

NUMERICAL STUDY OF UNSTEADY FLOW CHARACTERISTICS IN REGENERATIVE PUMP

Tarek A. Meakhail

*Mechanical Power Engineering Department, High Institute of Energy,
South Valley University, Aswan, Egypt*

(Received March 25, 2007 Accepted April 21, 2007)

Regenerative pumps are turbomachines that achieve lower mass fluxes but higher-pressure differences than comparable pumps at the same circumferential velocity. The construction of these machines is very simple and inexpensive. On the other hand their efficiency does not exceed 45 % and they are noisy. To improve efficiency and reduce noise generation, a detailed knowledge of the unsteady flow in the machine is essential. In this work, we present unsteady flow characteristics of a typical regenerative pump based on numerical results obtained by using CFX-Tascflow code. The numerical results show highly unsteady pressure fluctuations around the stripper near both inlet and outlet ports. The lower pressure predicted near the inlet port indicates that the machine is prone to suffer cavitation problems. The amplitude of pressure fluctuation near the inlet is much higher than the outlet.

KEYWORDS: *Regenerative pump, Unsteady flow, Sliding interface, CFD.*

1. INTRODUCTION

Regenerative pumps are widely used as pressure increasing devices. The principles of these machines are almost like those of torque converters. As shown in Fig. 1, the impeller derives the circulating flow in the side channel from the inlet port to the outlet port. The inlet and outlet ports are blocked by a so-called stripper. In comparison to other pumps, the regenerative pumps are of simple construction and cheap to build and they generate higher pressure difference. They are disadvantageous in the sense that the efficiency does not exceed 45 percent and that they are very noisy.

In spite of the low efficiency of regenerative pumps, they have widely been used for many applications where low specific speeds are required, i.e., for low flow rates at high-pressure heads, such as in automotive and aerospace fuel pumping, chemical and oil processing industry, process and engineering plants, etc.

The flow in regenerative pump can be described as follows. When the impeller starts rotating, the fluid in the blade-chambers moves with the same circumferential velocity as the impeller, but the fluid in the side-channel is still at rest or moves slowly. The fluid in the impeller chambers starts moving in the radial direction as a result of the centrifugal force leaving the impeller and enters the side-channel. This imparts

momentum to the fluid in the side-channel and consequently the so-called ‘circulating flow’ is established.

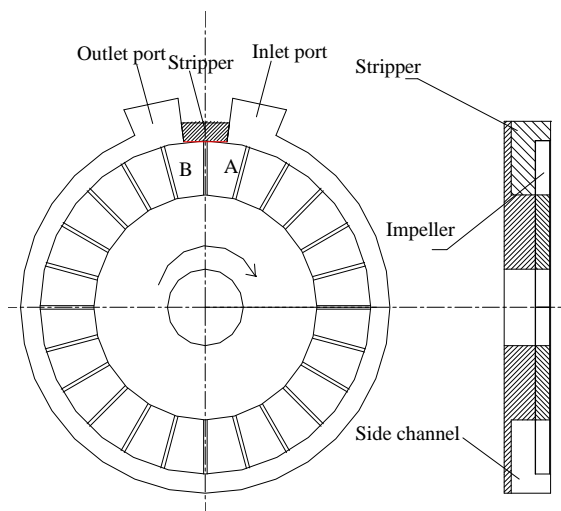


Fig. 1 Regenerative pump

Because of its practical applications, the regenerative pump has been the subject of theoretical and experimental studies. Wilson et al. [1] presented the first simplified model to permit the development of a theoretical analysis of the three dimensional fluid motion inside a regenerative pump. Meakhail and Park [2] and [3] presented a steady numerical simulation for the pump and improved the theory made by Wilson et al. [1]. The improved theory suggested the spiral flow path shown in Fig. 2. The analytical and experimental studies of Badami [4], Song et al. [5] and Meakhail et al. [6] have been carried out to describe some features of pump performance. It is worthwhile to mention Kunz and Lakshminarayana's work [7] on hydraulic torque converter, as the torque converter is similar to the regenerative pump except for the difference that the torque converter does not have any stripper.

So far, little attention is given to unsteady flow characteristics of the regenerative pump. As the circulatory flow is driven by a periodic passage of the impeller blades into and out of inlet/outlet ports, the flow is expected to be highly unsteady. The aim of this work is to investigate the unsteady flow characteristics through numerical simulation of the fluid dynamics of the regenerative pump using the CFX-Tascflow code [8].

2. NUMERICAL CALCULATION

For the simulation, a complete machine had to be taken into account because of the absence of any symmetry or periodicity. A high quality mesh of a single block H-grid through the blades was produced using CFX-Turbogrid software [9] and the CFX-Tascgrid was used for side-channel grid generation as well as inlet and outlet ports. The channel and the inlet and outlet ports are divided into four blocks (inlet, outlet, top

channel, side channel), and then they are appended together with the impeller to cover the complete machine. This type of blocked-grid arrangement yielded better minimum skew angle, which should not be less than 20 degree, and better maximum aspect ratio, which should not be more than 100. The blades were defined by blocking off grid elements. Figure 3 illustrates the grid of the whole machine. The total number of grid nodes was about 1,000,000 for the total pump configuration. This number of grid nodes was the maximum allowable number for the PC used in this work.

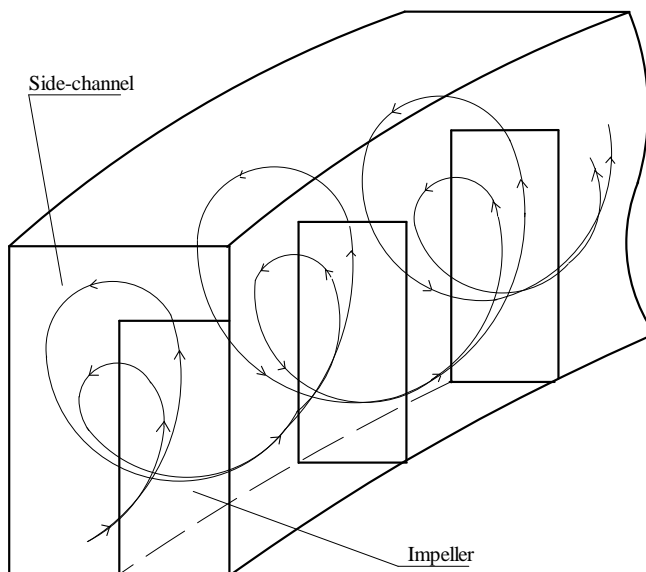


Fig. 2 Helical flow path as suggested by Meakhal and Park [3].

The rotation of a mesh region is required for unsteady calculations for the machine. One of the possible methodologies is called the “sliding mesh” or “sliding interface”. The cells at the interface between the stationary and rotating parts are disconnected and reconnected at every time step. A special form of this feature (an arbitrary sliding mesh), which does not require the same mesh structure on both sides of the interface was used for the present calculations.

Owing to poor initial guess for such complex flow, the unsteady calculations could not be performed directly. The interface surfaces between moving parts and stationary parts should be modeled. Figure 4 shows the location of the interface between the rotating impeller and the fixed side channel. CFX-Tascflow code has two models; one is called “Frozen Rotor Model” and is used to get the steady state solution for different components of the machine. That is, the interface condition is in steady state and the geometry remains fixed across the interface. The other model is called “Rotor-Stator Model” and is used for unsteady simulation. The components on each side of unsteady sliding interface are always in relative motion with respect to each other. Pitch change is automatically dealt with at the sliding interface. The profiles in the pitch-wise direction are stretched or compressed to the extent that there is pitch change across the interface. The results of the steady calculations are used as initial guess for unsteady simulation.

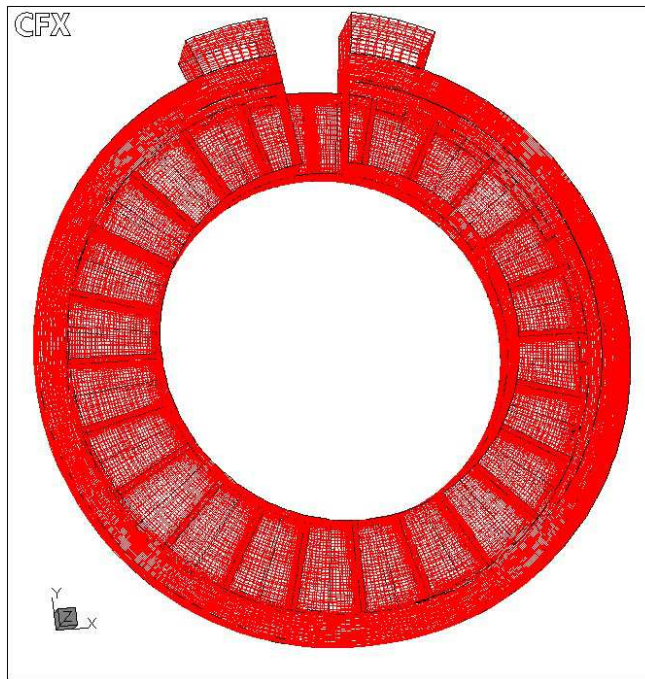


Fig. 3 The grid system

2.1 Boundary conditions

There are several different types of boundary condition that can be applied at the inlet boundary. For example, the velocity (or mass flow rate) or the total pressure can be specified. The true unsteady boundary conditions are not usually available.

For the present calculation, the total pressure was specified over the entire outlet face of the outlet port ($P_{out}=162200$ Pa), Abdel-Messih, and Mikhail [10]. At the inlet, the measured mass flow rate was applied as an average value over the inlet area of the inlet port ($m_{in}=5.6$ Kg/s). The impeller was rotating with a rotational speed of 1000 rev/min. A sliding interface model was set between the impeller and the side channel as shown in Fig. 4. Additionally, in the K- ϵ turbulence model in Tascflow requires an inlet value for the turbulence intensity (Tu) and the eddy length (L), which was given by the cubic root of the volume of the calculation domain. The computations for the present work was run in fully turbulent mode with Tu=5 percent and L=0.2 m and the Reynolds number (based on the length scale) was 4.5E4.

2.2 Solution procedures

The frozen rotor simulation is obtained first using a larger time step ($\Delta t=1$ E-3 sec). The subsequent simulation is done using smaller time step ($\Delta t=0.000125$ sec. which matches a 0.75 degree of impeller rotation) by using the frozen rotor simulation results as initial guess. The number of iterations per time step is 15 for unsteady simulation. The calculation results discussed here were run on Pentium-4 with 2 Gbytes of

memory. Typical CPU times were around 2 days for 200 iterations for frozen rotor simulation and 13 days for 500 time steps for transient simulation necessary for a run to converge down to maximum residuals (maximum dimensionless residual for each equation) of less than E-04.

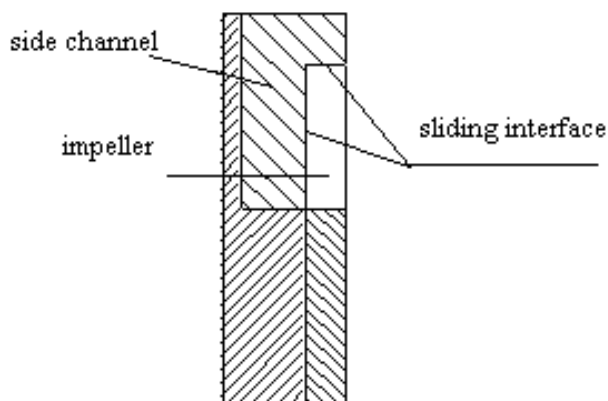


Fig. 4 The sliding interface

3. RESULTS AND DISCUSSION

Unsteady calculations are important in understanding the interaction between the rotating blades and the fixed stripper at the inlet and the outlet. These interactions are mainly responsible for the poor efficiency and unfavorable noise emission.

Two monitoring points are considered to show the pressure fluctuations near the stripper: one is point (A) near the inlet and the other is point (B) near the exit, at the mean radius of the impeller as shown in Fig. 1. As one can see from Fig. 5, the calculations are periodic after about 300 time steps. Also the present calculations show that the numerical estimated periodic pressure fluctuations are characteristics for regenerative machines. These pressure fluctuations are considered as high source of noise at Blade Passing Frequency (BPF), which can be calculated as

$$\text{BPF} = \text{No. of blades} \times \text{RPM} / 60$$

Pressure fluctuations also caused by the interaction of the circulating flow with blade edges and the stripper edges at BPF.

At point A, a low-pressure region appears periodically when two successive blades block the stripper region. At the same time, at point B, a pressure peak appears because this is the moment of maximum deceleration of the circulating flow. When the blade further rotates, the high-pressure region mixes with the low-pressure region and the final pressure at the inlet increases again. This phenomenon can cause cavitation at the inlet and vibration at the outlet. These pressure fluctuations explain also why the performance of these machines is low. Also from Fig. 5, it is clear that the amplitude of pressure fluctuation near the inlet is much higher than the outlet.

The vapor pressure at operating temperature (25° C) is about 3000 Pa, as shown in Fig. 5, from the present calculations, the pressure even decreases than the vapor pressure and that causes cavitation with its harmful effect.

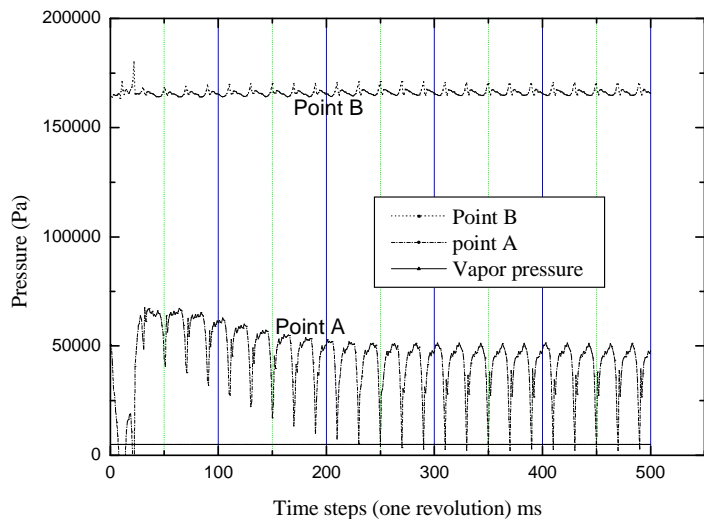


Fig. 5 Pressure at the monitoring points A and B.

A huge amount of data has been obtained during one revolution of the unsteady simulation process; hence, all of the impeller positions could not be presented here. However, as long as the flow is periodic the flow behavior is analyzed numerically through one pitch only. Figure 6 shows the details of pressure characteristics versus number of iteration during one pitch of impeller circulation. Five positions (1, 2, 3, 4 and 5) of the impeller relative to the stripper edge are selected to present details of the pressure at the middle cross section of the impeller near the inlet.

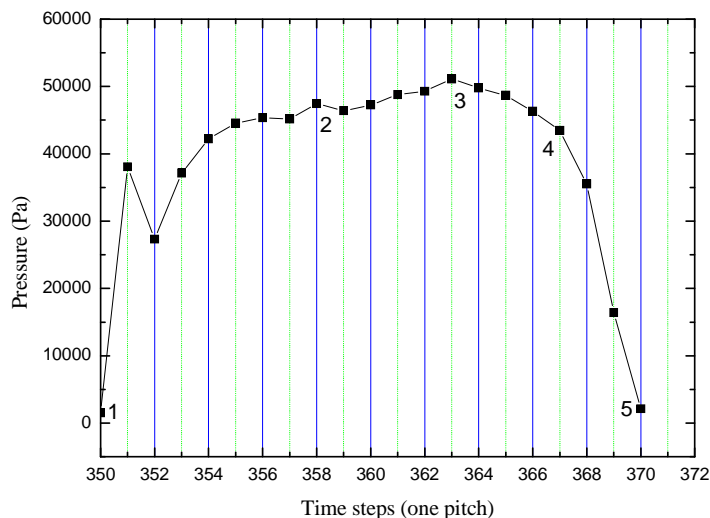


Fig. 6 Pressure characteristics for one pitch near the inlet

Figure 7 shows the absolute circumferential velocity vectors at different relative positions between the impeller and stripper at the middle section of the impeller around

the inlet port and outlet ports. From Fig. 7-a (at position 1) the two successive blades completely block the stripper region between the high-pressure region at the outlet and the low-pressure region at the inlet. At this moment, the flow in the stripper region has a higher value of radial flow velocity component unlike the other parts of the impeller, which have a nearly zero radial velocity component and high value of circumferential component in the direction of rotation. When the impeller rotates, (position 2), the flow in the high-pressure region escapes to the low-pressure region and hence the pressure at point 'A' increases. Further impeller rotation, (position3) the area increases and mass flow from the high-pressure region to the lower one increases where the pressure at point 'A' reaches its maximum value. When reaching position 4, the effect of high-pressure flow is decreased due to mixing with the low-pressure flow and hence the pressure decreases again. At position 5, again the two successive blades block the flow and the pressure at point 'A' becomes minimum.

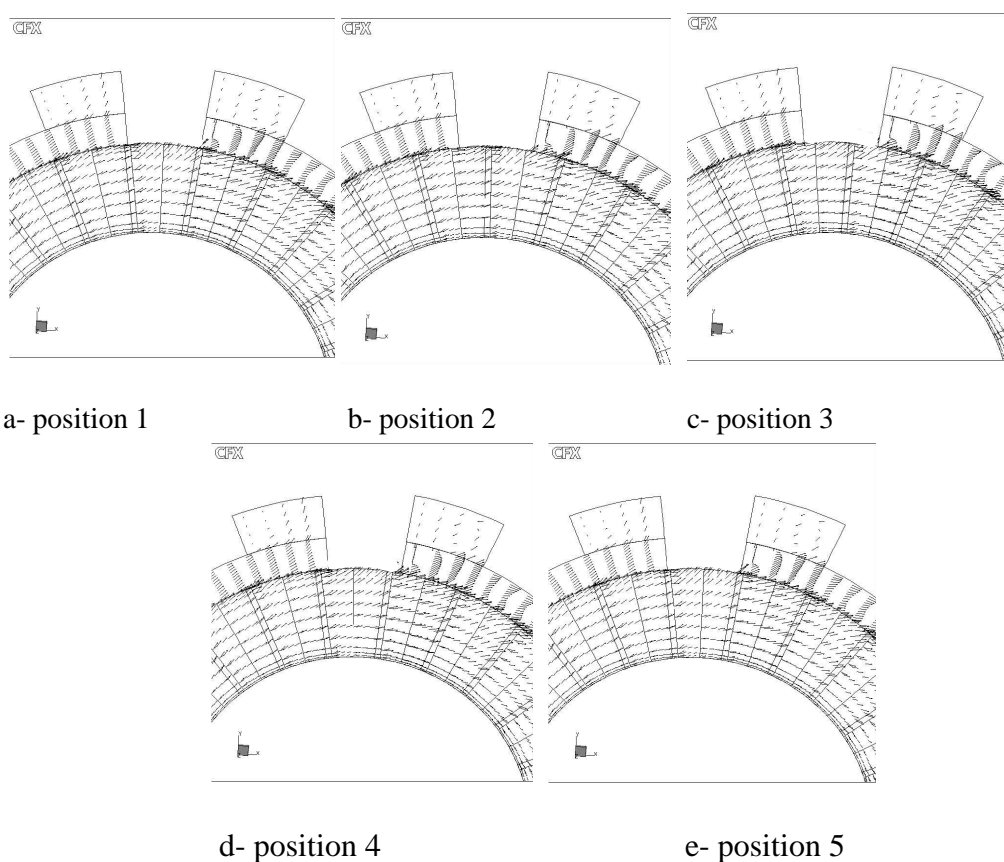


Fig. 7 Circumferential velocity vectors at the middle section of the impeller.

To show the pressure field, Fig. 8 shows the pressure contours at the same section where the two successive blades completely block the flow at this moment and a lower pressure region can be found near the inlet port and a higher pressure region near the exit at the same instant. When the impeller rotates the high-pressure fluid

mixes with the low pressure and hence making a highly unsteady flow around the stripper at the inlet.

To the best knowledge of the authors, no similar published work available for the unsteady flow in regenerative pumps. Weise and Beilke [11] carried out experimental and numerical investigations in a regenerative fan. They made their investigations only at a point near the exit, corresponding to the point B of Fig. 1.

Figure 9 shows the calculated pressure traces at a point near the exit and Fig. 10 shows the experimental ensemble averaged pressure traces at the same point (Weise and Beilke). From the two figures, there are some differences in the pressure traces but they give a good overview of the pressure fluctuation near the exit of the regenerative fan.

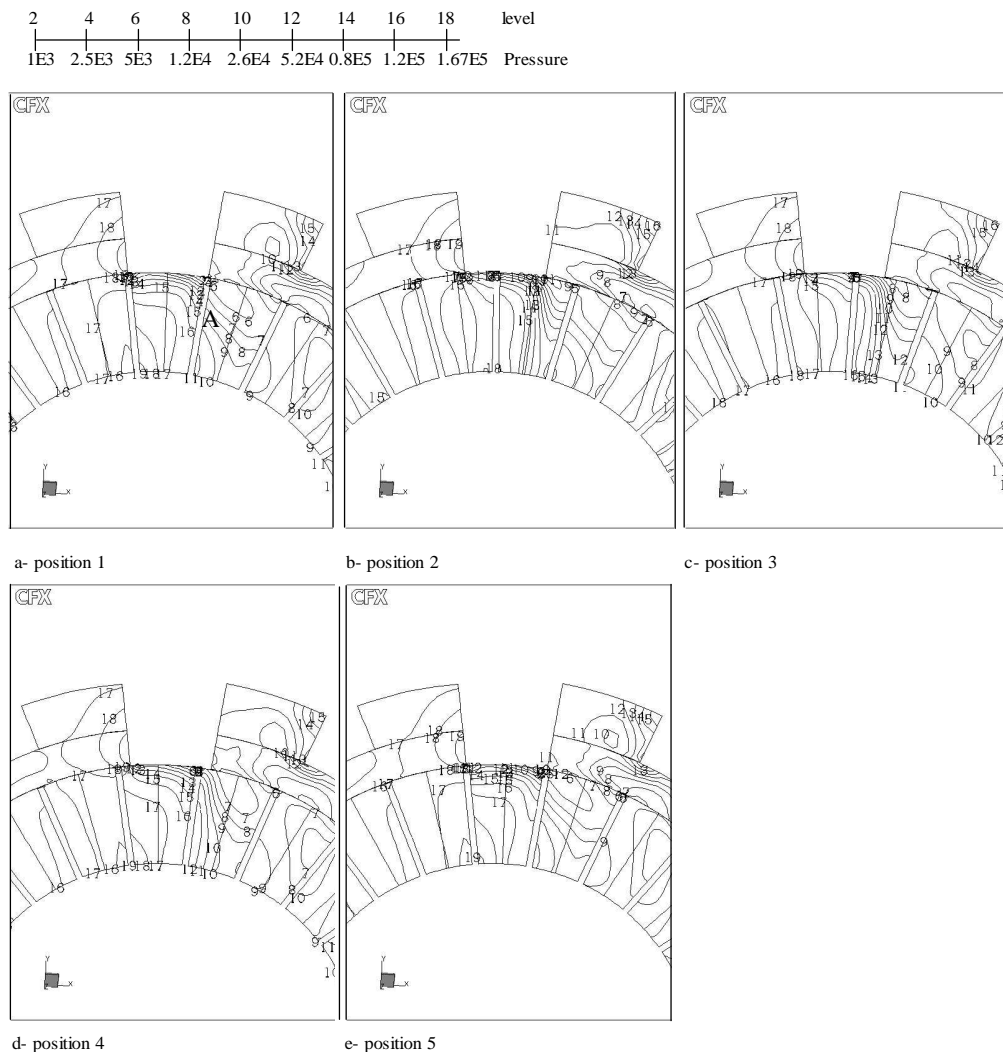


Fig. 8 Pressure contours at the middle section of the impeller.

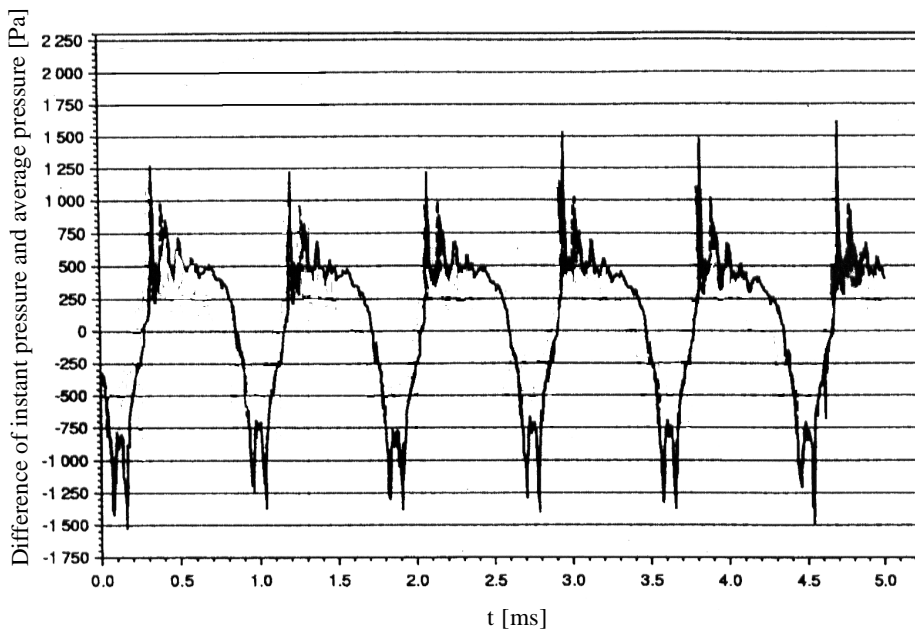


Fig 9 Calculated pressure traces at a point near the exit (Weise and Beilke-[11])

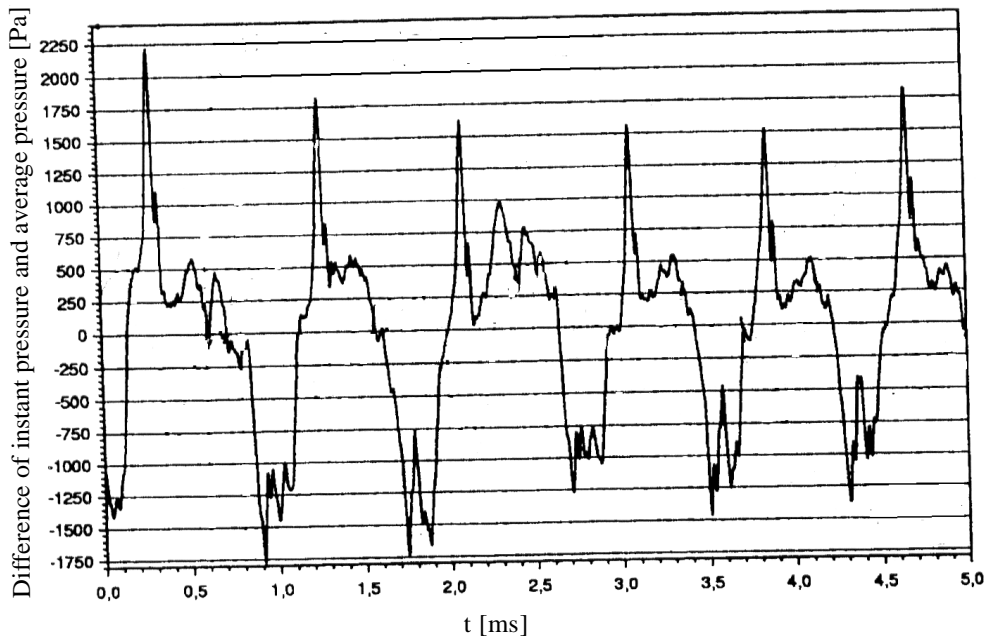


Fig. 10 Ensemble averaged pressure traces near the exit (Weise and Beilke-[11])

Comparing the present calculation for the pump with that obtained by (Weise and Beike) for a fan, a similar characteristic pressure history is shown in Fig. 11 near the point B.

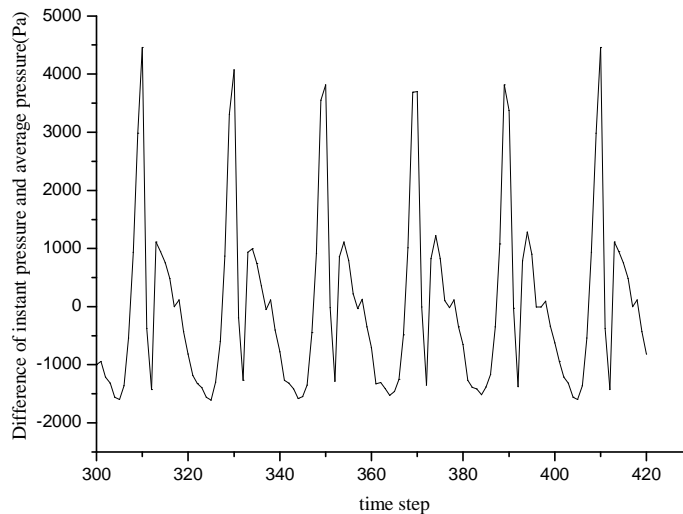


Fig. 11 Pressure traces for the pump at point B

5. CONCLUSIONS

The present calculations clearly demonstrate that the CFX-Tascflow code is well suited to compute the unsteady flow inside the regenerative pump. Unsteady results help to understand some of the mechanism of the noise production. Extremely pressure fluctuations have been found near the stripper near both inlet and outlet ports. The amplitude of pressure fluctuation near the inlet is much higher than the outlet that makes the machine very easy to undergo cavitations problems. The present results are similar to that obtained experimentally for a regenerative fan at a point near the exit.

REFERENCES

- [1] Wilson, W. A., Santalo, M. A. and Oelrich, J. A. "A theory of the fluid-dynamic mechanism of regenerative pumps". *Trans. ASME*, Vol. **77**, pp. 1303–1316, 1955.
- [2] Meakhail T., Seung O Park "A study of circulating flow in regenerative pump" *Proceedings of the KSAS 1st International session* November 14-15, 2003, Gyeongju, Korea, pp. 19-26.
- [3] Meakhail T., Seung O Park "An improved theory for regenerative pump performance." *Proc. Instn.Mech.Engrs. J Power and Energy*, Vol. 219 No. A3: pp. 213-222, 2005.
- [4] Badami, M. "Theoretical and experimental analysis of traditional and new periphery pumps" *SAE Technical Paper Series*, No.971074, pp. 45–55, 1997.
- [5] Song J. W., Engeda A., and Chung M.K., "A modified theory for the flow mechanism in a regenerative flow pump". *Proc. Instn.Mech.Engrs. J. Power and Energy*, Vol. 217 Part A: pp. 311-321, 2003.

- [6] Meakhail T., El-sallak M., Serag-Eldin M.A. and Mikhail S. “ Effect of guide blades fixed in the side channel on performance of peripheral pumps.” M.Sc. Thesis, Faculty of Engineering, Cairo University, 1996.
- [7] Kunz, R. and Lakshminarayana, B. “ Navier-Stokes analysis of the pump flow field of an automotive torque converter’ Trans. ASME J. of Fluids Engineering, Vol. 118, No. 12, pp. 665–676, 1996.
- [8] ASC, “CFX-TASCflow documentation Version 2.9.0” Advanced Scientific Computing, Ltd, Waterloo, Ontario, Canada 1999.
- [9] ASC, “CFX-Turbogrid documentation Version 1.4” Advanced Scientific Computing, Ltd, Waterloo, Ontario, Canada 1999.
- [10] Abdel-Messih R., and Mikhail S., “Investigation of some design elements on the performance of regenerative pumps.” M.Sc. Thesis, Faculty of Engineering, Cairo University, 1975.
- [11] Weise, V. and Beilke, J. “3D Flow in a Peripheral Fan” International Journal of Computer Applications in Technology, Vol. 11, Nos 3/4/5, pp.203-210, 1998.

دراسة نظرية لخصائص السريان غير المستقر داخل المضخات المحيطة

المضخات المحيطة (ذات اعادة التوليد) هي نوع من الآلات التوربينية التي تحقق ضغط عالي عند معدل سريان قليل بالمقارنة بمثلتها من المضخات التي لها نفس السرعة المحيطة. تكاليف هذا النوع من المضخات بسيط وغير مكلف. على الجانب الآخر كفاءتها لا تتعدى 45% بالإضافة الى انه يصحبها ضوضاء اثناء تشغيلها. لتحسين اداء هذه المضخة وخفض الضوضاء يجب ان يكون هناك معلومات تفصيلية متوفرة عن السريان غير المستقر داخل هذه المضخة وقد تم ذلك باستخدام برنامج CFX-Tascflow. النتائج الرقمية التي تم الحصول عليها اوضحت ان هناك سريان غير مستقر بصورة عالية بجوار الجزء الذي يفصل الدخول والخروج. الضغط المنخفض المحسوب عند نقطة قرب الدخول بينت ان المضخة عرضة لحدوث التكهف بصورة شديدة. بينت ايضا النتائج ان نطاق تغيير الضغط عن الدخول اكبر منه عند الدخول.