Numerical investigation on flow distortion in a vertical inline pump

Yajing Tang, Qiaorui Si, Xingcheng Gan
National Research Center of Pumps, Jiangsu University, Zhenjiang, Jiangsu, China
E-mail: siqiaorui@ujs.edu.cn

Abstract. The vertical inline pump is a centrifugal pump with special structures of curved inlet pipe before the impeller, and it is widely used in where the constraint is installation space. However, the elbow inlet with different angles and cross sections also deteriorates the impeller inflow and results in flow distortion in the impeller. For the purpose of further investigation of flow distortion in the vertical inline pump, the unsteady Reynolds averaged Navier-Stokes equations were solved by the commercial CFD code with the shear stress turbulence model. The normalized coefficients of inlet velocity, pressure, helicity, and non-uniformity were applied to quantitatively analyze the flow distortion in the pump. The computational results showed a great agreement with the experimental results. Under the nominal and overload conditions, the flow separation and counter-vortex pair were found on the outer side of the inlet pipe and the intensity of these phenomenon increases along with the rise of flow rate. However, the flow recirculation was observed under the part-load condition which becomes the major reason for the flow distortion in the inline pump.

1. Introduction

1.1 Background
The vertical inline pump is a centrifugal pump with special structures that the inlet and outlet of the pump are on the same line and usually have the same size. It is invented to pressurized transport and increase the outflow pressure It has the advantages of compact structure, small volume, easy installation, and low construction cost, so it is widely utilized in where with limited installation space, such as pump house. However, the elbowed inlet pipe with different angles and cross-sections also deteriorates the inlet conditions of the impeller, which results in serious flow distortion in the vertical inline pump such as recirculation, secondary flow generation at the impeller inlet and separation vortices in the impeller channel [1].

1.2 Literature review
The secondary flow field refers to the regular accompanying motion of the viscous fluid caused by the curve motion under a certain mainstream speed and a certain geometric boundary condition [2]. Among the existing research on secondary flow of the elbow, it is mainly the experimental research and computational fluid dynamics research. Taylor measured the important physical quantities of laminar and turbulent flow in the flow field of 90 degree elbow pipe, which provided important reference for subsequent research [3].
Xu Cheng et al. used the hot wire tachometer system to test the separated flow field in the elbow.
The results show that the separation zone of the inflow of the elbow is near the exit of the turning section, and the side pressure of the low curvature increases sharply, and the side of the high curvature. The pressure is drastically reduced, which is due to the combination of separation and bending [4]. Shang Hong et al. measured and analyzed the flow field in a 90° circular section elbow. The results show that the turbulent flow energy of the fluid on the low curvature side is larger than that on the high curvature side, and the flow energy in the region with a large velocity gradient is also large, and the energy loss inside the elbow is large. Focusing on the low curvature side, the energy loss is particularly severe on the outflow surface of the curved section [5]. Sudo et al. used the rotating probe technique to measure the flow field in the 90° circular cross-section elbow of 0.87, and obtained the Reynolds stress equivalent map, vector diagram and velocity contour map on the flow section [6]. Caifen Ma et al. (laser Doppler velocimeter) LDV measured the turbulent flow field in a rectangular cross-section elbow. The results show that under turbulent conditions, the lateral secondary flow transports the fluid with less turbulent flow energy from the outer wall. As for the inner wall surface with a large flow, there is a strong energy exchange inside the curve [7].

Jiangshan et al. used RNG $k - \varepsilon$ turbulence model to simulate the flow field in a 90 degree large curvature elbow. The simulation results and experimental data are similar, which proves that RNG $k - \varepsilon$ turbulence model is suitable for the simulation calculation of flow field in elbow [8]. In order to improve the irregular flow inside the elbow and reduce the flow loss, many scholars have carried out related research [9-12]. At present, the installation of the baffle in the elbow becomes the elimination of the secondary flow in the curved section of the elbow.

In rotating machinery, elbows are often used as inlet pipes for fluid media, and the structure of the inlet pipe directly affects the uniformity and overall performance of the inflow of the rotating machinery. Therefore, it is particularly important to study the disturbance of the non-uniform inflow of the inlet pipe to the downstream components. Disturbance to downstream components, Bulten et al. proposed non-uniformity to quantitatively describe the non-uniformity of the velocity flow distribution on the outflow section of a curved flow channel [13]. Sheoran experimentally studied the variation of compressor performance under the disturbance of swirling distortion of the inlet surface. The results show that the overall vortex on the inlet surface has the greatest influence on the performance of the compressor, and the vortex and offset vortex; Both will induce a decrease in compressor efficiency, but the effect on pressure ratio and stability will vary with the type of swirl [14-16]. Duerr uses numerical simulation to capture the secondary flow reverse vortex pair on the inlet surface of the water jet propulsion pump and confirms that the secondary flow reverse vortex pair under different conditions also changes [17]. S. Vagnoli and T. Verstraete studied the influence of the inlet and the straight inlet of the elbow on the internal flow field and performance of the compressor based on the unsteady value analysis method. The results show that at high flow, the elbow enters. The gas form significantly reduces the compressor's compression ratio [18]. Van Esch installed a rectifier in the straight pipe inlet to artificially uniform non-inflow, and compared with the uniform inflow, and found that the inflow distortion affects the radial force of the water jet propulsion pump, but he didn't explain the specific disturbance mechanism [19].

Even though domestic and foreign scholars have made certain achievements in the research of flow distortion phenomena from different viewpoints, few of them can fully grasp the inlet recirculation’s characteristic, mechanism and flow patterns.

### 1.3 Paper organization

In this research, a unsteady numerical investigation was carried out to obtain the inflow distortion patterns in the curved inlet pipe of an industrial vertical inline pump. Six different monitor sections and four normalized parameters were defined to quantitatively study the degree of distortion. Finally, in order to ensure the reliability of the computational results, a validation experiment was adopted.
2. Computation model

This study was carried out on an industrial vertical inline pump with the specific speed of 132 (in Chinese standard, the definition is given in formula (1)) and the main design parameters are shown in Table 1. The flow domain was divided into four parts: inlet pipe, impeller, volute and delivery pipe, as shown in Figure 1.

\[ n_s = \frac{3.65 \times n \cdot \sqrt{Q_d}}{H^{0.75}} \]  

(1)

where:
- \( n_s \): specific speed;
- \( n \): rotating speed, r/s;
- \( Q_d \): design flow rate, m\(^3\)/h;
- \( H \): design head, m.

| Parameters                     | Value  |
|--------------------------------|--------|
| Flow rate, \( Q_d \) (m\(^3\)/h) | 50     |
| Total head, \( H \) (m)        | 20     |
| Rotational speed, \( n \) (rpm) | 2910   |
| Specific speed, \( n_s \)      | 132    |
| Impeller inlet diameter, \( D_1 \) (mm) | 73     |
| Impeller outlet diameter, \( D_2 \) (mm) | 136    |
| Inlet width, \( b_1 \) (mm)    | 34.5   |
| Outlet width, \( b_2 \) (mm)   | 17.8   |
| Inlet vane angle, \( \beta_1 \) (deg) | 28.6   |
| Outlet vane angle, \( \beta_2 \) (deg) | 30.3   |
| Number of blades, \( z \)      | 6      |
| Suction pipe diameter, \( D_s \) (mm) | 80     |
| Delivery pipe diameter, \( D_d \) (mm) | 80     |

**Figure 1. Flow Domain of the Vertical Inline Pump**
3. Numerical methodology

3.1 Computational grids

The meshing process is one of the most important procedures before the numerical simulation because the quality of grids has a great influence on the computational speed and precision. Hence, in this research, a structured cell with multi-block strategies was applied to discretize the flow domain using the commercial pre-treatment software ANSYS ICEM CFD 19.2.

A grid sensitivity study was carried out to ensure the reliability of the calculation. The results of this study are given in Table 2. The head coefficient (the definition is given in formula (2)) of the target pump was going to be stable when the impeller grid number was greater than 0.93 million. Hence, the final grid distribution applied in optimization is shown in Table 3.

The grid conditions are shown in Figure 2. Critical areas such as bends of the inlet pipe, leading edge and tongue were refined better to obtain the flow features. The desired nondimensional wall distance ($y^+$) near the critical surfaces was from 5 to 10, and the maximum value was less than 80, which could satisfy the requirement of the turbulence model.

$$\psi = \frac{2gh}{u_2^2}$$ (2)

where:

- $u_2$: tangential component of impeller outflow velocity, m/s;
- $g$: gravity factor, 9.81 m/s$^2$;
- $H$: head, m.

| Grid Index | Grid Number /million | Head Coefficient |
|------------|----------------------|------------------|
| A          | 0.04                 | 0.893            |
| B          | 0.06                 | 0.874            |
| C          | 0.14                 | 0.878            |
| D          | 0.24                 | 0.895            |
| E          | 0.38                 | 0.896            |
| F          | 0.68                 | 0.898            |
| G          | 0.93                 | **0.900**        |
| H          | 1.08                 | 0.900            |
| I          | 1.32                 | 0.900            |

Table 2. Grid Sensitivity

| Domain | Inlet Pipe | Impeller | Volute | Delivery Pipe |
|--------|------------|----------|--------|---------------|
| Grids  | 1,361,122  | 933,510  | 1,216,305 | 779,544       |

Table 3. Grid Distribution
3.2 Computational setup
The three-dimensional, unsteady Reynolds averaged Naiver-Stokes equations with shear stress transport (SST) turbulence model were solved by commercial CFD code ANSYS CFX 19.2 to obtain flow behavior in the inlet pipe of an industrial vertical inline pump the between 60% and 140% of nominal flow rates ($Q_d$) in the interval of 20%.

The total pressure (stable) was imposed at the inlet of the suction pipe and mass flow rate at the outlet of the delivery pipe. Boundary conditions were as close to the real working conditions in the pump and with the reference pressure as 101.325 kPa. All solid walls are no slip and the interfaces between rotor and stator were set as “Frozen Rotor”. The roughness of walls was set as 25 μm. The high-resolution scheme was adopted for the discretization of the convective and diffusion terms with the second-order backward Euler transient scheme. Residual of continuity and momentum equations are reduced to the magnitude of $10^{-4}$ as a convergence criterion and the same criteria were used for turbulence kinetic energy and dissipation rate.

3.3 Monitor setup
For the purpose of studies in the inlet pipe, six sections along the mean streamline of the inlet pipe were considered as shown in Figure 3. The sections are denoted as the capital alphabet and inner curve(ic) describes the topmost curve of the inlet pipe surface and which is near the pump casing, whereas the outer curve(oc) is the bottom-most surface of the inlet pipe. The total length($l$) of the inlet pipe along the mean streamline is 247mm and $x$ is the distance measured from section A along the mean streamline. The relative position ($x/l$) of each monitor sections are 0, 0.2, 0.4, 0.6, 0.8, and 1 respectively.
3.4 Normalized parameter definition

In order to quantitatively analyze the flow features in the vertical inline pump, three normalized parameters were defined as follow.

(1) Pressure coefficient

\[
C_p = \frac{1}{0.5 \rho u_s^2} \times \frac{1}{N} \sum_{i=1}^{N} (P_i - P_{ref})
\]

where \(C_p\) is the normalized pressure coefficient, \(\rho\) is the density of water, \(P_i(x, y, z, t)\) is the pressure of timestep \(t\) on location \((x, y, z)\), and \(P_{ref}\) is the reference pressure.

(2) Velocity coefficient

\[
C_{v_s} = \frac{1}{v_s} \times \frac{1}{N} \sum_{i=1}^{N} v_i
\]

where \(C_{v_s}\) is the normalized velocity coefficient, \(v_s\) is the average velocity on plane A and \(v_i(x, y, z, t)\) is the velocity of timestep \(t\) on location \((x, y, z)\).

(3) Three Dimensional Helicity

\[
H_y = (\nabla \times \mathbf{v}) \cdot \mathbf{v}
\]

where \(H_y\) is helicity, \(\mathbf{v}\) is the velocity, \(\nabla\) is gradient factor.

4. Result and discussion

4.1 Experimental validation

In order to verify the reliability of the computational results, a validation experiment on the original inline pump was carried out on an open-loop test rig equipped with calibrated equipment for flow rate, suction pressure, delivery pressure, input power, and speed measurement. The experimental pump is shown in Figure 4. The total uncertainties of head and efficiency were less than ±0.2%, and the figure for flow rate was less than ±0.2%. The suction and delivery pressures were measured by pressure transmitter of the corresponding specification, and the flow rate was obtained through an electromagnetic flow meter. The input power was recorded through a Wattmeter and the speed was controlled through a variable frequency drive. The readings were taken at several positions of the delivery valve from shut-off condition to full open. A validation experiment was conducted more than once to ensure the repeatability of the results. The detailed data are shown in Figure 5.

As shown in Figure 5, the computational results showed a great agreement with the experimental results. At the design point, the computational and experimental head were 19.80m and 19.54m, and the figure for efficiency was 72.43% and 77.65%, respectively. The head coefficient \(\psi\) was defined as for
formula (2) and the capacity coefficient $\phi$ was defined as follow.

$$\phi = \frac{Q}{nd_2^3}$$  \hspace{1cm} (6)

where:

$Q$: flowrate, m$^3$/s;

$n$: rotational speed, r/min;

$d_2$: impeller outlet diameter, mm.

4.2 Pressure distribution analysis

The pressure coefficient distribution on the sections of the inlet pipe of the pipeline pump under different flow conditions with the rotating speed of 2910 r/min is shown in Figure 6. Section A-B-C is defined as the first bend, and section D-E-F is the second bend.

It can be observed from Figure 6 that the pressure distribution showed a great non-uniformity with a large pressure gradient. The high-pressure region concentrated on the inner curve of the first bend and the outer curve of the second bend. There is a lateral pressure difference between the upper side and the lower side because of the centrifugal force.

As shown in Figure 6, under the five selected operating conditions, the pressure coefficient declined to the nadir near the inner curve between section C and suction F, and the pressure drop in this area was obvious with respect to the outer side of the first bend because of the effect of the return vortex at the impeller inlet. The low-pressure zone caused by the return vortex had a large influence range and extended along the upper sidewall toward the impeller inlet and the shroud with the decrease of flow rate. In the actual operation condition, it is liable to cavitation, which affects the stability of the pump. Pullan et al. [20] also believe that the low-pressure zone is associated with a vortex that is free from the front end of the impeller.
Under the design flow and large flow conditions, the pressure distribution is similar, and the non-uniform pressure distribution is mainly caused by flow separation.

Figure 6. Pressure Distribution on the Mid Plane

4.3 Velocity distribution analysis
In order to analyze the formation process of the inlet pipe flow distortion, a velocity distribution analysis was adopted. Figure 7 extracts the velocity coefficient distribution on the midplane of the inlet pipe. Figure 8 shows the velocity coefficient distribution of the six sections at 0.6 times the design flow rate. Figure 9 shows the velocity coefficient distribution on section C under the five selected operating conditions.

It can be seen from Figure 7 that, under the five working conditions, the velocity coefficient reached a peak at the outlet of the impeller due to the work of the blades. Under nominal and overload conditions, the high-velocity area mainly concentrated on the inner side of the second curve where was also the low-pressure area. The mainstream had a higher velocity, and the flow separation can be observed on the outer side of the first bend because of the inertia, which extremely increased the hydraulic losses in the inlet pipe.

Under small flow conditions, the flow in the inlet pipe is much more complicated. As shown in Figure 7(a) and (b), a recirculation vortex appeared in the low-pressure region near the upper side of the inlet pipe and it extended toward the inlet. The return vortex in the low-pressure zone causes blockage in the
inlet pipe that resulted in a rapid velocity rise in near section D. However, it also caused the decrease of the flow area, which in turn improved the flow separation on the outer side of the first bend. As shown in Figure 9, the recirculation zone at section C increases as the flow rate decreases.

As shown in Figure 7(a) and (b), under part-load condition, weak flow separation can be found near the trailing edge of the suction side of the blade. When the flow rate is further reduced, the separation phenomenon got much stronger which can be observed on the pressure side of the blades, and the flow recirculation near the impeller outlet can be clearly observed. As shown in Figure 7 (c), (d), and (f), the flow separation on the outer side of the first bend in the inlet pipe was intensified with the flow rate increases.

![Figure 7. Velocity Distribution on the Mid Plane](image)

As shown in Figure 8, the velocity coefficient distribution of section A to section F at 0.6 times the design flow rate was extracted. Figure 8(a) shows that the velocity distribution had a great axisymmetric when it arrived at the inlet of the pipe and it developed to a layered distribution along the main streamline. The velocity coefficient near the inner side was much higher than the outer side. The curvature difference and bending angle outside the elbow inlet elbow cause a lateral pressure difference between the upper side and the lower side which caused secondary flow and form a complex spiral flow with the mainstream.

As shown in Figure 8(b), after the fluid entered the elbow, a pair of typical reverse vortices appeared
on the upper side of the section B. The fluid overcomes the lateral differential pressure under the effect of centrifugal force, and the flow separation occurs on the lower side when flowing through the first bend. As shown in Figure 9(c), a pair of reverse vortices appeared in the low-pressure zone of section C, and a significant backflow vortex appeared in the high-pressure zone because of the difference of the curvature between the upper profile and the lower profile. So, the separation vortex position is opposite to the lower pressure zone of the second bend. Influenced by the upstream, the non-uniformity of inflow of the impeller increased. As shown in Figure 8(e) and Figure 8(f), there were obvious backflow vortices and disturbing micelles on the impeller inlet surface.

The analysis shows that the recirculation zone in the inlet pipe of the inline pump was facing the low-pressure zone, and the fluid around the perturbation micro-cluster may spiral in the circumferential direction or the exhibition direction. Inoue[21] also described a similar low-pressure zone at the recirculation boundary in the study of the compressor's Spike stall precursor and confirmed that the low-pressure zone is the counter-flow of the impeller inside the impeller and then attached to the upstream wall.
(e) Section E  (f) Section F

Figure 8. Velocity Distribution on Different Sections at 0.6 \( Q_d \)

Figure 9. Velocity Distribution on Section C

4.4 Helicity distribution analysis

The regularized helicity analysis method is applied to analyze the strength of the vortex core in the recirculation vortex along the midplane and six transverse sections of the inlet pipe.

Figure 10 shows the helicity distribution in the middle section of the inlet pipe at different flow rates. Figure 11 shows the helicity distribution on section F under the five operating conditions. Figure 12 is a diagram showing the helicity distribution of section A to F of the inlet pipe under 0.6 times the design condition.

Under 0.6 times the design condition, the vortex inside the inlet pipe is mainly composed of the recirculating vortex, as shown in Figure 10, the vortex intensity gradually decreases with the extension process along the upper sidewall; the recirculation vortex is mainly concentrated on the upper side and
has larger intensity, smaller area, and more concentrated distribution. As shown in Figure 12 (a) and (b), the vortex distribution at section A and B of the inlet pipe has a degree of symmetry, and the vortex was concentrated on the boundary with the mainstream concentrated in the central region. The helicity distribution in Figure 11(b) was obviously symmetrically reversed. Due to the influence of the return vortex at section F, the impeller inflow had a large non-uniformity, the flow condition of the rear flow passage was deteriorated, which increased the hydraulic losses and declined the pump performance.

Under 0.8 times the design condition, the separation vortex developed to the outer side of the first bend of the inlet pipe and moved toward the inlet along with the increase of flow rate.

Under the nominal and overload conditions, the flow separation occurs near the outer side of the inlet pipe and a large separation vortex is formed near the outer side of the second bend. The intensity was small, but the affected area is larger. Mainstream focused on the inner side of the second bend and a large velocity gradient was generated. With the increase of flow rate, the intensity of the separation vortex gradually raised, and the influence range extends from the outer side to the middle region of the inlet pipe.

Under nominal condition and overload condition, it can be observed from Figure 11 that the helicity distributions on section F were similar, and there were two significant reverse vortex pairs, where the impeller inlet is prone to cause local radial secondary flow and had little effect on the incoming flow in the inlet pipe. The degree of flow distortion of both sections increased with the increase of flow rate. However, under the small flow rate, there was no obvious reverse vortex pair on section F in most conditions, but it can be seen from Figure 12 that under the design condition of 0.6 times, a distinct reverse vortex pair appeared at section B.
Figure 10. Helicity Distribution on the Mid Plane
Figure 11. Helicity Distribution at Plane F
5. Conclusion

In this research, a numerical investigation based on three-dimensional unsteady Reynolds averaged Navier-Stokes equations was carried out to obtain the flow distortions in the inlet pipe of an industrial vertical inline pump. The 3D case was solved using the commercial CFD code ANSYS CFX 19.2 with the shear stress transport turbulence model and six different monitor sections were created for further flow features.

The computational results showed that there was a great agreement between the numerical simulation and the validation experiment. The flow analysis reported that the major reasons for inflow distortion in the inline pump were separation and secondary reverse vortex pair under the nominal condition and overload condition, and it becomes recirculation under the part-load condition. As an overall summary, the flow patterns in the inlet pipe were not good enough even under the nominal flow rate, hence, it is necessary to redesign the profile of the inlet pipe to obtain better performance with minimized the flow separation and recirculation.

Acknowledgment

This work is supported by National Natural Science Foundation of China (Grant No. 51509108), China Postdoctoral Science Foundation funded project (Grant No. 2016T90422) and National Key Research and Development Program (Grant No. 2018YFB0606103).

Nomenclature

- $b_1$: Impeller inlet width, $mm$
- $b_2$: Impeller outlet width, $mm$
- $D_1$: The diameter of impeller inlet, $mm$
- $D_s$: The diameter of the suction pipe, $mm$
- $D_d$: The diameter of the delivery pipe, $mm$
- $H$: Pump head, $m$
- $n$: Rotating speed of impeller, $rpm$
- $n_s$: Specific speed of the pump
- $Q$: Flow rate, $m^3/s$
The volume flow rate of design flow condition, $m^3/h$

Impeller peripheral velocity at the outlet, $m/s$

Number of blades

Impeller inlet vane angle, degree

Impeller outlet vane angle, degree

The efficiency of the pump

Flow coefficient

Head coefficient

Normalized Pressure coefficient

Normalized Velocity Coefficient

Non-uniformity coefficient

Three-dimensional helicity

Reference

[1] Crane 1957 *Flow of Fluids through Valves, Fittings and Pipe* (New York: Crane Co)
[2] Hao C and Jianwen L 2006 *Secondary Flow Theory* (Changsha: Central South University Press)
[3] Taylor K P, Whitelaw J H, Yiannes M 1982 Curved ducts with strong secondary motion: velocity measurements of developing laminar and turbulent flow *Journal of Fluid Engineering* **104** 350-359
[4] Cheng X and Rongwei G 1994 Experimental investigation for separated flow in three-dimensional bend duct *Acta Aeronautica Et Astronautica Sinica* **15(9)** 1095-1099
[5] Shang Hong 1994 Experimental research on three-dimensional turbulent flow field in 90° circular section bend *Journal of Aerospace Power* **3(9)** 263-266
[6] Sudo K, Sumida M and Hibara H 1998 Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend *Experiments in Fluid* **25** 42-49
[7] Caifen M and Shi F 1990 Experimental research of turbulent flow in a 90° curve *Journal of Engineering Thermophysics* **11(1)** 37-40
[8] Shan J and Jingwei Z 2008 Numerical simulation of inner flow in 90° bending duct of circular-section based on fluent *Chinese Journal of Ship Research* **3(1)** 37-41
[9] Xinghao J and Xianghe P 2011 Numerical simulation and optimization of the flow field of the baffle *Journal of Southwest University (Natural Science Edition)* **3(33)** 139-143
[10] Zhenhua X and Yanrong Z 2009 Optimization of the installation of deflector in 90 degree square section elbow *Journal of basic science and engineering* **4(17)** 566-572
[11] Jianhong M, Xinwei J and Yi Z 2011 The effect of splitters layout at variable cross-section inclined flue on AIG in let flow field *Journal of Zhejiang University (Engineering Science)* **8(45)** 1453-1457
[12] Weiwei Z, Fan L, Xiaofeng Z, et al. 2014 Research progress on secondary flow of curved pipe and current sharing technology of power plant boiler *Journal of engineering for Thermal Energy and Power* **29(6)** 603-609
[13] Bulten, N W H 2006 *Numerical analysis of water propulsion system* (Technical University of Eindhoven)
[14] Sheoran Y, Bouldin B and Krishnan P M 2009 *Advancements in the design of an adaptable swirl distortion generator for testing gas turbine engines* (Orland: American Society of Mechanical Engineers)
[15] Sheoran Y, Bouldin B and Krishnan P M 2012 Compressor performance and operability in swirl
A versatile design of a controlled swirl distortion generator for testing gas turbine engines (Berlin: American Society of Mechanical Engineers)

Investigation of marine waterjet inlets during turning maneuvers (Boca Raton: Florida Atlantic University)

Numerical investigation of inlet distortion on the stall inception radial compressor ASME GT2014-25516

Performance and radial loading of a mixed-flow pump under non-uniform suction flow Journal of Fluids Engineering 131(5) 051101

Origins and structure of spike-type rotating stall. Journal of Turbomachinery 137(5) 051307

Comparative studies on short and long length-scale stall cell propagating in an axial compressor rotor Journal of Turbomachinery 123(1) 24-30.