is described. In this study also the relationship between TOA and mass flow, $\dot{m} = f(\text{TOA})$ is implemented according to Fig. 6 with equation of this curve presented on the right bottom corner of Fig. 6.

Based on this function the mass flow was designated and transformed to dimensionless parameter $\phi$. Parameter $A_c$ was set as passage outlet area and was equal to 676 mm$^2$ and air density $\rho_a = 1.168$ kg/m$^3$. Also pressure rise was transformed into dimensionless value $\psi$. Fragment of pressure signal from $\text{TOA} = 4.24\%$ is presented in Fig. 7. This signal was measured in the lowest mass flow conditions, so it is deepest surge condition obtained in the experiment, and its spectral analysis is presented. The frequency of strongest observed oscillations signal is 10.87 Hz, which gives $\omega = 68.31$ rad/s. Amplitude of signal obtained by spectral analysis is equal 0.6292 but as one can see the value of amplitude of real signal is bigger. It can be accounted to a fact that in real machine many factors have their impact on amplitude. Despite the fact that real signal seems to have bigger oscillations, the surge amplitude about 0.6292 was assumed. Surge frequency can be also detected when surge amplitude is not biggest from all components.
Compressor modeling using Greitzer model validated by pressure.

Figure 6: Curve used in mass flow calculation in TOA function.

Figure 7: Dimensionless pressure signal and its frequency spectrum-TOA 4.24%.

In frequency spectrum. In such situation the highest peak near predicted surge frequency is picked. It is noticeable that this oscillations have limit cycle character and are strongly noised. Mean value of signal is 0.9761. Same analysis and information was extracted for other operating points.
2.3 Compressor performance curve approximation

First step in modeling compressor is to approximate performance curve based on signals from different operation points. It is easy to plot points from stable work conditions because pressure signal is almost stable but different situation is when surge occurs and pressure signal is highly unstable. In this research in both situations mass flow rate was taken form curve presented in Fig. 6 and pressure rise was designated to be mean value of signal. In bigger machines assumption that in surge pressure oscillates around operating point can be a big mistake because in that kind of machines pressure mean value is above operating point. However in this research this assumption is acceptable. Experimental points from stable and unstable operation are shown in Fig. 8.

The most common method of compressor performance curve approximation is third order polynomial [6,7,11,24,35]. In this research it was impossible to fit all experimental points with this method, so two curves have been chosen. First one fits stable work points and this curve is called ‘right-sided’ approximation

$$\Psi_c(\Phi_c) = 1.06 + 0.0992 \left[ 1 + \frac{3}{2} \left( \frac{\Phi_c}{0.6736} - 1 \right) \right] - \frac{1}{2} \left( \frac{\Phi_c}{0.6736} - 1 \right)^3, \quad (9)$$

and second fits unstable points and is called ‘left-sided’ approximation

$$\Psi_c(\Phi_c) = 0.9 + 0.1792 \left[ 1 + \frac{3}{2} \left( \frac{\Phi_c}{0.6736} - 1 \right) \right] - \frac{1}{2} \left( \frac{\Phi_c}{0.6736} - 1 \right)^3, \quad (10)$$

Because none of those two curves fit all points hence new approximation is proposed, new is the fourth order polynomial which fits all points

$$\Psi_c(\Phi_c) = 0.1116\Phi_c^4 - 0.6654\Phi_c^3 + 0.9218\Phi_c^2 + 0.8524. \quad (11)$$

All three approximations are presented in Fig. 8.

2.4 Compressor modeling

The result of compressors operation simulation, using two equation Greitzer model, is signal similar to sinusoid. Three main quantities describing sinusoid are mean value, amplitude and frequency. Therefore to model compressor, simulation have to be adjust so as to match its quantities with experimental signal. This quantities are called criteria of simulation. The modeling process is presented by simulation one operation point for TOA = 4.24%.
Mean value criteria can be fulfilled by choosing proper compressor performance curve because of assumption that in surge pressure and mass flow oscillations are around operation point. Therefore the ‘right-sided’ approximation is neglected since its pressure value in operation point is much higher than in experimental signal. ‘Left-sided’ approximation has similar pressure to experimental signal but due to its deviations in stable operating region it is not correct also. Therefore fourth order polynomial is chosen as a compressor performance curve because it is most adequate to approximate experimental points.

Two other criteria, frequency and amplitude, can be received from spectral analysis done by fast Fourier transform (FFT). Frequency of highest amplitude obtained by simulation is considered as surge frequency and its value is received from spectral analysis. Amplitudes gained from FFT are sometimes understated due to spectral leakage but its real value can be estimated in way described later in this paper.

Criteria which determinate simulation agreement with experimental signal for TOA = 4.34% are summed up in Tab. 3.

In this paper centrifugal blower DP1.12 has been modeled using two equation Greitzer model, where two parameters need to be determined: parameter $B$ and Helmholtz frequency, $\omega_h$. Firstly, the impact of Helmholtz frequency and $B$ parameter on Greitzer model has been analyzed. Simulation for two different $\omega_h$ and for $B$ varied from 0.001 to 3.5 with 0.01 step has been made. In mild surge, which occurs in experiment, surge frequency of pressure oscillations is slightly

Figure 8: Experimental points and performance curve approximation.
Table 3: Summary of simulation criteria.

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Criteria value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean value</td>
<td>0.9761</td>
</tr>
<tr>
<td>Frequency</td>
<td>$\omega = 68.31 \text{ rad/s}$</td>
</tr>
<tr>
<td>Amplitude</td>
<td>$A = 0.6292$</td>
</tr>
</tbody>
</table>

less than the Helmholtz frequency. That is why first analyzed frequency was $\omega_h = 68.31 \text{ rad/s}$ (signal frequency). Second frequency was much higher to obtain significant differences between results and was 100 rad/s. After this analysis new optimized method of modeling has been developed and whole compressor in its unstable region has been modeled. However, to obtain simulations of different operation points throttle characteristic had to be determined, and in accordance with literature, it is mostly done by quadratic function $\Psi_t = k\Phi_t^2$ [7]. Because operation point is in cross of compressor performance curve with throttle characteristic some parameter $k$ has been introduced to achieve desired operation point. For presented example of TOA = 4.24% parameter $k$ is equal to 5.57. Simulation time was equal to experimental signal time, but sampling time was 10 kHz and the same sampling time has FFT.

3 Results

Figures 9 and 10 present results of two equation Greitzer model simulation. Analyzed variables in simulation was $B$ parameter and two Helmholtz frequencies. These figures presents impact of parameter $B$ and Helmholtz frequency, $\omega_h$ on Greitzer model simulation. In figures two criteria which were mentioned before are shown by horizontal lines, amplitude and frequency. It visualize at which value of $B$ parameter criteria is fulfilled. Points on beginning of Fig. 9 looks very chaotic. Amplitude connected with surge amplitude is very small (surge is just beginning) so FFT picked different frequency. Therefore region from 0 to critical value of $B$ parameter is neglected [28,33]. Critical $B$ parameter at which surge begins is signed as vertical line on both figures. Fluctuations of amplitude in Fig. 10 are because of spectral leakage which was mentioned before. In fact, real amplitude of simulation is obtained by connecting peaks of those fluctuations (marked as big points on fluctuations peaks).
4 Discussion

4.1 The impact of Helmholtz frequency

Impact of Helmholtz frequency on Greitzer model simulation is easily noticeable in Figs 9 and 10. It seems that Helmholtz frequency has impact on simulation frequency, moreover, one can see that amplitude is equal for both frequencies. It is noticeable that curve for $\omega_h = 100$ rad/s oscillation, generated by spectral...
leakage, are almost two times more frequent than for $\omega_h = 68.31$ rad/s. This results from almost two times higher of this frequency then $\omega_h = 68.31$ rad/s. Fact that Helmholtz frequency does not have impact on amplitude is very important observation for compressor modeling. Basing on this observation Helmholtz frequency now can be used to adjust simulation frequency without changing amplitude. Next two conclusions are based on results shown in Fig. 10. In this figure one can see that the higher Helmholtz frequency is the higher frequencies in the simulations. Last conclusion is that simulation never reaches assumed Helmholtz frequency no matter what parameter $B$ is approach, because compressor system is same as Helmholtz resonator only when throttle is fully closed (TOA = 0%). When it is even slightly open the frequency would decrease. In Fig. 9 both curves has step shape and that is the influence of FFT sampling.

In literature there is no information about the influence of Helmholtz frequency on Greitzer model simulation, but only values of it were presented [11,24,37]. Only in [26] it was mentioned to control the first resonance mode of surge. Conclusion that simulation never reaches assumed Helmholtz frequency is confirmed also in [4,5,8,9]. As one can see neither of analyzed Helmholtz frequencies are chosen correctly but after observations of Helmholtz frequency impact it could be expected that the assumed Helmholtz frequency should be slightly higher than 68.31 rad/s.

4.2 The impact of $B$ parameter

Parameter $B$ was variable in simulation of both applied Helmholtz frequencies. Its influence on Greitzer model simulation is clearly visible in Figs. 9 and 10. Influence of B parameter on amplitude is presented in Fig. 10. In the beginning of surge with rise of $B$ parameter amplitude is sharply rising up to its maximum, and then amplitude start slightly decreasing. Situation is similar in Fig. 9. At the beginning frequency grows a little with $B$, but after reaching maximum it slightly decreases. It is important $B$ parameter has impact on both, amplitude and frequency, hence is most influential parameter and most important in compressor modeling.

In literature impact of $B$ parameter is said to be well known but only in [38] it is said that $B$ has relevant impact on amplitude, while influence on frequency is neglected. There is no value of parameter $B$ in which frequency criteria is fulfilled but for amplitude criteria $B$ which satisfy it is 0.7813 or approximately 1.93 but in this second point frequency is much different than required.
4.3 Method of compressor modeling

New method of compressor modeling has been formulated on the base of conclusion made in previous section. First step is to approximate the compressor performance curve and throttle characteristic for current operation point. This operation will fulfill mean value criteria if we assume that surge oscillations fluctuates around operation point. Because Helmholtz frequency does not change amplitude so at the beginning it can be set up as equal to experimental signal frequency. To fulfill amplitude criteria $B$ parameter must be determined, therefore it is set up to be equal to 0 or equal to critical value to minimize number of iterations. Then by iterations $B$ is increased until amplitude of simulation is equal to signal amplitude. After that by increasing Helmholtz frequency, frequency of simulation is adjust until it fulfill frequency criteria. After this procedure one operation point of compressor is modeled, however, to model another operation point throttle constant, $k$, should be changed. This method can be summing up in few consecutive steps:

- Approximate compressor performance curve $\Psi_c(\Psi_c)$ and throttle characteristic $\Psi_t(\Psi_t)$.
- Determine Helmholtz frequency, $\omega_h$, to be equal to experimental signal frequency.
- Set $B$ parameter to be equal to 0 or critical value of $B$ parameter.
- Increase $B$ until amplitude of simulation is equal to experimental signal amplitude.
- Increase $\omega_h$ until frequency of simulation is equal to experimental signal frequency.
- Change throttle constant, $k$, and model another operation point.

By this method 16 points of unstable operation have been modeled. Results of this modeling are presented below. Precision of the method was measured as relative errors of amplitude and frequency. In Fig. 11 errors for ten points of unstable operation are shown. One can see that errors are very small and that leads to conclusion that presented method is precise and can be used in compressor modeling. In Fig. 12 parameters $B$ and Helmholtz frequencies obtained for different TOA (different operation points) are shown. Values of $B$ and Helmholtz frequency does not vary in wide range and the differences between next values are small. It suggests that is possible that $B$ is constant what corresponds well with theory [11,15].
5 Conclusions

The influence of parameter $B$ and Helmholtz frequency on two equation Greitzer model was presented. After analysis of this impact, a new model of compressor modeling based on experimental signal of pressure oscillations was proposed and verified.

Two different Helmholtz frequencies were assumed and simulation performed for $B$ parameter varying from 0.001 to 3.5. After this simulations following conclusions were learned Helmholtz frequency impact:
• Helmholtz frequency has no impact on simulation amplitude;
• the higher Helmholtz frequency leads to the higher simulation frequency is;
• simulation frequency is always lower than Helmholtz frequency.
and $B$ parameter impact:
• $B$ parameter has impact on both, frequency and amplitude;
• with increasing parameter $B$, amplitude sharply rise until it reach peak and then slightly falls. It has the same impact on simulation frequency;
• parameter $B$ is stability parameter and after crossing critical value surge appears.

Based on this conclusions new method of compressor modeling was proposed and verified. Value of biggest error was approximately 5.5% for amplitude and 0.26% for frequency. Therefore this method is very accurate and can be used to model a compressor. It is also much simpler than traditional modeling with Greitzer model. In this method only $A_c$ (pipe across area) parameter is needed but it is ease to designate by outlet surface from rotor. In future it can be used in designing anti surge systems what is general purpose of understanding surge. It is hard to predict where is the limit of this method.

Received in July 2016

References


Compressor modeling using Greitzer model validated by pressure. . . 89


ISSN 0079-3205 Transactions IFFM 133(2016) 69–89
Mathematical surge modeling based on the pressure oscillations in the stable operation range of the compressor

Institute of Turbomachinery, Lodz University of Technology, 219/223 Wólczańska, 90-924 Łódź, Poland

Abstract
Prevention of surge phenomenon is a very active field of research in last decades. The Greitzer model is the most widely applied mathematical model describing the surge phenomenon. Two parameters that characterize the given compressing system: the Helmholtz oscillator frequency and the so-called B parameter have to be given as the input to Greitzer model. Although those parameters are easy to determine for simplified compressor models, they are extremely hard to predict in case of real industrial compressors. Moreover, in most cases it is impossible to analyse compressor unstable work, which makes this prediction even more speculative. Therefore the method that determines the parameters basing on the compressor stable operation is indispensable. In paper the regularly perturbed Greitzer model based method of predicting the behaviour of a compressor in the unstable operation basing on signals from its stable operation is proposed and discussed.

Keywords: Radial compressors; Surge; Greitzer model

Nomenclature

\begin{align*}
  a_u & \quad \text{speed of sound} \\
  f_h & \quad \text{Helmholtz frequency} \\
  k & \quad \text{throttle constant} \\
  L_c & \quad \text{length of compressor}
\end{align*}

*Corresponding Author. Email adress: andrzej.jaeschke@p.lodz.pl

ISSN 0079-3205 Transactions IFFM 133(2016) 91–108
1 Introduction

1.1 Unstable phenomena in centrifugal compressing units

When a centrifugal compressor is operating at given rotational speed and the mass flow rate is reduced beyond a certain critical value, the operation of the compressor is no longer stable. As the instabilities can cause a failure of compressor which can cause significant financial losses, it is very important to accurately predict the point where instabilities are likely to occur. Unfortunately, the mechanisms that are responsible for creation and behaviour of instabilities are very complex which makes the prediction difficult. Emmons et al. [1] was the first to identify and analyse the nonstable phenomena in the centrifugal compressors. Two main unstable phenomena have been identified: surge and rotating stall. Surge refers to global flow fluctuations in the axial direction. Rotating stall is a phenomenon of the formation of vortex structures which are stationary in a certain rotating frame of reference, so-called stall cells [2]. Those states can be identified as forms of natural fluid oscillation. In stable operation they are fully suppressed, while in the unstable region the damping is not strong enough and rotating stall or surge are likely to appear [3,4].

1.2 Greitzer model of surge phenomenon

In 1976 Greitzer introduced the mathematical model of unstable behaviour of compressing units [5] and confirmed it by experiment [6]. Later a development towards description of the shape of fully-formed transients has been made by
Moore and Greitzer [7,8]. Although Greitzer introduced his model for axial compressors it was also proven to be correct for centrifugal machines [9–12].

Greitzer modeled the compressing system as three interacting elements: compressor, plenum and throttle. The compressing system simplified by Greitzer is presented in Fig. 1. Compressor and throttle are characterized by their lengths $L_c$ and $L_t$, and areas $S_c$ and $S_t$, respectively. Plenum is characterized by its volume $V_p$. Pressure $p$ at each component and mass flow rates ($\dot{m}_c$ and $\dot{m}_t$) at compressor and throttle are the variables. The subscripts $c$, $t$ and $t$ indicate components of the compressing system, respectively, compressor, plenum, and throttle. Additional model assumptions are listed below [13,14]:

- flow within pipes is incompressible, inviscid and one dimensional,
- compression in plenum is isentropic,
- temperature is constant in the whole system,
- pressure in plenum is uniform,
- valve is quasi-static,
- gravity forces are neglected.

![Figure 1: Compressing system in Greitzer model [13].](image)

Greitzer model consists of system of four ordinary differential equations, where first three are derived from conservation of mass and momentum for three components of the system. Fourth equation describes the effect of rotating stall on the system. In case of centrifugal units behaviour of rotating stall phenomena is far too complicated to be accurately approximated by one simple ordinary differential equation. Therefore, for case of centrifugal compressors it is a general practice not to use this equation and use Greitzer model for simulation of only surge phenomenon (not considering rotating stall). The Greitzer model for centrifugal units
in the nondimensional form reads:

\[ \frac{d\Phi_c}{dt} = B(\Psi_c - \Psi_p) \tag{1a} \]
\[ \frac{d\Phi_t}{dt} = \frac{B}{G}(\Psi_p - \Psi_t) \tag{1b} \]
\[ \frac{d\Psi_p}{dt} = \frac{1}{B}(\Phi_c - \Phi_t) \tag{1c} \]

where dimensionless mass flow rate coefficient \( \Phi = \frac{\dot{m}}{\rho a U_{tip} S_c} \) with index corresponding to component of the system, dimensionless pressure rise coefficient \( \Psi = \frac{2\Delta p}{\rho a U_{tip}^2} \) with index corresponding to component of the system, dimensionless time coefficient \( \hat{t} = \frac{t f_h}{2\pi} \), model parameters \( B = \frac{U_{tip}}{L_c} \) and \( G = \frac{S_c L_t}{S_t L_c} \), where \( \rho_a \) denotes gas density, \( U_{tip} \) – impeller tip speed, \( \Delta p \) – difference of outlet and inlet pressure of component, \( f_h \) is the Helmholtz resonator frequency, given by formula

\[ f_h = \frac{1}{2\pi a_a} \sqrt{\frac{S_c}{V_p L_c}} \tag{2} \]

with \( a_a \) denoting the speed of sound.

Although the Greitzer model is very simplified and has many questionable assumptions it has been proven to be very efficient tool for surge modeling. Greitzer model is also a base for development of more complicated mathematical surge models [15–17]. Although there were some other attempts to formulate mathematical surge models in the past 40 years, the Greitzer model is still the one most often used [18].

The Greitzer model has one well-known, severe limitation. The parameters: \( B, G \) and \( f_h \) are very hard to be estimated using only the geometry of the compressing system. Although the meaning of geometrical parameters (e.g., \( L_c \)) is clear on the level of Greitzer’s simplified compressing system, it is vague for real compressing units. This significantly limits the applicability of the method.

A common practice is to remove the second equation describing the throttle dynamics from the model. This can be done assuming that the length of throttle is negligible and therefore value of \( G \) is small, which implies very fast response of throttle. As the time of response is infinitesimal, the second equation can be omitted. This reduces the set of problematic parameters to \( B \) and \( f_h \) [15]. The necessary parameters were successfully estimated using pressure signals from the surge region [12,19]. Unfortunately, most of machines can not operate in unstable region, so another methodology of estimation of Greitzer model parameters is still to be developed.
1.3 Aim of study

The Greitzer model was proven to be a suitable tool for analysis and prediction of dangerous surge phenomenon in centrifugal compressors. Unfortunately, it is hard to evaluate the necessary parameters for the model using only a geometry of unit. It was shown that it is possible to find those parameters having the pressure signal from the unstable operation, but many machines cannot be operated within the unstable region for safety reasons. Therefore, the authors of this article decided to focus on investigation of possible methods of evaluation of necessary parameters using only pressure signals from stable operation of centrifugal compressor. To realise it the perturbed Greitzer model approach was proposed and compared with previously published experimental signals.

2 Method

2.1 Experimental rig

For validation of proposed mathematical model the experimental results obtained by Liskiewicz et al. have been used in [20] as a mean. In the considered research a single stage centrifugal blower DP1.12 was investigated. There were several tests performed on this machine concerning its unstable operation [13,20–22]. The cross-section and main dimensions of the blower are presented in Fig. 2. The air entered the rig through the inlet pipe (A). Then the flow was accelerated in the Witoszynski nozzle (B) [23]. Downstream the nozzle there is the impeller (C). From impeller the flow was entering a vane-less diffuser (D) from where it entered a circular volute (E). A throttling valve was placed at the end of the outlet pipe. The most important dimensions of the test rig are presented in Tab. 1.

The rotor was driven by an asynchronous AC motor (400V/15 kVA). The blower was designed for operation at ambient conditions. The design point of the blower was at mass flow rate \( \dot{m} = 0.8 \) kg/s and pressure ratio \( PR = 1.12 \). In this study to avoid impeller damage the blower was operated at lower rotational speed of \( f_{rot} = 100 \) Hz, with nominal mass flow rate \( \dot{m}_n = 0.75 \) kg/s and pressure ratio (the ratio of pressure after compression to the pressure at the inlet of the compressor) \( PR = 1.08 \). Obtained impeller tip speed was equal to \( U_{tip} = 103 \) m/s. The impeller had \( z = 23 \) blades which corresponds to blade passing frequency of \( f_{BP} = 2.3 \) kHz. The volume of outlet pipe was equal to \( V_{out} = 0.0968 \) m\(^3\) that in this setting corresponded to the Helmholtz frequency of \( f_H = 11 \) Hz [12].

The test stand was equipped with two dynamic subminiature Kulite transducers connected to Iotech Wavebook 516/E data acquisition system. One transducer...
Figure 2: Cross-section of the blower used in the study.

Table 1: Table of most important dimensions of the test rig.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Notation</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet pipe diameter</td>
<td>$D_{in}$</td>
<td>300 mm</td>
</tr>
<tr>
<td>Rotor inlet diameter at hub</td>
<td>$D_{1hub}$</td>
<td>86.3 mm</td>
</tr>
<tr>
<td>Rotor inlet span</td>
<td>$b_1$</td>
<td>38.9 mm</td>
</tr>
<tr>
<td>Rotor outlet diameter</td>
<td>$D_2$</td>
<td>330 mm</td>
</tr>
<tr>
<td>Rotor outlet span</td>
<td>$b_2$</td>
<td>14.5 mm</td>
</tr>
<tr>
<td>Blade tip gap (constant along blade)</td>
<td>$\delta$</td>
<td>0.8 mm</td>
</tr>
<tr>
<td>Diffuser outlet diameter</td>
<td>$D_3$</td>
<td>476 mm</td>
</tr>
<tr>
<td>Volute tongue gap</td>
<td>$\delta_g$</td>
<td>5 mm</td>
</tr>
<tr>
<td>Diameter of outlet pipe / volute outlet</td>
<td>$D_{out}$</td>
<td>150 mm</td>
</tr>
<tr>
<td>Length of the outlet pipe</td>
<td>$l$</td>
<td>5.5 m</td>
</tr>
<tr>
<td>Volume of the outlet pipe</td>
<td>$V_{out}$</td>
<td>0.0968 m$^3$</td>
</tr>
</tbody>
</table>

($P_{s-in}$) was placed at the inlet of the rig and the other ($P_{s-out}$) at the volute outlet. Placement of the gauges is presented in Fig. 3. Sampling frequency was equal to 100 kHz. The measurements were done over 20.97 s ($2^{21}$ samples).

Position of the throttling valve placed at the end of outlet pipe was described by dimensionless throttle opening area parameter (TOA). TOA = 0% corresponds
to fully closed valve, while TOA = 100% corresponds to fully open valve.

2.2 Compressor modeling with perturbed Greitzer model

The operation of the blower is simulated using the Greitzer model (1) with the second equation omitted. With simplification $\Psi_p = \Psi_t$ resulting from assumption of instant response of the throttle this model reads:

$$\frac{d\Phi_c}{dt} = B(\Psi_c(\Phi_c) - \Psi_t),$$  \hspace{1cm} (3a)

$$\frac{d\Psi_t}{dt} = \frac{1}{B}(\Phi_c - \Phi_t(\Psi_t)).$$  \hspace{1cm} (3b)

In the formulas above it was indicated that $\Psi_c$ is a function of $\Phi_c$. This dependency is given by performance curve of compressor at given rotational speed. It was also shown that $\Phi_t$ depends on $\Psi_t$. This dependency is given by the reverse throttle curve. Performance curve of the analysed compressor is according to [19] given by the fourth order polynomial

$$\Psi_c(\Phi_c) = 0.1116\Phi_c^4 - 0.6654\Phi_c^3 + 0.9218\Phi_c^2 + 0.8524.$$  \hspace{1cm} (4)

The throttle curve is given by quadratic function [14]

$$\Psi_t(\Phi_t) = k\Phi_t^2,$$ \hspace{1cm} (5)
where the throttle constant $k$ was specified in each case to achieve given point of operation. The point of operation was computed from the compressor performance curve knowing the value of mass flow rate attained at the experimental test stand. The values of mass flow rates for given values of TOA coefficient were computed using empirical relation

$$\dot{m} = 0.0943 \text{TOA}^{0.5758}. \quad (6)$$

The coordinates of point of operation were used as the initial conditions in numerical model.

Obviously, in case of operation in the stable region the Greitzer model is not producing any oscillations. In the real machine, however there are many perturbances pushing the compressor away from its operating point. They are damped in time (as the compressor operates in the stable range) but their presence and the process of their damping (which is not immediate) creates some frequency/amplitude pattern. To capture a trace of this behaviour in the stable region with Greitzer model, an artificial perturbance was added to the pressure value with constant frequency. It was decided to use frequency of $f_p = 100$ Hz which corresponds to frequency of rotation of the rotor and magnitude of perturbance $\delta_p = 0.01$. The magnitude is of the same order as the perturbances observed in the oscillation’s spectrum of this machine in [20]. Technically, the perturbing was done by dividing the process of solving of the system of ordinary differential equations (3) in time into subproblems of length 0.01 s and perturbing final time solution of previous subproblem before using it as the initial condition for the next one.

In case of both numerical and experimental results the fast Fourier transform (FFT) algorithm [24] was used to allow the spectral analysis of pressure oscillations. Entire analysis was performer on the nondimensional coefficients.

3 Results

3.1 Numerical results

The perturbed Greitzer model was used for simulation of compressor operation at TOA = 4% (deep surge) and TOA = 50% (stable operation). The Greitzer model input parameters were taken from [19] where they were calculated to best fit the pressure signals from deep surge the values were $B = 0.725$ and $f_h = 11.43$ Hz. It should be noted that this value is slightly different than one obtained analytically according to [12]. At TOA = 4% the compressor is expected to work in highly
unstable region, while TOA = 50% corresponds to fully stable operation. Results after the Fourier transform done by the means of the FFT algorithm are presented in Figs. 4 and 5. In the unstable operation a large peak close to the Helmholtz frequency can be observed, interestingly there is a slight amplification in the vicinity of this value also in the stable operation. Therefore, the investigation of positioning in the spectrum of the maximum oscillation amplitude and its value for different values of TOA was performed. To avoid finding a maximum oscillations at 100 Hz that corresponded to added perturbation the search interval was limited to frequency \( f < 90 \text{ Hz} \). Figures 6 and 7 present the behaviour of the value and positioning in the spectrum of the maximum oscillation amplitude for variable values of \( B \) while Figs. 8 and 9 depict the situation in case of variable \( f_h \). Lines marked with X in Figs. 6–9 correspond to \( B = 0.7 \) and \( f_h = 11 \) which are close to the values applied by Grapow in [19].

![Figure 4: The frequency spectrum of perturbed Greitzer model with \( B = 0.725, f_h = 11.43 \text{ Hz}, \) and TOA = 4% obtained by means of FFT algorithm (A – amplitude, Fs – frequency).](image)

### 3.2 Experimental results

Figures 10 and 11 present the non-dimensionalized experimental signals for TOA = 4% and TOA = 50%, respectively, after application of the FFT algorithm. Figure 12 presents the values of the largest amplitudes observed in experiment for variable TOA. In Fig. 13 the positioning of maximum oscillation amplitude versus the values of TOA is presented together with the general trend after averaging the result on span of 10 values. To avoid strong noises at higher frequencies, the
Figure 5: The frequency spectrum of perturbed Greitzer model with $B = 0.725$, $f_h = 11.43$ Hz, and TOA = 50% obtained by means of FFT algorithm (A – amplitude, Fs – frequency).

Figure 6: The largest observed amplitude versus TOA for $f_h = 11.43$ Hz and different values of $B$.

maximum oscillations positioning was searched only in the region $9 \text{ Hz} < f < 20 \text{ Hz}$. Results presented in Figs. 12 and 13 are compared with corresponding numerical results (for $B = 0.725$ and $f_h = 11.43$ Hz).
4 Discussion

4.1 Investigation of the model

Application of the perturbed Greitzer model to the case of operation in the unstable region gave expected results: there was a large peak at the frequency slightly lower than the one corresponding to assumed $f_h$. For larger value of TOA corresponding to stable operation there was also a small amplification present at the slightly higher frequency. This was caused by the fact that according to the Greitzer model, even for very stable systems the damping of oscillations is not immediate, so for the regular perturbation it left a track in the oscillation spectrum. Similar effect was observed in experimental study [20]. Importantly, the amplitude values were of significantly lower magnitude for stable operation than in case of unstable operation. The peak in both spectra at $f = 100$ Hz was caused by the regular perturbation itself. It was also followed by its harmonics.

The further analysis of positioning of the maximum amplitude in the spectrum was performed. It was shown that the frequency is decreasing in the region of low TOA values and then it starts to significantly increase with increasing TOA. This behaviour can be very useful for determining the $f_h$ parameter using pressure signals from stable operation of compressor. It was also observed that the
Figure 8: The largest observed amplitude versus TOA for \( B = 0.725 \) and different values of \( \omega_h \), \( (\omega_h = 2\pi f_h) \).

Figure 9: The frequency corresponding to the largest observed amplitude versus TOA for \( B = 0.725 \) and different values of \( \omega_h \) \( (\omega_h = 2\pi f_h) \).

values of the largest amplitudes were high at low values of TOA and then were steeply decreasing in certain region followed by zone of slow steady decreasing.
Mathematical surge modeling based on the pressure oscillations.

The highest amplitudes observed in Fig. 6 correspond to the surge phenomenon. In this region the damping forces are not strong enough to prevent the pressure oscillation in the compressor, so amplitudes are significant. The region of steep decrease of amplitude can be regarded as the border between unstable and sta-

Figure 10: The experimental signal for TOA = 4%, after FFT (A – amplitude, Fs – frequency).

Figure 11: The experimental signal for TOA = 50%, after FFT (A – amplitude, Fs – frequency).
Figure 12: The largest amplitude versus TOA obtained from experiment compared with corresponding numerical results.

Figure 13: The frequency corresponding to largest amplitude versus TOA obtained from experiment. Line marked with O visualises a general trend computed by averaging the original result (line marked with X) on span of 10 samples. Experimental results are compared with corresponding numerical results.

able regions of the compressor. Further, slower decrease is caused by increase of damping forces that diminish the influence of perturbations.

Further analysis was devoted to the influence of Greitzer model parameters $B$.
and $f_h$ on the dependencies discussed above. Knowledge of their influence would allow to estimate values of $B$ and $f_h$ by finding the best fit with the curves from experimental pressure signals.

One can easily notice that the higher the value of $B$ the wider the surge zone. This is closely connected with the stability limits discussed in details in [13]. The influence on the frequency at which the maximal amplitude occurs is not clear, so no direct conclusions could be made. The estimation of the value of $B$ parameter based on the stable work of compressor should be an object of further investigation.

Although the influence of $f_h$ on the curves of value of maximal amplitude versus TOA is not significant, one can notice that it strongly influence a positioning of the largest amplitude in the spectrum. With the good accuracy one could state that it only translates the curve vertically. This is important observation as it is enough to know the nature of curve and the frequency at which we observe the largest oscillations at any point of operation to estimate the value of $f_h$. Although the nature of the curve depends strongly on $B$ parameter, in the operation at TOA $< 50\%$ this dependency is weaker, which makes it possible to know the approximate curve shape without prior estimation of $B$.

The possibility of estimation of Greitzer model parameters by comparison with pressure signals from stable operation of compressor is very important for realisation of active antisurge systems based on Greitzer model. Such systems were proposed by Willems [9,25] and Grapow [19], but their applicability was questionable due to limitations of accuracy of model parameters.

4.2 Comparison with experimental results

At TOA $= 4\%$ the frequency at which the peak of oscillation in the numerical result from the unstable operation occurred, matches perfectly the one from experimental signal spectrum. It was expected as the parameters in Greitzer model were chosen such that, they match the amplitude and frequency of oscillation in the deep surge [19]. In the experimental data in stable regime there exist a wide dispersed peak similar to the one shown in numerical results. It occurs, however, at higher frequency.

Similar analysis of the value of the largest amplitude and frequency corresponding to it was done as in case of numerical results. In the experimental data the unstable region was wider than in corresponding numerical results. This can be caused by using two equations model which does not take into account rotating stall which can be the cause of oscillations at the interface between stable and surge regions. The errors in estimation of compressor performance curve, throttle
constant or assumed Greitzer model parameters, could have the other causes. The analysis of frequency of the largest amplitude versus TOA was complex due to very strong noise in the signal and relatively weak dependency which was sought. The interval of frequencies in which the maximal amplitude was searched was trimmed to only 9 Hz < f < 20 Hz. The obtained results, however very noisy, after averaging on span of 10 samples gave general trend similar to this obtained form numerical analysis. The need of trimming and very noisy experimental results question the applicability of perturbed Greitzer model to estimation of $f_h$ and $B$. Therefore smoothing of the signal or modifying the measurement methodology should be the object of further works in this topic.

5 Summary

In this paper the regularly perturbed Greitzer model was proposed. It can be used to obtain the oscillations spectra even for stable operation of compressor. Comparing them with experimental signals gives the possibility to estimate and adjust the parameters in the Greitzer model for the given system without entering the unstable operation.

The analysis of the results from this method showed that estimation of the Greitzer model parameters is possible. It could be realized by optimization of the parameter values to achieve the best fit of the resulting curve of positioning of the highest amplitude in spectrum for different values of TOA with experimentally obtained one.

The experimental results confirmed the general trends obtained from numerical analysis but before the unknown parameters could be estimated the effort should be made to improve the signal smoothing method or measurement methodology.

Received in July 2016

References


ISSN 0079-3205 Transactions IFFM 133(2016) 91–108


Marcin Lackowski*, Helena Nowakowska

Electric field influence on a height of rise of a dielectric liquid between two parallel electrodes

Institute of Fluid Flow Machinery, Polish Academy of Sciences, 80-231 Gdańsk, Fiszera 14, Poland

Abstract
A differential equation for temporal dependence of the liquid height of rise in a microchannel consisted of two parallel plates has been modified to account for a dielectrophoretic force, which is a result of an electric field occurring between the plates. The equation takes into account viscous, surface tension, gravitational forces, capillary entrance pressure and dielectrophoretic force. Numerical calculations have been performed for 2-propanol as a liquid. Time dependence of the height of rise and the velocity of liquid front has been obtained for two cases of the initial liquid height and several values of a voltage applied to the plates.

Keywords: Dielectrophoretic force; Microchannel flow controller

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>half of the electrode distance, m</td>
</tr>
<tr>
<td>E</td>
<td>electric field, V/m</td>
</tr>
<tr>
<td>g</td>
<td>gravitational acceleration, m/s²</td>
</tr>
<tr>
<td>h</td>
<td>fluid height of rise, m</td>
</tr>
<tr>
<td>p</td>
<td>pressure, Pa</td>
</tr>
<tr>
<td>t</td>
<td>time, s</td>
</tr>
<tr>
<td>V</td>
<td>voltage, V</td>
</tr>
<tr>
<td>W</td>
<td>half of the electrode length, m</td>
</tr>
</tbody>
</table>

*Corresponding Author. Email adress: marcin.lackowski@imp.gda.pl

ISSN 0079-3205 Transactions IFFM 133(2016) 109-115
1 Introduction

Understanding and controlling the fluid-flow at microscale is of great importance due to its growing range of applications nowadays, among others in medicine, bio-engineering, and chemistry. Microfluidic devices are also used for heat transfer in microelectronic systems as well as in refrigeration, cooling, and organic Rankine cycle (ORC) systems. To enhance a capillary-force driven flow that is characteristic for microfluidic devices, the electric field is often employed to induce electrowetting, electrocapillarity or dielectrophoretic (DEP) effects [1].

A microchannel flow controller based on DEP effect has been proposed lately [2]. The pressure-driven flow of a liquid in a channel with rectangular cross-section is enhanced by capillary and DEP forces in this device. It was shown that by changing the electric voltage applied to the electrodes placed on vertical walls of the channel, the liquid flow rate can be effectively regulated. However, an optimization of this device, particularly with the goal to control its time-response properties, is required because the controller is meant as a part of a feed-back system for temperature regulation. To this end, knowledge of the influence of the forces on the liquid height of rise is necessary, because its value is correlated with the liquid flow rate in the controller [2–4].

A time evolution of the height of rise for capillary driven flow was analyzed theoretically first for tubes of circular cross-sections [5]. Classical Lucas-Washburn equation takes into account viscous, surface tension, and gravitational forces. More advanced models include also inertial force and entrance pressure loss effects [6,7]. Newer papers take into consideration parallel-plate and rectangular capillaries [8,9] as well. A generalized theoretical analysis [8] gives results which agree very well with experimental results for different geometries, dimensions and liquids.

Despite the fact that the effect of the capillary rise seems a very well understood issue, to our knowledge, there is no an analysis on the temporal evolution of the height of rise for the case when DEP forces exist together with capillary forces. Papers that deals with the electric field effects on the capillary dynamics

The goal of the presented paper is to perform theoretical and numerical analysis of the influence of the DEP forces on the temporal dependence of the liquid height of rise in capillary system consists of two parallel plates constituting two electrodes.

2 Mathematical model for capillary flow

The presented model is based on that described in [9], where the temporal dependence of the height of rise, called there the penetration depth, \( h(t) \), for two geometries of the capillary, namely parallel plates and circular tube, is considered. The model [9] starts from the integral momentum equation for homogeneous, incompressible, Newtonian fluid and takes into account the following forces acting on the control deformable fluid volume: the capillary force, viscous force, the gravitational force, and the pressure force at the inlet to the channel. The latter force – not taken into account in early analyses [5,6] – is obtained based on [12], where it is assumed that an infinite reservoir represented by a hemispherical control volume is placed at the capillary entrance. From the momentum balance in the control volume, the fluid pressure at the entrance can be found. Other shapes of the control volume have been also considered, but their influence is not significant for the case when channel length is much greater than the width [7].

In the present study, when the capillary with a dielectric fluid is placed in the electric field, the DEP force [13] is added to the momentum balance equation. Taking into account this force leads to a modification of the differential equation for a time dependence of the height-of-rise presented in [9]. For a channel consisted of two rectangular electrodes separated by the distance \( 2B \) and of the length \( 2W \) this equation takes the form

\[
(h + 1.11\sqrt{BW})\frac{d^2h}{dt^2} + 0.958 \left( \frac{dh}{dt} \right)^2 + \left( \frac{3}{B^2} h + \frac{1.772}{\sqrt{BW}} \right) \frac{\mu}{\rho} \left( \frac{dh}{dt} \right) + gh - \frac{\sigma}{\rho} \left( -\frac{\cos \theta}{B} - \frac{1}{W} \right) = \frac{p_{DEP}}{\rho} = 0 ,
\]

(1)

where \( \mu \) and \( \rho \) are the dynamic viscosity and the density of the liquid, respectively, \( \sigma \) is the surface tension of the liquid in contact with air, \( \theta \) is the dynamic contact angle of the air-liquid-electrode interface, and \( g \) is the gravitational acceleration. The pressure \( p_{DEP} \) follows from the DEP force acting on the liquid surface and
according to [13] can be expressed as

\[
p_{\text{DEP}} = \frac{\varepsilon_0 (\varepsilon_r - 1) V^2}{8B^2},
\]

(2)

where \(\varepsilon_0\) is the electric permittivity of vacuum, \(\varepsilon_r\) is the relative permittivity of the liquid, and \(V\) is the electric potential between the electrodes, related to the electric field \(E\) by the relation \(E = V/(2B)\). It is worth to note that the numerical coefficients in Eq. (1) are the result of theoretical analysis and are not taken from experiments.

3 Results

Equation (1), which is a nonlinear nonhomogeneous second order equation, was solved using commercial COMSOL ver. 5.1 software [14] employing its Global ODEs and DAEs interface. To validate the calculation method, the results were compared with that from [9] for different liquids, where no electric field was applied. The results agreed perfectly for all cases.

The calculations were performed for the same liquid that was used in the microchannel described in [2,3], namely 2-propanol. Its physical properties were taken assuming temperature 25°C and are as follows [15]: the density 786 kg/m³, surface tension 20.9 mN/m, dynamic viscosity 2.34 mPa.s, relative electric permittivity 19.9 and the contact angle 0 rad. The channel dimensions were: \(2B = 3 \times 10^{-4}\) m, \(2W = 5 \times 10^{-3}\) m.

Two cases were analyzed: (i) the initial level of liquid is the same as in the external reservoir, \(h(0) = 0\), and (ii) the initial level corresponds to the level for the stationary equilibrium of all forces except of the DEP force, \(h(0) = h_{\text{cap}}\). This value can be found from

\[
h_{\text{cap}} = \frac{\sigma}{\rho g} \left( \frac{\cos \theta}{B} - \frac{1}{W} \right)
\]

(3)

and for the selected parameters is equal to about \(1.71 \times 10^{-2}\) m.

Figure 1 presents how the height of rise changes with the time for different applied voltages. It is seen that the height of rise increases with the applied voltage and after about 5 s reaches the equilibrium level of rise. This time is smaller for lower voltage. The velocity of the liquid front is shown in Fig. 2. In case (i), the velocity reaches almost 0.5 m/s for \(V = 500\) V in the initial stage of rising because both forces capillary and DEP act on the liquid surface. In case (ii) the velocity is about ten times smaller for the same case because only DEP force causes the liquid flow.
The presented results show that changing the voltage applied to the electrodes it is possible to effectively modify the liquid level height of rise. The voltage above 100 V should be applied to achieve a noticeable effect whereas for voltage 300 V the level rises about 50%.

Figure 1: Temporal dependence of the height of rise for different voltage applied to the electrodes (a) $h(0) = 0$ and (b) $h(0) = h_{cap}$.

Figure 2: Temporal dependence of the velocity of liquid front for different voltage applied to the electrodes (a) $h(0) = 0$ and (b) $h(0) = h_{cap}$. 
4 Conclusion

Applying the electric voltage to the electrodes with a dielectric liquid between them causes a flow of the liquid due to presence of DEP force. The temporal dependence of the height of rise of the liquid when the voltage is applied can be obtained by a term describing the DEP pressure to the equation derived for temporal dependence of height of rise in ordinary capillary systems. By changing the electric voltage applied to the electrodes it is possible to control the liquid level and the velocity of the liquid flow.

Acknowledgements This research has been supported by National Science Centre within the Project No. 2012/05/B/ST8/02742

Received in July 2016

References


A combined numerical-experimental model of air foil bearing compliant structure

Institute of Turbomachinery, Lodz University of Technology,
219/223 Wólczańska, 90-924 Łódź, Poland

Abstract
The paper concerns the development process of numerical-experimental model of air foil bearing compliant structure. Theoretically, static and dynamic characteristics of the foil bearing are the result of elastic combined properties of the two serially connected elements. One of them is a thin gas film of very small thickness and relatively high static and dynamic stiffness. The second elastic element is a pretensioned bump foil spring. This paper focuses on the properties of compliant foil structure and leaves aside the gas film behavior. At the beginning of the model development, the global stiffness and damping properties of compliant structure were obtained from the test stand measurements. In the next step, some assumptions concerning the model were made. The main one was the replacement of the single bumps of the corrugated foil by the set of elastic-damping numerical elements. At last, the fine-tuning of the model was carried out. The tuning involved changing of subelements local damping and stiffness properties, which in effect influenced the global properties of foil bearing. The tuning criterion for the model was defined as follows: the bearing global stiffness and damping properties of the model should not differ from the experimentally obtained values more than 10%.

Keywords: Airfoil bearing; Numerical model; Gas bearing coefficients

1 Introduction
The gas foil bearings are a type of aerodynamic bearings that utilize air as their lubricating medium. The name ‘foil’ comes from their design (Fig. 1), because the shaft is supported in a thin metal foil structure that is mounted in the bearing
sleeve. At high rotational speed, a gas film occurs between the shaft and the foil structure. The first foil bearing was designed and built in 1953 but that concept wasn’t able to compete with commonly used bearings. The main drawback was the lack of a proper coating layer. The coating layer in the foil bearing ensures low friction coefficient and ability to withstand contact conditions during startup and shutdown. The foil bearings found their first successful applications in aerospace industry. In 1969 an air cycle machine from the McDonnell Douglas DC-10 jet airliner was supported in foil bearings and fulfilled engineers expectations. Since that time foil bearings evolve through many concepts and found many applications but the base structure did not change much.

![Figure 1: Foil bearing structure [1].](image)

Foil bearing operation is similar to oil hydrodynamic bearings but the working fluid is not oil but air which results in many advantages as well as disadvantages. The top foil is responsible for proper forming of the gas film. The bump foil clenches the top foil around the shaft. This pre-clamp provides higher bearing load capacity, but on the other hand increases frictional torque during the start-up. Using air as the lubricating medium simplifies the foil bearing construction, and moreover, no other device like supply pump or compressor is needed. Low viscosity of air results in minimal bearing losses, however lower viscosity means also lower bearing load capacity. One of the most important disadvantages is friction between shaft and top foil during run-up and run-down of machine, when rotation speed is low. Difference between oil and foil bearing is shown in Fig. 2.

Correctly operating foil bearings are design solutions that have wide possibilities of applications, unavailable for rolling or oil bearings [1]. Nowadays in many scientific centers in the world, the main research on foil bearings is devoted to elimination of their basic disadvantages such as high start-up moment and wear.
of component surfaces. The next important goal is to predict and design properly the behavior of a complex support system consisting of the gas film and the compliant – damping foil structure. This can be achieved by building a trustworthy foil bearing dynamic model that accurately imitates the real bearing properties. The complexity of the theoretical model of such a bearing is deepened by the issue of the relative motion of both foils and the friction of the bump and cylindrical foils that takes place [2,3].

2 Dynamic properties of the compliant structure of the bump foil bearing

Theoretically, static and dynamic characteristics of the foil bearing are the result of elastic combined properties of the two elements serially connected (see Fig. 3). One of them is a thin gas film of very small thickness and relatively high static and dynamic stiffness, $K_G$. The second elastic element is a pretensioned bump foil spring with stiffness, $K_F$, and damping, $C_F$.

It should be noticed that depending on the bump foil pre-tension, a stiff gas film appears for speeds from few thousand to over a several dozen kilos revolutions per minute. Above this speed limit, a continuous gas film occurs, and the foil bearing operates properly when the rotating journal loses contact with the top foil.
The accepted physical model of the start-up of aerodynamic foil bearings allows one to formulate the following statements:

- for the journal rotational speed below $n_{lim}$, where $n_{lim}$ denotes the rotational speed at which the continuous gas film appears, it can be assumed that the dynamic properties of the bearing depend on the stiffness of bump foil springs, because $K_G \ll K_F$;
- at the journal rotational speed above $n_{lim}$, theoretically, the dynamic properties of the support system depend on the combined stiffness of the two elements serially connected.

The complexity of the analysis of the foil bearing theoretical model is caused by friction between the bump foil and the sleeve and between both foils. The friction comes from a relative motion of the foils and a relative motion of the bump foil and the sleeve. This physical phenomenon results in highly nonlinear dynamic properties of the foil bearing support system [4,5].

Some experimental attempts were made to identify these properties and a foil bearing test rig was built for this purpose (Fig. 4). The test rig consisted of a fixed journal, frictionlessly supported sleeve and a modal shaker. The sleeve of 1.6 kg weight was excited with a sinusoidal waveform force by the shaker to simulate real synchronous excitation caused by rotor unbalance. During the experiment, the excitation force and the sleeve displacement were measured. The measured object was a foil structure of third+ generation bearing. The dimensions of the bearing were $34 \times 40$.

A typical response of the shaking system is a hysteresis loop, presented in Fig. 5. From this image, one can obtain the bump spring overall stiffness, $k$, and an area of the hysteresis loop, $W$, which represents the energy dissipated in a single motion cycle. The energy dissipation is related to the friction between the
A combined numerical-experimental model of air foil...  

Figure 4: Functional diagram and a photo of the test rig built for the experimental identification of dynamic properties of the third generation foil bearing support structure [7].

bearing foils and can be estimated from:

\[ W = \oint F dx = \pi \omega C_{eq} X^2, \]  

where: \( W \) – area of the hysteresis loop, \( F \) – force amplitude, \( x \) – displacement, \( X \) – vibration amplitude, \( \omega \) – circular frequency. Transforming Eq.(1) one can obtain the equivalent damping coefficient.

\[ C_{eq} = \frac{W}{\pi \omega X^2}. \]  

Figure 5: Image response as a hysteresis loop associated with friction and elasticity of the compliant foil bearing assembly [7].

During measurements foil structure was subjected to sinusoidal force at three values of force per one excitation frequency. Six different frequencies from range 100 to 600 Hz were applied. The applied excitation force and the bearing amplitude
were measured. An important parameter during measurements was sampling frequency. To avoid aliasing, the sampling frequency was at least forty times higher than excitation frequency. The range of applied excitation forces is shown in Tab. 1.

Table 1: Measurement program

<table>
<thead>
<tr>
<th>Sampling rate, Hz</th>
<th>Excitation frequency, Hz</th>
<th>Amplitude of excitation force, N</th>
</tr>
</thead>
<tbody>
<tr>
<td>5000</td>
<td>100</td>
<td>5 7 10</td>
</tr>
<tr>
<td>10000</td>
<td>200</td>
<td>5 7 10</td>
</tr>
<tr>
<td>12500</td>
<td>300</td>
<td>5 7 10</td>
</tr>
<tr>
<td>25000</td>
<td>400</td>
<td>5 7 10</td>
</tr>
<tr>
<td>25000</td>
<td>500</td>
<td>5 7 10</td>
</tr>
<tr>
<td>25000</td>
<td>600</td>
<td>5 7 10</td>
</tr>
</tbody>
</table>

3 Numerical model

The model of foil structure was written in commercial software ANSYS Parametric Design Language 16.1 [11] to imitate the real foil bearing. Similarly to the test rig, the shaft is fixed to the support, and the bearing sleeve in the model is excited by sinusoidal force applied to node 49 (see Fig 6). The mass of the sleeve equals to 1.6 kg and is focused also in node 49. The sleeve was considered to be rigid in comparison to the foil structure. Every single bump of the bumpfoil in bearing was simulated by two COMBIN14 elements which have their own internal damping and stiffness coefficients. By changing these coefficients the model behaviour can be adjusted to the measurement results for each configuration of frequency depend on the excitation force. The main goal was to adjust model to measurement results with 10% aberration margin, although it was impossible in some cases. Schematics diagram and realization in ANSYS are shown in Fig. 7. The whole model of foil bearing is shown in Fig. 6.

4 The comparison results

Figure 8 presents the amplitude response of the bearing model to sudden application of sinusoidal force at substep 1 (for time $t = 0$ s). As can be observed, the
A combined numerical-experimental model of air foil.

Figure 6: Numerical model of the full foil bearing. On the right: zoom of the modeled structure of the bump foil.

Figure 7: A simplified model of single bump represented by two COMBIN14 elements.

model achieves stable amplitude after approx. 1000 substeps. For more reliable results, the hysteresis loop from above 1500th substep was taken for a comparison with the experimental results. A representative response is presented in Fig. 9. This is an exemplary result for frequency $f_{ex} = 100$ Hz and force $F_{ex} = 5$ N. The diagram shows two histeresis loops: one obtained from the experiment (the solid line) and the other obtained from the model adjustment (the dotted line).

The model and measurement comparative analyses were conducted for each
frequency. The comparison covered few important bearing properties, i.e., the global damping coefficient, the global stiffness, the dissipated energy and minimal, maximal and mean amplitude. Results of a comparative analysis for 100 Hz are shown in Tab. 2.

Figure 8: Amplitude of model sleeve vs. number of substep. Exemplary results for frequency $f_{ex} = 100$ Hz and force $F_{ex} = 5$ N.

Figure 9: Hysteresis loops obtained from both numerical model and from measurement.
Table 2: Result of comparative analysis for frequency of 100 Hz.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Deviation</td>
<td>-5.89%</td>
<td>-0.84%</td>
<td>5.24%</td>
<td>1.90%</td>
<td>-9.37%</td>
<td>-6.86%</td>
</tr>
<tr>
<td>7</td>
<td>Measurement</td>
<td>213.933</td>
<td>57.380</td>
<td>-46.180</td>
<td>51.780</td>
<td>40.423</td>
<td>139119</td>
</tr>
<tr>
<td></td>
<td>Model</td>
<td>215.731</td>
<td>54.072</td>
<td>-46.065</td>
<td>50.068</td>
<td>43.597</td>
<td>143798</td>
</tr>
<tr>
<td></td>
<td>Deviation</td>
<td>0.84%</td>
<td>-5.77%</td>
<td>-0.25%</td>
<td>-3.31%</td>
<td>7.85%</td>
<td>3.36%</td>
</tr>
<tr>
<td>10</td>
<td>Measurement</td>
<td>1225.685</td>
<td>101.150</td>
<td>-69.330</td>
<td>85.240</td>
<td>85.460</td>
<td>138443</td>
</tr>
<tr>
<td></td>
<td>Model</td>
<td>931.204</td>
<td>87.878</td>
<td>-73.922</td>
<td>80.900</td>
<td>43.597</td>
<td>143798</td>
</tr>
<tr>
<td></td>
<td>Deviation</td>
<td>-24.03%</td>
<td>-15.12%</td>
<td>6.62%</td>
<td>-5.09%</td>
<td>-15.66%</td>
<td>-4.24%</td>
</tr>
</tbody>
</table>

5 Results discussion

The comparative analysis covered the whole spectrum of measurements described in Tab. 1. After obtaining the experimental object properties, the tuning of the model was performed. This allowed obtaining the model that accurately behaves like a real object. The values of global stiffness and damping coefficients for both model and real object are presented in Figs. 10 and 11.

As one can see in above figures, damping is dropping significantly for frequency 500 Hz. However, for 600 Hz it is almost three times higher than the average from range 100 to 400 Hz. Such behaviour of the foil structure is favourable, because foil bearings usually work with high rotation speed machines and we can expect that this coefficient has a rising tendency for frequencies higher than 600 Hz. Stiffness coefficient in the range from 100 to 500 Hz has almost linear rising tendency and differences between excitation forces are minimal. In that range, the stiffness coefficient depends only on frequency. For 600 Hz, it reaches the highest value for each excitation force and for that frequency we can see that the coefficient depends not only on the frequency but also on the excitation force. The unique nonlinear behaviour of the foil structure is quite interesting and in further studies should be examined thoroughly.

The numerical model in most cases was quite easy to tune its behaviour to experimental performance of the real bearing. The results are shown in Figs. 12 and 13. One of the most important conclusions concerns the applied values of
Figure 10: Global bearing damping coefficient versus excitation frequency.

Figure 11: Global bearing stiffness coefficient in function of excitation frequency.

COMBIN14 element coefficients responsible for stiffness and damping. As we can see in above figures, that applied damping coefficient of COMBIN14 element has

ISSN 0079-3205 Transactions IFFM 133(2016) 117–129
similar curve as the global damping coefficient presented in Fig. 10. In the other hand, the applied stiffness coefficient values of single COMBIN14 do not imitate the values of global stiffness of the bearing, but the obtained stiffness curve looks very similar to the damping coefficient curve which is really interesting.

We also check if there is any simple relation between COMBIN14 and real bearing global coefficients. This was done by dividing one by another, allowing describing them by using easy equation. The quotient of damping coefficients was called $A_1$, and the quotient of stiffness coefficients was called $A_2$. Results are pre-
presented in Figs. 14 and 15. As we can see the simple relation does not exist, which indicates that model is too simple to be useful for engineering purposes. Never the less, we see that for 600 Hz factors $A_1$ and $A_2$ depend only on frequency, what leads to the conclusion that for higher frequencies, the COMBIN14 coefficients could be more useful to describe a reliable numerical model of airfoil bearing.

Figure 14: Result quotient of measured global bearing and COMBIN14 element damping coefficients.

Figure 15: Result quotient of global bearing and COMBIN14 element stiffness coefficients.
6 Conclusion

Measurements of foil structure parameters indicate that better operational ones at high speed rotation are not the only result of gas aerodynamic reaction. It can be observed that with the rising frequency, stiffness and damping coefficient of foil structure increases. It will be a good idea to check that structure in a wider range of frequencies in the future, because the most interesting results show at the end of the measure range. Due to the limitation of the modal shaker, it is not possible at present.

Received in July 2016

References

Resonant characteristics of flow pulsation in pipes due to swept sine constraint

Institute of Turbomachinery, Lodz University of Technology, 219/223 Wólczańska, 90-924 Łódź, Poland

Abstract

A fast and easy to use method for analysing the amplitude frequency characteristics of pulsating flow in pipes with a sine sweep input function is presented. The key element is the sweeping excitation of the pulse generator frequency which generates fast whole-spectrum resonance (from 20 Hz to 160 Hz). A case study describes tests conducted into the applicability of the method for research into pulsating flows. First, the test rig is briefly described, along with the main assumptions of the research. Next, the amplitude frequency resonance due to swept oscillations is estimated for increasing, decreasing and changeable input mass flow rates. These characteristics are approximated based on a second-order transfer function. Following such assumptions, the damping coefficients and resonance frequencies are estimated. The influence of the direction of the frequency sweep method (increasing or decreasing) on the resultant resonance frequency is also investigated.

Keywords: Pulsating flow in pipes; Sweep sine excitation; Amplitude frequency characteristics

1 Introduction

The purpose of this work was to estimate the amplitude frequency characteristics of pulsating flows in pipes with a sine sweep input function. Sweep harmonic constraints enable faster acquisition of whole-spectrum resonance characteristics. The sweep input is a natural extension of the classical sinusoidal signal used for dynamic system response testing. This method is required for simulations.
designed to estimate body characteristics. It is commonly used in turbomachinery (for spectral and modal analysis of rotors [1]), process and mechanical engineering (sine sweep vibration testing), acoustics [2,3] and automotive engineering (sweeping automotive suspension [4]). The proposed method was used in acoustic investigation of pipe flows [5]. Sweeping was limited to only one tone in each sweep step, because of the high level of turbulent noise generated by the mean flow [6]. Multireference sine sweep tests are conducted in the aircraft industry for estimating ground vibrations. Testing time can be minimized and the block size maximized by performing wrapped sweeps, in which the frequency difference between each source is maximized over the full frequency range of the test [7]. However, to the best knowledge of the author, no results have been published for swept inputs and pulsating flows in pipes. In particular, ions of investigations of the influence of down- and up-sweep input method on the resultant resonance frequency are presented.

The main contributions of this paper are the following:

- Real measurements taken during transitional states of pulsation frequency changes (from 40 Hz to 160 Hz) in pipes.
- A simplified Simulink model designed to estimate local pulsation amplitude and amplitude-frequency characteristics.
- An analysis of the influence of frequency change in a range of amplitudes from 40 Hz to 160 Hz on air pressure during pulsation change at three control sections.
- Measurements showing resonant frequency changes with increasing and decreasing pulsation frequencies.
- Proposed parameters for estimating second-order inertial elements (the damping coefficient and resonant frequency) to describe the observed phenomena.

1.1 Basic principles of the investigation

The laboratory test rig was built to research the transitional states of changing pulsation frequencies in pipes. The main features of the test rig are presented in Fig. 1, including the pulse generator (PG) and three control sections (0, K, 3). The main flow parameters required to evaluate pressure, temperature and mass flow were measured at three control sections. The test procedure also examined the frequency change domain, in terms of initial and final conditions. The change in frequency was calculated based on the step function. Finally, the amplitude frequency characteristics were estimated using the curve fitting function in Matlab software [8]. This estimate was based on second-order inertial elements, and provided quite a good representation of the acquired data.
2 Measurement procedure

The proposed method for identifying swept amplitude-frequency characteristics can be summarized as follows, Fig. 2:

- The swept pulsation frequency change is set using an inverter connected to an electric motor. This system drives the pulse generator using the ramp frequency constraint mode.
- The transient states of the analysed phenomenon are measured.
- The results are processed using commercial Matlab-Simulink environment, including: filtering of noisy signals, estimation of instantaneous wave parameters (amplitude, mean value), and determination of amplitude-frequency characteristics. Finally, second order oscillating element approximation parameters such as resonance frequency and damping coefficient are estimated.

The proposed method of swept frequency response characteristics estimation (SFRCE) couples accurate transient measurement with the possibilities provided by block diagram environment for modeling and simulation software. It is thereby possible for these characteristics to be processed and plotted automatically in a very short time after the measurements (about 2 min).

2.1 Setting sweep-ramp pulsation frequency changes

The NI USB 6259 high speed multifunction data acquisition (DAQ) module was used as an inverter controller. The analog output from DAQ devices (board), which was a standard voltage signal in a range from 0 V to 10 V, was transferred into a standard current signal (4–20 mA) using the Aplisens ZSP-41 signal converter. In this way, the inverter was switched to external frequency input (current type). This facilitated setting the ramp change (20 Hz per second) for the required inverter frequency. The frequency variation in the time domain is presented in Fig. 3. The high repeatability of the lines, which is clearly visible, confirms that
the average frequency change coefficient was equal to: \( \frac{\Delta f}{\Delta t} = 20.09 \text{ Hz/s} \), (see Tab. 1), \((\Delta f - \text{change of frequency, } \Delta t - \text{time step})\). The standard deviation of the achieved ramp coefficients was estimated as 0.043 Hz/s, based on the data in Tab. 1.

### 2.2 Measurement and acquisition of transient states

Figure 4 shows a test rig built at the Institute of Turbomachinery (Lodz University of Technology) for identifying amplitude-frequency characteristics in pipes with pulsating flow, following the procedure described above. Extensive research on dynamic phenomena in straight pipes supplied with a pulsating flow of gas shows that the variation of flow parameters (pressure, temperature, and velocity) depends on the nature of the pipe outlet [8,9,10]. Based on research by Olczyk [8], experimental data were incorporated into a simulation model, presented in [8,10]. The simulation and experimental parameters of the dynamic states of pulsating flow were compared. The results of their synthesis are presented below. The main parameters of the simulated flow were as follows [8]:

- range of desired values for the frequency of the pulse generator 
  \[ f = 40–160 \text{ Hz} \];
- pipe diameter \( D_p = 42 \times 10^{-3} \text{ m} \);
Resonant characteristics of flow pulsation. . .

Table 1: Frequency change coefficient

<table>
<thead>
<tr>
<th>series</th>
<th>$\Delta f/\Delta t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20.15</td>
</tr>
<tr>
<td>2</td>
<td>20.02</td>
</tr>
<tr>
<td>3</td>
<td>20.13</td>
</tr>
<tr>
<td>4</td>
<td>20.08</td>
</tr>
<tr>
<td>5</td>
<td>20.07</td>
</tr>
<tr>
<td>6</td>
<td>20.11</td>
</tr>
<tr>
<td>7</td>
<td>20.14</td>
</tr>
<tr>
<td>8</td>
<td>20.02</td>
</tr>
<tr>
<td>9</td>
<td>20.10</td>
</tr>
<tr>
<td>10</td>
<td>20.09</td>
</tr>
<tr>
<td>11</td>
<td>20.06</td>
</tr>
<tr>
<td>12</td>
<td>20.05</td>
</tr>
<tr>
<td>aver.</td>
<td>20.09</td>
</tr>
<tr>
<td>std.dev.</td>
<td>0.043</td>
</tr>
</tbody>
</table>

Figure 3: Frequency variation in time step domain for fourteen test probes.

- pipe length $L_p = 0.544$ m, determined with resonance at 70 Hz and 140 Hz;
- nozzle diameter $D_n = 10 \times 10^{-3}$ m. The nozzle is mounted at one end of the pipe, at cross-section (3);
- desired flow temperature $T = 313.15$ K;
- mean flow speed $u = 20 \text{ m/s (mean value)}$;
- mean pressure $p = 115000$ Pa.

Transient and mean values for pressure, temperature and specific mass flow rate were measured at control sections (0) and (3), shown in Fig. 4. Transient pressure was also measured in section (K), located in the middle of the length of pipe (Fig. 1).

The pulse generator enables measurements of variable reliable and repeatable flow pulsations (Fig. 5). The stream line shown in the figure was prepared using Flow Express Package at Solidworks software [9]. This unit is almost frictionless and is driven by an electric motor with a variable speed drive (inverter) to control the speed of pulse generator rotor. The following equipment was used for measurements [8]:

- piezoresistive transducers for pressure measurements: Endevco 8510C-15 and 8510C-50 [10];
constant current thermometers (CCT) for temperature measurements;
constant temperature anemometers (CTA) for specific mass flow rate measurements.

The CCT and CTA probes used 5 μm tungsten wire.

2.3 Proces modeling and simulation

The Simulink model performs the following tasks:

- **Importing acquired transient flow parameters.** Signals from the test rig are imported into the workspace as an eight column array. The time domain for all compared series is the same: 20 kHz sampling frequency per channel and 160,000 probes, with an 8 s probe length for each series. The cyan blocks in Fig. 6 import the appropriate columns from the measured
Figure 6: Simulink model of measurement data processing: a) frequency estimation, b) triggered subsystem, c) amplitude estimation.
array to generate transient signals in the simulation.

- **Filtering of noisy signals.** Because of the constant frequency changes produced by the pulse generator, a method of filtering noisy signals is used based on transfer function filtering (TFF). Classical fast Fourier transform (FFT) method cannot be applied in this case, because of the swept sine constraint (changing pulse generator frequency), where the base frequency should approximately constant. There are other commonly-used short windowing methods based on short time Fourier transform (STFT) [4], which, unlike wavelet analysis, do not limit precision in the time domain or the frequency range. The noisy signal is transmitted via the transfer function (which can be defined as a frequency function with a time constant of $1/3000$ s), in dots in Fig. 6. The time constant is estimated by comparing the measured data with their filtered approximations, as shown in Fig. 7, to find a compromise between representative approximated values and their noisiness. The accuracy of the approximation is compared with nonswept measurements and with a standard FFT approximation (with eight harmonics), resulting in a correlation coefficient of about 0.98. The noisy range is estimated with the first-order derivate of the corresponding filtered parameter.

![Figure 7: Comparison of measured data with their filtered approximations.](image)

- **Measured signal processing.** Transfer of measured signals (0–10 V, 0–250 mV) is performed using the static characteristics of the appropriate transducers. Static characteristics are defined according to the calibration process described in detail in [15]. They are implemented into Simulink model as a user-defined function (in ‘fen’ simulink block);
- **Identification of time stamp.** A photoelectric revolution transducer gives
one peak per each revolution of the electronic-motor-drives. These peaks are also acquired for each measurement series. Counting the measured pulse generator frequency proceeds according to the following steps: counting the particular pulse and its transformation into a true/false signal (‘counter’ element of the model presented in Fig. 6a); ‘Counter’ element of triggered measurement of the time between two peaks (indicates time of the one particular pulse generator revolution) and then the frequency is estimated (model presented in Fig. 6b).

- **Identifying the amplitude of the pulse transient parameter.** Transient parameters (temperature, mass flow rate and pressure) are estimated by finding their maxima and minima, then their averages calculated, and finally the amplitude of the pulse is estimated, as presented in Fig. 6c. Data processed in this manner are shown in Fig. 8.

![Figure 8: Graphical representation of amplitude estimation algorithm.](image)

- **Amplitude-frequency characteristics.** Having estimated the amplitude of the analysed parameters, it is possible to calculate the quotient of amplitudes at two crucial cross-sections (0) and (3), $P_3/P_0$, for pressure amplitude-frequency characteristics (Fig. 9). Approximation is made using Eq. (1) [4] and the Curve Fitting Tool, using custom equation settings and default 95% confidence bounds.
Figure 9: Amplitude-frequency characteristics of pressure in two cross-sections ($P_0$ and $P_3$).

\[ M(f) = \left\{ \left[ 1 - \left( \frac{f}{f_n} \right)^2 \right]^2 + \left[ \frac{2\zeta f}{f_n} \right]^2 \right\}^{-1/2}, \]

where: $M(f)$ – magnitude of oscillations, $f$ – frequency, $f_n$ – resonance frequency, $\zeta$ – relative damping coefficient.

3 Experimental results

The results of the research program presented in Fig. 10 and Tab. 2 are as follows:

- Increasing (147 Hz) and decreasing (137 Hz) the rotary speed of the pulse generator results in different resonant frequencies. These are caused by the acoustic phenomenon in the tested object, which results in hysteresis of the proposed approximation with second-order oscillating elements.

- The amplitude-frequency characteristics has been estimated using second-order inertial elements. The damping coefficient was 0.125 when the rotary speed was decreased and 0.131 when it was increased (see Fig. 10).

- The resonant frequency for the stationary case as a mean value of the two previously mentioned probes has been found (see Fig. 10). Parameters calculated in this way are comparable with stationary results reported in [10].

- The influence of the mass flow rate on the damping coefficient, approximated using the second-order oscillating transfer function, has been found. Because of the limited range of probes, it is difficult to determine the correlation function between the mean mass flow rate in pipes and the relevant damping
coefficient. Generally, increasing the mean mass flow rate decreased the damping coefficient in the proposed approximation of pulsating flows in the analysed pipe.

The tabulated results are given in Tab. 2. There are assumed the whole probes according to the presented method. There were procured presented procedures for each probe of the eighteen listed at the Tab. 2. The experiment covers three cases according to the mass flow rate domain (average rates 11, 22, 32 × 10⁻³ kg/s). For each case there were performed three sweeping up and three sweeping down probes. For all probes there were estimated magnitude of oscillations and relative damping coefficient.

4 Conclusion

The proposed method of swept frequency response characteristics estimation (SFRCE) couples accurate transient measurement with the possibilities provided by block diagram environment for modeling and simulation, employing a transfer function filtering method proposed by the author. Using this method, it was possible for swept frequency response characteristics to be processed and plotted automatically in a very short time once the measurements have been taken. It is also possible to create a 3D map showing the amplitude frequency characteristics for pressure in the tested pipe as a function of pulsation frequency and the
Table 2: Test series assumption.

<table>
<thead>
<tr>
<th>No. of series</th>
<th>File name</th>
<th>Trend</th>
<th>Mass flow rate $\times 10^{-3}$ [kg/s]</th>
<th>Average mass flow rate $\times 10^{-3}$ [kg/s]</th>
<th>Magnitude of oscillations, $M(f_n)$ [-]</th>
<th>Average $M(f_n)$ [-]</th>
<th>Relative damping coefficient $\zeta$ [-]</th>
<th>Average relative damping coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>49</td>
<td>up</td>
<td>10.9</td>
<td>2.35</td>
<td>2.4</td>
<td></td>
<td>0.280</td>
<td>0.27</td>
</tr>
<tr>
<td>2</td>
<td>50</td>
<td>down</td>
<td>11.1</td>
<td>2.42</td>
<td>0.230</td>
<td></td>
<td>0.290</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>51</td>
<td>up</td>
<td>2.43</td>
<td>2.36</td>
<td>0.250</td>
<td></td>
<td>0.250</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>52</td>
<td>down</td>
<td>11</td>
<td>2.31</td>
<td>0.300</td>
<td></td>
<td>0.300</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>53</td>
<td>up</td>
<td>2.41</td>
<td>2.41</td>
<td>0.260</td>
<td></td>
<td>0.260</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>54</td>
<td>down</td>
<td>22</td>
<td>3.5</td>
<td>0.220</td>
<td></td>
<td>0.220</td>
<td>0.21</td>
</tr>
<tr>
<td>7</td>
<td>55</td>
<td>up</td>
<td>3.52</td>
<td>3.8</td>
<td>0.198</td>
<td></td>
<td>0.198</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>56</td>
<td>down</td>
<td>22.4</td>
<td>3.76</td>
<td>0.219</td>
<td></td>
<td>0.219</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>57</td>
<td>up</td>
<td>3.48</td>
<td>3.83</td>
<td>0.200</td>
<td></td>
<td>0.200</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>58</td>
<td>down</td>
<td>22.4</td>
<td>3.83</td>
<td>0.221</td>
<td></td>
<td>0.221</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>59</td>
<td>up</td>
<td>3.83</td>
<td>3.83</td>
<td>0.196</td>
<td></td>
<td>0.196</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>60</td>
<td>down</td>
<td>33</td>
<td>3.8</td>
<td>0.131</td>
<td></td>
<td>0.131</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>61</td>
<td>up</td>
<td>3.75</td>
<td>4.2</td>
<td>0.125</td>
<td></td>
<td>0.125</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>62</td>
<td>down</td>
<td>32</td>
<td>4.1</td>
<td>0.128</td>
<td></td>
<td>0.128</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>63</td>
<td>up</td>
<td>3.82</td>
<td>4.22</td>
<td>0.130</td>
<td></td>
<td>0.130</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>64</td>
<td>down</td>
<td>31.9</td>
<td>4.22</td>
<td>0.126</td>
<td></td>
<td>0.126</td>
<td></td>
</tr>
</tbody>
</table>

mass flow rate function (Fig. 11). The main advantages of the method presented here are:

- acquisition of measurement data is ten times faster than measurements of each frequency separately;
- the inverter used in the test rig enables measurements for the sweep sine method because of the high stability of its ‘amp’ frequency excitation (see Tab. 1);
- an easy to use automatic algorithm for processing measurement data enables swept frequency response characteristics estimation just in a few minutes after measurements;
- it is also possible to indicate subharmonic frequencies, as shown in Fig. 9 in the dotted ellipse area. These subharmonics were not indicated in the approximation process because of the order level of the model (second order).
Figure 11: Amplitude frequency characteristics for pressure in tested pipe as a function of pulsation frequency and mass flow rate.

Received in July 2016

References


ISSN 0079-3205 Transactions IFFM 133(2016) 131–144


Roman Rusanov*, Łukasz Jędrzejewski*, Piotr Klonowicz*, Grzegorz Żywica*
Piotr Lampart*, A. Rusanov*

Design and performance study of a small-scale waste heat recovery turbine

* Institute of Fluid Flow Machinery, Polish Academy of Sciences, 80-231 Gdańsk, Fiszera 14, Poland
* A.N. Podgorny Institute of Mechanical Engineering Problems National Academy of Sciences of Ukraine, Kharkov, Ukraine

Abstract
The paper presents the design process of a radial-axial turbine working with SES36 working fluid. First, the mean-line design process is performed and then the geometry is developed. In the next stage the numerical verification is performed taking into account the real properties of the working fluid. The properties are implemented via a look-up table and by a modified Benedict-Webb-Rubin equation of state. The presented turbine is characterized by a very small stator outflow angle which is about 4.5° but despite this small value, the efficiency of the machine is relatively high and equal to about 88%. The influence of internal leakages has also been investigated.

Keywords: Cogeneration unit; Low-boiling working media; Numerical method; Turbine; Radial-axial stage

1 Introduction

A promising direction of development of energy-saving technologies for Europe is application of small power cogeneration plants operating with the low-boiling working media organic Rankine cycle (ORC), [1–4]. Such systems can also be used to utilize low-temperature waste heat, and work with renewable fuels – various

*Corresponding Author. Email adress: rrusanov@imp.gda.pl

ISSN 0079-3205 Transactions IFFM 133(2016) 145-162
types of biomass. The turbine is an important element of such cogeneration plants. The peculiarity of these turbines is their usually relatively small size, which complicates the task of achieving an acceptable level of gas-dynamic efficiency.

2 Plant layout. The initial data for the creation of a turbine

The ORC systems are similar to the steam cycles used in the large power industry. The fields of application of both technologies are roughly shown in Fig. 1 [5].

![Figure 1: Fields of application of ORC systems and traditional steam systems [5].](image_url)

The Institute of Fluid Flow Machinery, Polish Academy of Sciences (IMP PAN) has built a combined gas-vapour cycle with the nominal power of about 400 kW and a high electrical efficiency. The primary generation unit is an internal combustion engine fuelled by natural gas which can also be fuelled by biogas and syngas in real applications and distributed cogeneration. At least two variants of such a system can be considered. For example, the ORC itself can work for cogeneration and accept both the exhaust heat and heat from the engine cooling loop. Alternatively, the ORC can work to produce only electrical power while the heat from the engine’s cooling loop can directly be used for heating.

The system built at IMP PAN makes use of the second option. The schematic of the cogeneration plant and its photographs are shown in Figs. 2 and 3. As a working fluid, the substance SES36 has been used. The medium delivering

ISSN 0079-3205 Transactions IFFM 133(2016) 145–162
the heat from the engine exhaust to the ORC is a high temperature thermal oil Veco 5HT. The heat from the engine is transferred to the oil in a heat exchanger and then delivered to the preheater and the evaporator. The working fluid is evaporated and directed to the expansion unit coupled with the electrical generator. After the expansion process and transferring heat in the regenerator the fluid is directed to the condenser. The condenser transfers the remaining working fluid heat to the mixture of the water and glycol.

Figure 2: Schematic of the cogeneration plant with a combustion engine topped by an ORC cycle (D – combustion engine, M – valve control.)

3 Justification of the radial-axial turbine choice

The turbines characterized by the radial-axial direction of flow in many cases seem to be an attractive design solution. One of the main reason of using this type of turbine stages is their high efficiency for small specific speed values. Small values of this parameter result from small volume flow rate flowing through a device. Such situation takes place for example in small gas turbines or turbochargers [6]. The ORC systems are an alternative for traditional Rankine cycles but in fact ORC becomes really competitive for small power applications where the efficiency is not the most important factor and other characteristics such as simplicity, compact size and low investment costs are desirable. Small volume flow rates result
not only from the small scales of the systems. The fluids used in ORC plants have large particles which leads to large densities. Large density means small specific volume hence small volume flow rates. The tendency of using turbines of radial-axial direction of the flow in ORC systems can be found in many publications. All of them underline relatively high efficiencies of the small power machines [7–11].

A well described example of a 30 kW turbogenerator with a radial-axial turbine has been presented by Kang [7]. Figures 4 and 5 show the 3D model and a photograph of the device. The measured power and efficiency near the design point were found at 32.7 kW and 78.7%, respectively. It must be noted that this is the electrical efficiency of the turbogenerator which besides the turbine internal losses takes into account the shaft and disc friction losses, mechanical losses and electrical losses. In comparison with other small ORC turbines described in the literature this value is considered high [12,13].

Good parameters of radial-axial stages were one of the reasons why this kind of design was applied in the case of a turbogenerator working with the Solkatherm SES36 fluid at IMP PAN. This fluid is an azeotropic mixture [14], its thermodynamic parameters – both in 0D and computational fluid dynamics (CFD) simulations – have been acquired from the CoolProp thermodynamic properties library [15].
4 Radial-axial single stream turbine

The most important design parameters of the turbine are shown in Tab. 1. In order to obtain a satisfying blade height at the stator a very small nozzle outflow angle $\alpha_1$ was set equal to 4.4°. The velocity triangles for the stage have been presented in Fig. 6 and the main dimensions of the stage can be found in Fig. 7. The flowpath geometry was designed in the BladeGen software which belongs to the Ansys software package [17]. The designed nozzle (Fig. 8) has a convergent geometry despite supersonic character of the flow at the outlet. The reason for that is a relatively low Mach number which is equal to 1.33. For these conditions a convergent-divergent nozzle gives negligible efficiency benefit [16]. The stator and rotor assembly as a 3D model has been shown in Fig. 9.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet pressure</td>
<td>1.464 MPa(abs)</td>
</tr>
<tr>
<td>Inlet temperature</td>
<td>414.15 K</td>
</tr>
<tr>
<td>Outlet pressure</td>
<td>0.22 MPa(abs)</td>
</tr>
<tr>
<td>Mass flow rate</td>
<td>1.22 kg/s</td>
</tr>
<tr>
<td>Rotational speed</td>
<td>15000 rpm</td>
</tr>
</tbody>
</table>

Table 1: SES36 turbine design parameters.
The 3D simulations were performed in a commercial code Ansys CFX [17]. The whole computational domain consisted of 0.8 million nodes. The applied turbulence model was $k-\omega$ SST. The set of boundary conditions included the total parameters at the inlet and the static pressure at the outlet, also the inlet turbulence intensity. The summary of the 0D and CFD calculations has been presented in Tab. 2.

It turns out that despite the supersonic character of the flow and small stator angle, the efficiency of the stage is relatively high and equal to 88% (according to 3D results). This value does not include the internal leakage losses. Some complimentary distributions of the Mach number and velocity vectors have been shown in Figs. 10 and 11.
Figure 9: 3D model of the rotor and stator assembly.

Table 2: Results of the performed computations, part I.

<table>
<thead>
<tr>
<th>Model 0D</th>
<th>Model 3D</th>
<th>Number of blades</th>
<th>Torque</th>
<th>Rotational speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P$ [kW]</td>
<td>$\eta$ [-]</td>
<td>$P$ [kW]</td>
<td>$\eta$ [-]</td>
<td>$T$ [Nm]</td>
</tr>
<tr>
<td>29.4</td>
<td>0.86</td>
<td>30.2</td>
<td>0.88</td>
<td>22</td>
</tr>
</tbody>
</table>

Table 3: Results of the performed computations, part II.

<table>
<thead>
<tr>
<th>Inlet pressure</th>
<th>Pressure between the blade rows</th>
<th>Outlet pressure</th>
<th>Inlet temperature</th>
<th>Temperature between the blade rows</th>
<th>Outlet temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>1464</td>
<td>492</td>
<td>220</td>
<td>414.15</td>
<td>386.45</td>
<td>370.1</td>
</tr>
</tbody>
</table>

5 Radial-axial double stream turbine

The main disadvantage of the single stream radial-axial construction is a significant axial thrust. This shortcoming is absent in the double stream turbine, in which the flow starting from the radial section is divided into two symmetrical axial flows (in different directions). This kind of design can be realized in two variants – back-to-back rotor wheels and front-to-front rotor wheels (this variant
is axially longer as there must be space between the rotors to form the outlets).

Figure 12 presents a view of the flow part and Tab. 4 shows the main characteristics of the double stream radial-axial turbine (single side). 3D calculations of the designed flow part were performed using the software package IPMFlow [12]. The calculation is performed on the grid with a total number of more than 1 million cells (about 500 thousand of cells in one row) using the Benedict-Webb-Rubin equation of state with 32 members [18].

Figures 13 and 14 show a visualization of the flow path, whereas Tab. 5 gathers main integral characteristics obtained from 3D calculations. Despite the fact that the flow part consists of a single stage, which operates at a high thermal gradient, there is a favorable flow pattern. The maximum value of the Mach number in the whole of the flow part is less than 2, there are no strong shocks and flow separations. The proposed flow part has a high gas-dynamic efficiency, its internal efficiency is 88.5%. Despite the advantageous properties of the double stream
Design and performance study of a small-scale waste...

Table 4: The geometrical characteristics of the radial-axial flow part.

<table>
<thead>
<tr>
<th>$r_{in}$, stator</th>
<th>$r_{out}$, stator</th>
<th>$l_{in}$, stator</th>
<th>$l_{out}$, stator</th>
<th>$z$, stator</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 mm</td>
<td>86.0 mm</td>
<td>3 mm</td>
<td>3 mm</td>
<td>41</td>
</tr>
<tr>
<td>$r_{in}$, rotor</td>
<td>$r_{mid, out}$, rotor</td>
<td>$l_{in}$, rotor</td>
<td>$l_{out}$, rotor</td>
<td>$z$, rotor</td>
</tr>
<tr>
<td>81 mm</td>
<td>36.3 mm</td>
<td>3 mm</td>
<td>16 mm</td>
<td>16</td>
</tr>
</tbody>
</table>

where $r_{in}$ – inlet radius, $r_{out}$ – outlet radius, $r_{mid, out}$ – outlet mid radius, $l_{in}$ – blade inlet height, $l_{out}$ – blade outlet height, $z$ – number of blades.

Figure 12: Relative Mach number ($x/l = 0.5$); left – stator, right – rotor.

geometry the single stream variant has been selected for actual manufacturing as it is a cheaper and simpler solution.

Table 5: The main integral characteristics of the flow part.

<table>
<thead>
<tr>
<th>Inlet pressure</th>
<th>Inlet temperature</th>
<th>Temperature between blade rows</th>
<th>Rotor inlet absolute velocity</th>
<th>Outlet absolute velocity</th>
<th>Rotor inlet relative velocity</th>
<th>Outlet relative velocity</th>
<th>Stage power</th>
<th>Stage efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_1$ [kPa]</td>
<td>$T_1$ [K]</td>
<td>$T_2$ [K]</td>
<td>$c_1$ [m/s]</td>
<td>$c_2$ [m/s]</td>
<td>$w_1$ [m/s]</td>
<td>$w_2$ [m/s]</td>
<td>$P$ [kW]</td>
<td>$\eta$ [-]</td>
</tr>
<tr>
<td>440.5</td>
<td>394.67</td>
<td>379.77</td>
<td>152.1</td>
<td>29.9</td>
<td>26.7</td>
<td>72.8</td>
<td>30.1</td>
<td>0.885</td>
</tr>
</tbody>
</table>
6 Mechanical design

High rotational speed and significant axial thrust (reaction turbine) require an adequate bearing system \[19,20\]. One of the best choices would be to use oil lubricated slide bearings. Unfortunately, one of the design conditions was a hermetic casing in order to minimize the working fluid loss. Lubrication with oil would also...
require a special mechanical sealing and oil separation.

Using magnetic bearings would be an option but this kind of bearing system is the most expensive and does not pay off in a small machine. Another option would be to use static or dynamic (e.g. foil) bearings but the drawback of this solution is its relatively small capacity, which in the case of a reaction turbine (significant axial thrust) requires a balancing piston. A balancing piston is a device that increases the internal leakage loss and requires a longer rotor (shaft) which is particularly undesirable in an overhang design.

Some companies apply slide bearings lubricated by a liquid fraction of the working fluid (greater capacity) but they require an additional cooling cycle in order to avoid local boiling in the lubrication film.

All of the above methods could be considered in a commercial application but in the presented laboratory tests, the study of the flowpath was the priority. For this reason it was decided to apply angular contact ball bearings which are not that durable but offer high stiffness, replacement simplicity, high capacity and are easily available. The 3D model of the turbogenerator designed at IMP PAN is shown in Fig. 15 and the assembly drawing of the whole turbogenerator has been presented in Fig. 16.

![3D model of the SES36 turbogenerator.](image)

7 Internal leakages

The aspect of the internal sealings should be considered not before but after the mechanical design concept is derived, especially in small machines in which the simplicity comes first.

All parts of the casing, generator together with the stator-rotor section and
outlet part, were designed as a vertical split case. This type of design is more tight but it has some limitations for sealing solution application. The axial assembly causes that the full labyrinth sealing cannot be applied. A serial labyrinth must be used instead. This type of sealing with one wall smooth is much less efficient than the full labyrinth because of the fact that kinetic energy of flow passing the gap is not fully decreased within one single sealing chamber [21].

Several variants of a serial labyrinth seal of the rotor were analysed. More sophisticated solutions like coal sealing, abradable sealing or brush sealing were not analysed. The outlet part of rotor shroud was specially shaped to be able to work with labyrinth sealing. The sealing section was designed as a removable part, mounted directly to the outlet pipe, Fig. 17. This solution gives more possibilities for making any changes in the sealing method by replacing the whole unit. All analysed variants of the rotor sealing are presented in Fig. 18.

The gap thickness was established as 0.25 mm (a radial clearance). The single labyrinth thickness is equal to 0.5 mm. A fully theoretical solution for this type of sealing in not possible [21], therefore there are several methods that use correlations and data from experimental works. In general the flow through the
Design and performance study of a small-scale waste...

The function of pressure drop \( \frac{p_2}{p_1} \) (\( p_2 \) – pressure at the sealing outlet) includes the information about the \( \alpha \) coefficient of flow and it is dependent on the shape of the labyrinth seal and the \( a/s \) ratio, where \( a \) is the distance between the seals. Figure 19 shows a chart with the flow function shapes for serial labyrinth...
sealings, data comes from experimental research [21]. The values assumed in the calculations and the results for different variants of seals are summarised in Tab. 6.
Table 6: Parameters and values assumed during sealing calculations.

<table>
<thead>
<tr>
<th>No.</th>
<th>Value</th>
<th>Symbol</th>
<th>Units</th>
<th>Variant 1</th>
<th>Variant 2</th>
<th>Variant 3</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Number of blades</td>
<td>z</td>
<td>–</td>
<td>5</td>
<td>11</td>
<td>6</td>
<td>12</td>
</tr>
<tr>
<td>2</td>
<td>Gap clearance</td>
<td>s</td>
<td>m</td>
<td>0.0025</td>
<td>0.0025</td>
<td>0.0025</td>
<td>0.0025</td>
</tr>
<tr>
<td>3</td>
<td>Distance between blades</td>
<td>a</td>
<td>m</td>
<td>0.001</td>
<td>0.001</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>4</td>
<td>a/s ratio</td>
<td>a/s</td>
<td>–</td>
<td>8</td>
<td>4</td>
<td>8</td>
<td>12</td>
</tr>
<tr>
<td>5</td>
<td>Flow coefficient in gap</td>
<td>a</td>
<td>–</td>
<td>0.7</td>
<td>0.7</td>
<td>0.7</td>
<td>0.7</td>
</tr>
<tr>
<td>6</td>
<td>Shaft diameter</td>
<td>d</td>
<td>m</td>
<td>0.106</td>
<td>0.106</td>
<td>0.106</td>
<td>0.106</td>
</tr>
<tr>
<td>7</td>
<td>Effective area of flow</td>
<td>$A_n$</td>
<td>m²</td>
<td>5.83×10⁻⁵</td>
<td>5.83×10⁻⁵</td>
<td>5.83×10⁻⁵</td>
<td>5.83×10⁻⁵</td>
</tr>
<tr>
<td>8</td>
<td>Pressure at the sealing inlet</td>
<td>$p_1$</td>
<td>Pa</td>
<td>492000</td>
<td>492000</td>
<td>492000</td>
<td>492000</td>
</tr>
<tr>
<td>9</td>
<td>Pressure at the sealing outlet</td>
<td>$p_2$</td>
<td>Pa</td>
<td>220000</td>
<td>220000</td>
<td>220000</td>
<td>220000</td>
</tr>
<tr>
<td>10</td>
<td>Specific volume before sealing</td>
<td>$\nu$</td>
<td>m³/kg</td>
<td>0.0312</td>
<td>0.0312</td>
<td>0.0312</td>
<td>0.0312</td>
</tr>
<tr>
<td>11</td>
<td>Pressure ratio</td>
<td>$p_2/p_1$</td>
<td>–</td>
<td>0.447</td>
<td>0.447</td>
<td>0.447</td>
<td>0.447</td>
</tr>
<tr>
<td>12</td>
<td>Square of the flow function value</td>
<td>$\Phi^2$</td>
<td>–</td>
<td>0.19</td>
<td>0.16</td>
<td>0.17</td>
<td>0.08</td>
</tr>
<tr>
<td>13</td>
<td>Flow function value</td>
<td>$\Phi$</td>
<td>–</td>
<td>0.436</td>
<td>0.4</td>
<td>0.412</td>
<td>0.283</td>
</tr>
<tr>
<td>14</td>
<td>Mass flow of the leak</td>
<td>$m_n$</td>
<td>kg/s</td>
<td>0.101</td>
<td>0.093</td>
<td>0.095</td>
<td>0.065</td>
</tr>
</tbody>
</table>

In each sealing geometry seals were assumed sharp. Results show similar values of leak. It is possible to obtain better results, to decrease the flow rate of leak but the number of seals and the distance between them should be increased, or the gap clearance should be decreased. More seals means that the sealing module and the rotor shroud would be longer. This scenario is limited by strength and dynamic conditions. It is important to realise that this method can introduce some errors into the values of flow because this algorithm is not dependent on the rotor rotational speed and the real shape of the labyrinth sealing. The calculations were verified by another method, where leakage mass flow rate is extracted directly like for full labyrinth sealing, than a correction coefficient is introduced, suitable for proper seal geometry [22]. The formula

$$G = 0.933\alpha\mu\gamma F_p \sqrt{\frac{p_1}{\nu_1}},$$

(3)
takes into account geometry of the seal where: $\alpha$ – relative flow through the sealing, function of number of seals and inlet-outlet pressure ratio, $\mu$ – coefficient.
of leakage through the gap, dependent on the geometry of the seal (experimental data), \( \nu \) – correction factor for half labyrinth sealing, function of number of seals and the coefficient of kinetic energy utilization by steam passing through following spaces between seals, \( F_p \) – geometrical area of the sealing gap.

The results from this algorithm are slightly different compared to the previous one and the mass flow rate is higher in each analysed case, Tab. 7. It is important to underline that both methods are based on empirically derived factors and averaged experimental data thus errors can be expected.

Table 7: Results from second algorithm.

<table>
<thead>
<tr>
<th>Value</th>
<th>Symbol</th>
<th>Units</th>
<th>Variant 1</th>
<th>Variant 2</th>
<th>Variant 3</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow rate</td>
<td>( m_n )</td>
<td>kg/s</td>
<td>0.13</td>
<td>0.132</td>
<td>0.125</td>
<td>0.090</td>
</tr>
</tbody>
</table>

Figure 20: CFD results for the internal sealing domain.

In order to investigate the leakage by one more method a CFD analysis has been performed taking into account the labyrinth sealing domain (only for variant 1), Fig. 20. The obtained mass flow rate of the leakage is equal to 0.117 kg/s. This value is almost exactly between the results given by the two empirical methods. It must be seen, however, that the significant part of the pressure drop occurs also upstream of the labyrinth during the centripetal flow. This effect is caused by the radial equilibrium formed in this part of the domain as the fluid is in rotational motion. Thus, it can be concluded that if the CFD results are reliable, then the empirical methods underestimate the value of the leakage. What is more, the mass flow rate of the internal leakage is almost 10% of the total mass.
flow rate through the machine. It means that the work done by the fluid on the rotor is almost 10% less than without the leakage and that corresponds to the similar drop in the efficiency.

8 Conclusions

A case study of a small turbogenerator design process has been presented. Two variants of radial-axial turbine flowpaths working with SES 36 fluid have been considered. Both variants reveal a relatively high gas dynamic efficiency equal to about 88% in each case. However, single stream turbine is preferable because it is a simpler design, cheaper to manufacture and safer if considered from the point of rotodynamics.

The concept of the mechanical design has been also briefly described, mainly from the point of view of the bearing selection. The set of angular contact ball bearings has been selected as an advantageous option from the point of view of a laboratory test rig.

The analysis of internal leakages has been described in more detail. It can be seen that the internal sealing is a crucial component, especially in small machines. Despite a relatively small radial clearance between the rotor and the casing labyrinth the internal leakage is about 10% of the main flow ratio. It corresponds to a similar drop in the turbine stage efficiency. Perhaps, for small cogeneration units more sophisticated sealing technologies could be beneficial.

Received in July 2016

References


ISSN 0079-3205 Transactions IFFM 133(2016) 145–162


Notes for Contributors

TRANSACTIONS OF THE IFFM publishes original papers which have not previously appeared in other journals. The journal does not have article processing charges (APCs) nor article submission charges. The language of the papers is English. The paper should not exceed the length of 20 pages. All pages should be numbered. The plan and form of the papers should be as follows:

1. The heading should specify the title (as short as possible), author, his/her complete affiliation, town, zip code, country. Please indicate the corresponding author and his e-mail address. The heading should be followed by Abstract of maximum 15 typewritten lines and Keywords.

2. More important symbols used in the paper can be listed in Nomenclature, placed below Abstract and arranged in a column, e.g.:
   
   \[ u \quad \text{velocity, m/s} \]
   \[ v \quad \text{specific volume, m/kg} \]
   etc.

   The list should begin with Latin symbols in alphabetical order followed by Greek symbols also in alphabetical order and with a separate heading. Subscripts and superscripts should follow Greek symbols and should be identified with separate headings. Physical quantities should be expressed in SI units (Système International d’Unités).

3. All abbreviations should be spelled out the first time they are introduced in text.

4. The equations should be each in a separate line. Standard mathematical notation should be used. All symbols used in equations must be clearly defined. The numbers of the equations should run consecutively, irrespective of the division of the paper into sections. The numbers should be given in round brackets on the right-hand side of the page.

5. Particular attention should be paid to the differentiation between capital and small letters. If there is a risk of confusion, the symbols should be explained (for example small \( c \)) in the margins. Indices of more than one level (such as \( B_{f_a} \)) should be avoided wherever possible.

6. Computer-generated figures should be produced using bold lines and characters. No remarks should be written directly on the figures, except numerals or letter symbols only. The relevant explanations can be given below in the caption.

7. The figures, including photographs, diagrams, etc., should be numbered with Arabic numerals in the same order in which they appear in the text. Each figure should have its own caption explaining the content without reference to the text.

8. Computer files on an enclosed disc or sent by e-mail to the Editorial Office are welcome. The manuscript should be written as a MS Word file – *.doc, *.docx or LaTeX file – *.tex. For revised manuscripts after peer review, figures should be submitted as separate graphic files in either vector formats (PostScript (PS), Encapsulated PostScript (EPS), preferable, CorelDraw (CDR), etc.) or bitmap formats (Tagged Image File Format (TIFF), Joint Photographic Experts Group (JPEG), etc.), with the resolution not lower than 300 dpi, preferably 600 dpi. These resolutions refer to images sized at dimensions comparable to
those of figures in the print journal. Therefore, electronic figures should be sized to fit on single printed page and can have maximum 120 mm x 170 mm. Figures created in MS Word, Excel, or PowerPoint will not be accepted. The quality of images downloaded from websites and the Internet are also not acceptable, because of their low resolution (usually only 72 dpi), inadequate for print reproduction.

9. The references for the paper should be numbered in the order in which they are called in the text. Calling the references is by giving the appropriate numbers in square brackets. The references should be listed with the following information provided: the author’s surname and the initials of his/her names, the complete title of the work (in English translation) and, in addition:
   (a) for books: the publishing house and the place and year of publication, for example:


   (b) for journals: the name of the journal, volume (Arabic numerals in bold), year of publication (in round brackets), number and, if appropriate, numbers of relevant pages, for example:


For works originally published in a language other than English, the language should be indicated in parentheses at the end of the reference. Authors are responsible for ensuring that the information in each reference is complete and accurate.

10. As the papers are published in English, the authors who are not native speakers of English are obliged to have the paper thoroughly reviewed language-wise before submitting for publication.

**Manuscript submission**  Manuscript should be mailed to the Transactions of the Institute of Fluid-Flow Machinery Editorial Office (Wydawnictwo IMP PAN) at the Szewalski Institute of Fluid-Flow Machinery PAS, Fiszera 14, 80-231 Gdańsk, Poland, or (electronically) sent as e-mail attachment to the Editorial Office at redakcja@imp.gda.pl. Three hard copies of the manuscript including figures, photographs, diagrams and tables should be submitted simultaneously with the electronic copy. Manuscript submitted only electronically (document or source file) must be supplied with its PDF file.

The illustrations may be submitted in color, however they will be printed in black and white in the journal, so the grayscale contributions are preferable. Therefore, the figure caption and the entire text of the paper should not make any reference to color in the illustration. Moreover the illustration should effectively convey author’s intended meaning when it is printed as a halftone. The illustrations will be reproduced in color in the online publication.

**Further information** All manuscripts will undergo some editorial modification. The paper proofs (as PDF file) will be sent by e-mail to the corresponding author for acceptance, and should be returned within two weeks of receipt. Within the proofs corrections of minor and typographical errors in: author names, affiliations, articles titles, abstracts and keywords, formulas, symbols, grammatical error, details in figures, etc., are only allowed, as well as necessary small additions. The changes within the text will be accepted in case of serious errors, for example with regard to scientific accuracy, or if authors reputation and that of the journal would be affected. Submitted material will not be returned to the author, unless specifically requested.
A PDF file of published paper will be supplied free of charge to the Corresponding Author.

Submission of the manuscript expresses at the same time the authors consent to its publishing in both printed and electronic versions.

**Transfer of Copyright Agreement**  Submission of the manuscript means that the authors automatically agree to assign the copyright to the Publisher. Once a paper has been accepted for publication, as a condition of publication, the authors are asked to send by email a scanned copy of the signed original of the Transfer of Copyright Agreement, signed by the Corresponding Author on behalf of all authors to the Managing Editor of the Journal. The copyright form can be downloaded from the journal's website at http://www.imp.gda.pl/transactions-of-iffm/ under Notes for Contributors.

*The Editorial Committee*