Numerical analysis of centrifugal compressor operating in near-surge conditions

M Kulak, F Grapow and G Liśkiewicz
Institute of Turbomachinery, Lodz University of Technology, Łódź, Poland
E-mail: michal.kulak@p.lodz.pl

Abstract. In order to reproduce the near-surge operation of centrifugal compressor a series of numerical simulations in ANSYS CFX have been conducted. Different settings of boundary conditions have been investigated in purpose of providing a combination allowing to model surge without non-physical flow constraints. This, in turn, would be a step towards modelling of anti-surge system. As the conclusions, it was observed that there is no possibility of simulating near-surge working regime without mass flow rate definition as a boundary condition. However, it is achievable to determine a throttle characteristic, which can be used as a function defining the outlet pressure and indirectly the mass flow rate. In case of transient simulations, it would allow to analyse both steady and unsteady regimes of the flow in a centrifugal compressor.

1. Introduction

1.1 The surge in centrifugal compressors
The most dangerous flow instability occurring in centrifugal compressors is the surge. It occurs when the mass flow rate is decreased below its critical value. The surge is known as low-frequency fluctuations of the pressure and the mass flow rate that in most severe cases might even lead to reverse flows within the compression system [1]. Surge always leads to significant decrease of the machine performance but is also capable of destroying compressor in just few seconds. This is due to the fact that flow oscillations generate high thermal and mechanical loads acting on machine structure and also increase blades vibration level which can be reason of a fatigue fracture [2]. Moreover, the surge is a global instability affecting whole compression system including other components of industrial rig like pipes, reactors etc. [3]. Undisturbed operation of the compressor is a key in generating incomes in most of industries and this is why avoiding or suppressing of this phenomenon is essential. The setting of so-called the surge margin is the most widely used method of prevention from the surge. Compressor operating range is limited by automatized system which keeps operational point sufficiently far from the surge limit. It is usually set somewhere close to 10%-15% away from the surge limit. This results in narrowing the operation range of the machine.

It is also uneasy to investigate surge on real machines because of its destructive nature and the surge limit is in many cases set based on theoretical calculations. This leaves a space for CFD analysis that might be more suitable for that application. Numerical analysis allows to investigate the surge margin without causing risk of machine failure. Nevertheless, it is very prone to non-physical conditions such as the turbulence model, timescale setting and many other simulation parameters. It is also not clear which boundary conditions are best in reproducing the conditions present in the compression system affected by a global flow instability such as the surge.
1.2 CFD analysis of an unstable centrifugal compressor operation

In literature, several papers considering CFD analysis of an unstable, near-surge compressor operation can be found. Most commonly it is an unsteady RANS (URANS) simulation implemented in commercial codes like ANSYS CFX or Fluent, however, the inlet and the outlet boundary conditions vary in wide range. In [4] set of boundaries recommended by CFX modelling guide [5] has been used. Those settings include constant mass flow rate on the inlet and constant static pressure on the outlet of the compressor. Advantages of such approach are robustness and full control of operational point while they unnecessarily stiffen the simulation and may introduce uncertainties. Those settings have been partially reversed in [6] where the mass flow rate is set at the outlet and at the inlet the total pressure is set to preserve real energy level of inflowing air. In this case the mass flow may still reveal some unnatural flow structures but the inlet boundary ensures better energy conservation. The simulations without direct fixing of the mass flow rate have been described in the literature. For example, in [7], the total pressure is set at the inlet, while the static pressure is set at the outlet. In [8] the opening boundary with pressure dependent of plenum mathematical model is used at the outlet in pair with total pressure boundary at the inlet. Despite the fact that there is no direct setting of the mass flow rate in the CFX model, the constant mass flow rate is defined as an outlet boundary in equations describing the plenum.

Another direction of researching near-surge operating conditions is via in-house made codes like those presented in [9–12]. In all of those papers the total pressure is set as the inlet condition. In [9,11] the static pressure is defined at the outlet while in [10,12] more sophisticated pressure function dependent on plenum model has been used. In neither of mentioned papers the outlet plenum has been modeled and in only few of them the inlet pipe was added. Very often the boundary is set not even at the end of volute but at the outlet of diffuser. Only one of those studies contained surge analysis [8] and the remaining ones - the near-surge unsteady flows. It is caused by the fact that it is difficult to enter the surge region without the mass flow definition at inlet/outlet boundary. This is why those simulations do not provide clear information whether they are suitable for the surge investigation.

Next to RANS simulations also few LES simulations have been performed [13–15]. All of them had mass flow rate at inlet and static pressure at outlet as boundaries. Also inlet [13–15] and outlet [13,15] pipe was included in simulations. The precision of this type of simulation is considered as a considerable advantage over other types of simulations but they are highly time and resource consuming which makes them unpractical for everyday use.

1.3 Aim of study

In this paper CFD simulations of the near-surge compressor operation using commercial ANSYS CFX solver have been conducted. Six settings of boundary conditions have been investigated in purpose of providing a combination allowing to model surge in centrifugal compressor without non-physical flow constraints. Findings of such a study would be very useful in development of numerical models aiming at fault-free detection of the surge margin.

2. Method

The geometry of the investigated centrifugal compressor is a virtual model of an experimental test stand located at the Institute of Turbomachinery, Lodz University of Technology, Poland. The test rig is constructed using impellers and casing of a large industrial compressor VRK-3. The computational domain consists of six elements, which are presented in Figure 1.

Flow enters the inlet pipe (1) with a defined inlet boundary condition, and throughout the converging nozzle (2) it enters an impeller (3). Its geometry was reconstructed from a 3D laser scan of compressor stage mounted on the test rig. From the impeller exit, the air goes through the parallel wall, a vaneless diffuser (4), afterwards being collected by a spiral volute (5). Afterwards, the straight pipe with a contraction is modelled (6) at the end of which the outlet boundary condition is defined.
Figure 1. Computational domain.

The CFD simulations have been performed with ANSYS CFX 18.2. The computational domain consisted of around 2.5 million elements (hexahedral in inlet and outlet pipes as well as rotor and tetrahedral in remaining regions). A mesh independence study was performed according to [16]. GCI (Grid Convergence Index) value for majority of monitored parameters was in a satisfactory level (<1%) even for the coarse mesh, therefore it was chosen as a good compromise between results quality and computation time. Shear Stress Transport turbulence model was applied, along with high resolution advection scheme and turbulence equations spatial differencing. As for the heat transfer, the kinetic energy effects were of importance as well as transport of enthalpy, therefore the total energy approach was used – a static, ambient temperature was set at 293 K. The rotor domain was set to rotate at 15855 rev/min. At the inlet pipe/rotor and rotor/diffuser connections frozen rotor interfaces were applied including a GGI (General Grid Interface) mesh connection method.

In this study six sets of boundary conditions combinations were analysed – all of them are collected and presented in Table 1. All sets will be described in details in next section.

Table 1. Comparison of applied boundary conditions.

| Set no. | Simulation type | Inlet boundary condition | Outlet boundary condition |
|---------|-----------------|--------------------------|--------------------------|
| 1       | Steady State    | Total pressure           | Mass flow rate           |
| 2       | Steady State    | Total pressure           | Opening                  |
| 3       | Steady State    | Total pressure           | Static pressure          |
| 4       | Steady State    | Mass flow rate           | Static pressure          |
| 5       | Steady State    | Total pressure           | Static pressure as a function |
| 6       | Transient       | Total pressure           | Static pressure as a function |

3. Results

Amongst combinations of inlet and outlet boundary conditions commonly used in ANSYS CFX, velocity/mass flow at inlet and static pressure at outlet is considered as the most robust option [5]. Nevertheless, the simulation of the surge is much different from the classic simulation at the nominal point. In the case of global instability one may expect that the pressure and mass flow rate will fluctuate to a large extent not only within the compressor, but also in the plenum and the inlet pipe. Therefore, the inlet and outlet boundary condition would be influenced by this fluctuation. The response of a boundary condition to those changes is crucial in obtaining a physical model.
It is a considerable simplification to assume the fixed value of mass flow entering the computational domain. Therefore, a set of different boundary conditions combinations (see Table 1) was tested in search of a one, allowing to model unstable working conditions in near-surge region.

Set 1 incorporated total pressure at the domain inlet and the mass flow condition at the outlet. It is considered as rather robust boundary conditions combination but may introduce non-physical phenomena when the flow should be reversed. In set 2 the mass flow outlet condition was replaced by pressure condition of opening (entrainment), enabling the fluid to both enter and leave the domain. Velocity distribution at the plane in the axis of outlet pipe is presented in Fig. 2. Additionally, the streamlines in whole computational domain are included.

Set 3 included static pressure definition at the domain outlet. As it is described in [5] this combination is valid, with a mass flow as a part of solution. However, the solution is sensitive to initial conditions. Again, the velocity distribution and streamlines are presented in Fig.3. At that point, the decision was made to introduce a function for modelling the value of static pressure at the outlet, resulting from the value of the mass flow (equivalent to mimicking the throttle performance curve). In order to apply this approach, an additional step was necessary - boundary combination presented in set 4 was introduced. It consisted of mass flow defined at the inlet and static pressure at the outlet. This simulation (although considered as over-stabilized) provided a values of pressure \( p_{pre} \) and mass flow rate \( \dot{m}_{pre} \). They were used to prepare a definition of throttle function for calculation of \( p_{sim} \) on the basis of value of the mass flow rate \( \dot{m}_{sim} \). A function in the following form was introduced at the outlet BC in a pre-final set 5:

\[
p_{sim} = \frac{p_{pre}}{\dot{m}_{pre}^2} \dot{m}_{sim}^2
\]

Throughout the solution, monitored parameters such as mass flow rate or pressure ratio presented an oscillations of considerable amplitude, therefore an influence of various values of timescale applied locally was also studied. ANSYS CFX enables different time scales to be applied in different regions of the calculation domain – the smaller time scales are applied to regions where flow has a higher velocity [5]. Regardless of the local timescale choice, the average mass flow rate value was close to the expected one (see Fig. 4).
Finally, the outlet pressure defining function was incorporated into transient solution (set 6). The amount of iterations per one blade passage was set to 20. The flow parameters presented considerably lower fluctuations but at the cost of significant simulation time increase.

Figure 5 collects above mentioned results (excluding set 1 – non convergent solutions and set 4 – used only as a data source) and combines them with the results of preliminary simulations (series of numerical investigations performed on a simplified geometry including only rotor, diffuser and volute, in order to study a performance of a centrifugal compressor in different working conditions). Presented trendline is a fourth degree polynomial.

4. Discussion
Taking into consideration set 1, one can conclude that settling the amount of the medium exiting the domain leads to non-convergence of numerical solution in case of the centrifugal compressor. In reality the mass flow is regulated by the valve at some distance downstream the volute outlet, what could be
utilized by introducing a throttle function (working point at the intersection of throttling valve curve and compressor characteristic).

![Figure 5. Pressure ratio as a function of mass flow rate – simulations results.](image)

Introducing an opening condition at the outlet (set 2), enabling the fluid to flow in and out of the domain, allowed to relax this boundary, resembling experimental or real life conditions. As this approach resulted in correct flow representation only for stable working regime (mass flow in the range of 0.9 – 1.0 kg/s) it was decided to test yet another approach.

For set 3, similarly to set 2, it was possible to simulate correctly the working conditions for stable flow regime. For static pressure at the outlet higher than ca. 135 000 Pa, simulations did not converge (with a sudden mass flow rate drop to 0 and flow blockage). This fact confirmed that the steady state solutions without varying pressure condition at the outlet are able to provide reliable results only for stable work region. Steady state simulations without direct mass flow (set 2 and 3) were not capable of reaching peak of the compressor curve so were far from even near-surge conditions. Similar observation was made in [7]. In mentioned paper a transient simulation with pressure at both the inlet and outlet boundary condition were conducted to reach the near-surge operation. In this article this step was skipped because definition of pressures on both inlet and outlet cannot be used to enter the surge region.

The only discussed paper with surge simulation is [8]. In this paper an indirect mass flow definition was used in form of outlet pressure function based on mathematical plenum model in which mass flow rate is one of the boundary conditions. Similar operation was made in sets 5 and 6 were pressure is defined as function of the mass flow rate depending on the throttle characteristic. This allows to abandon unphysical mass flow condition and use pressure function instead. Those simulations, however, are very sensitive to the time scale and pseudo time scale since they are dependent of mass flow rate from the previous iteration. Too big time step leads to oscillatory character of simulation.

Set 5 with steady state simulation and throttle characteristic set at the outlet can be used to simulate stable and near surge iteration but only when the mean value is needed. Its oscillatory character may be misleading, especially when one needs to analyse a flow field or a particular physical quantity at the particular iteration.

Set 6 is a transient version of the set 5. It is most universal and complete variant useful in stable and surge simulations. As it was mentioned before it is highly sensitive to the time step which has to be
small enough. In literature for stable and unstable compressor operation with constant values of the boundary conditions, value of time step is chosen in a way which guarantees 15 to 20 iterations per blade passage [7,17,18]. It is time and resource consuming for a rotor with many blades or with high rotational speed but it also guarantees that function based on values taken from previous time step will operate with satisfactory stability.

One can notice that according to Fig. 5 simulation of compressor with and without inlet and outlet pipe differs from each other significantly. It may be the effect of pipe contraction located close to volute outlet. During simulations this region was highly unstable and generated local areas of stagnation or even led to errors in simulation. Additional study of this phenomenon is needed because contraction in the vicinity of volute outlet has not been modelled in the compressor unstable operation in the state-of-the-art publications. What is more, very often CFD simulations overestimate pressure rise with respect to the experimental values. This might be an independent confirmation of the fact, that the inlet and the outlet pipes should be included in computational models of the near-surge simulations.

Many researches of anti-surge systems are based on calculations using Greitzer model and similar one-dimensional mathematical models [19–21]. Despite the fact that surge is considered to be one dimensional instability, the influence of anti-surge system may lead to developing local two or three dimensional flow structures which cannot be observed in simple mathematical model. In contrast to mentioned models, CFD gives almost unlimited possibilities of implementing different anti-surge system actuators at different locations. Furthermore, the three dimensional flow analysis in any region of domain can be conducted. That can be considered as significant advantage of CFD approach to designing anti-surge systems requiring wide variety of possible modifications and easy observation methods. It is also important to point out the fact, that a given capability and level of accuracy are acceptable for designing process – the industry still accepts shortcomings of RANS (in contrary to cost, time and user skills necessary for scale-resolving computations) [22].

5. Conclusions

The presented research is a part of a project aiming at design and construction of anti-surge systems of different types. It includes both: experimental studies on an industrial compressor and numerical analysis of its virtual model. Eventually, it is expected to introduce the anti-surge systems concepts into the numerical simulations of centrifugal compressors. However, it is crucial to be able to obtain the surge or the near-surge working regime beforehand. In terms of analysing combinations of boundary conditions allowing to do that one can conclude as follows:

- it is difficult to enter the surge region without the mass flow definition;
- the direct mass flow rate definition as the outlet boundary condition may lead to a non-convergence of the steady-state solution;
- both opening pressure (entainment) and static pressure condition at the outlet allows the simulations to converge, however only in the range of the stable flow regime;
- it is possible to define a throttle characteristic, which can be used as a function defining the outlet pressure; it requires the knowledge about the mass flow rate and equivalent pressure at single operational point, which can be obtained in a disassociated simulation (with a most robust boundary conditions combination – mass flow rate at inlet and static pressure at outlet)
- the outlet pressure function allows to indirectly define the mass flow rate, not as a fixed value, enabling the flow fluctuations to occur;
- transient simulation with an application of described outlet pressure function is most universal because it allows to analyse both steady and unsteady regimes of the flow in a centrifugal compressor;

Acknowledgements

This work was funded by the National Centre for Research and Development (LIDER/447/L-6/NCBR/2015).
References

[1] Willems F and Jager B de 1998 Modeling and control of rotating stall and surge: an overview IEEE Int. Conf. Control Appl. 1 331–5
[2] Meuleman C, Willems F, de Lange R and de Jager B 1998 Surge in a low-speed radial compressor Int. Gas Turbine Aeroengine Congr.
[3] de Jager B 1995 Rotating stall and surge control: A survey Decision and Control, 1995., Proceedings of the 34th IEEE Conference on pp 1857–62
[4] Shahin I, Gadala M, Alqaradawi M and Badr O 2014 Unsteady CFD Simulation for High Speed Centrifugal Compressor Operating Near Surge Volume 2D: Turbomachinery (ASME)
[5] ANSYS 2016 CFX-Solver Modelling Guide--Release 17.0
[6] Ding Y, Groth C, Kacker S and Roberts D 2005 CFD Analysis of Off-design Centrifugal Compressor Operation and Performance 2006-Int-ANSYS-Conf/252
[7] Dickmann H-P, Secall Wimmel T, Szwedowicz J, Filsinger D and Roduner C H 2006 Unsteady Flow in a Turbocharger Centrifugal Compressor: Three-Dimensional Computational Fluid Dynamics Simulation and Numerical and Experimental Analysis of Impeller Blade Vibration J. Turbomach. 128 455
[8] Guo S, Chen H, Zhu X and Du Z 2011 Numerical Simulation of Surge in Turbocharger Centrifugal Compressor - Influence of Downstream Plenum Proc. ASME Turbo Expo GT2011-451 1–12
[9] Niazi S, Stein A and Sankar L 1998 Development and Application of a CFD Solver to the Simulation of Centrifugal Compressors AIAA Pap. 934
[10] Stein A, Niazi S and Sankar L N 2000 Numerical analysis of stall and surge in a high-speed centrifugal compressor 38th Aerospace Sciences Meeting and Exhibit p 226
[11] Stein A, Niazi S and Sankar L N 2001 Computational Analysis of Centrifugal Compressor Surge Control Using Air Injection J. Aircr. 38 513–20
[12] Stein A, Niazi S and Sankar L 2000 Numerical studies of stall and surge alleviation in a high-speed transonic fan rotor 38th Aerospace Sciences Meeting and Exhibit (Reston, Virginia: American Institute of Aeronautics and Astronautics)
[13] Hellstrom F, Gutmark E and Fuchs L 2012 Large Eddy Simulation of the Unsteady Flow in a Radial Compressor Operating Near Surge J. Turbomach. 134 051006
[14] Shahin I, Gadala M, Alqaradawi M and Badr O 2015 Large Eddy Simulation for a Deep Surge Cycle in a High-Speed Centrifugal Compressor With Vaned Diffuser J. Turbomach. 137
[15] Semlitsch B, Jyothishkumar V, Mihaescu M, Fuchs L and Gutmark E J 2013 Investigation of the surge phenomena in a centrifugal compressor using large eddy simulation ASME International Mechanical Engineering Congress and Exposition, Proceedings (IMECE) vol 7 A pp 1–10
[16] Celik I B, Ghia U and Roache P J 2008 Procedure for estimation and reporting of uncertainty due to discretization in CFD applications J. fluids Eng. ASME 130
[17] Ljevar S 2007 Rotating Stall in Wide Vaneless Diffusers (Eindhoven University of Technology) Doctoral dissertation
[18] ANSYS 2015 Fluent User’s Guide
[19] Backi C J, Gravdahl J T and Skogestad S 2016 Robust control of a two-state Greitzer compressor model by state-feedback linearization 1226–31
[20] Willems F and de Jager B 1998 Active compressor surge control using a one-sided controlled bleed/recycle valve IEEE Conference on Decision and Control vol 3 pp 2546–51
[21] Greitzer E 1976 Surge and rotating stall in axial flow compressors—Part I: Theoretical compression system model J. Eng. Gas Turbines Power 98 190-198
[22] Duraisamy K 2017 Status, Emerging Ideas and Future Directions of Turbulence Modeling Research in Aeronautics NASA Tech. Memo. 1–9