

PERFORMANCE PREDICTION OF CENTRIFUGAL PUMPS WITH CFD-TOOLS

ERIK DICK, JAN VIERENDEELS, SVEN SERBRUYNS
AND JOHN VANDE VOORDE

*Department of Flow, Heat and Combustion Mechanics,
Ghent University,
Sint-Pietersnieuwstraat 41, 9000 Gent, Belgium
erik.dick@rug.ac.be*

(Received 19 July 2001)

Abstract: The CFD-code FLUENT, version 5.4, has been used for the flow analysis of two test pumps of end-suction volute type: one of low specific speed and one of medium specific speed. For both, head as function of flow rate for constant rotational speed is known from experiments. FLUENT provides three calculation methods for analysis of turbomachinery flows: the Multiple Reference Frame method (MRF), the Mixing Plane method (MP) and the Sliding Mesh method (SM). In all three methods, the flow in the rotor is calculated in a rotating reference frame, while the flow in the stator is calculated in an absolute reference frame. In the MRF and MP methods steady flow equations are solved, while in the SM method, unsteady flow equations are solved. The SM method does not introduce physical approximations. The steady methods approximate the unsteady interaction between rotor and stator. The cost of the unsteady method is, however, typically 30 to 50 times higher than the cost of the steady methods. It is found that the MRF and MP methods lead to completely erroneous flow field predictions for flows far away from the best efficiency point. This makes the steady methods useless for general performance prediction.

Keywords: centrifugal pumps, CFD-analysis, performance prediction

1. Introduction

The CFD-code FLUENT provides three calculation methods for the analysis of turbomachinery flows: the Multiple Reference Frame method, the Mixing Plane method and the Sliding Mesh method. The first two methods are basically steady flow methods. In the Multiple Reference Frame method, the rotor is kept at a fixed position. The governing equations are solved for the rotor in a rotating reference frame, so including Coriolis and centrifugal forces. The governing equations for the stator are solved in an absolute reference frame. As coupling between both parts, continuity of velocity components and pressure is imposed. In principle, unsteady equations can be solved but as the main source of unsteadiness, *i.e.* the movement of the impeller is neglected, the unsteady solution has not much meaning. The flow field is dependent on the relative position of the impeller and the volute. For a complete analysis, the flow for different relative positions has to be calculated. The technique is also known under the name Frozen Rotor method. In the Mixing Plane method,

as in the previous method, relative and absolute reference frames are used, but exchanging circumferentially averaged flow quantities does the coupling between both. In principle, the result is independent of the relative position of impeller and volute. The Sliding Mesh method is a truly unsteady method. Also a rotating and an absolute reference frame are used but the effect of displacement due to rotation is taken into account. At each time step, the rotor is set at its correct position and fluxes are interpolated on the common sliding surface between both parts. Apart from some diffusion due to the interpolation, there is no major approximation in the representation of the unsteadiness. So, for the calculation of the hydraulic performance, the Sliding Mesh method can be seen as a reference method.

The computational effort for the sliding mesh technique is typically a factor 50 larger than the effort for a mixing plane technique and a factor 30 larger than the effort for a frozen rotor technique. So for practical applications, it is necessary to know how far the steady techniques are realistic.

The use of CFD-tools to analyse the flow field in turbomachines and to predict performance parameters has gained enormous popularity in recent years. This process has been stimulated by the availability of commercial packages allowing turbomachinery analysis. Although it is very well known that the flow field in a turbomachine is inherently unsteady, most calculations nowadays are done with steady methods. All this has to do with the cost of unsteady calculations. The simplest steady calculation method is, as already mentioned, the mixing plane method. It has the particular advantage that the calculation of the rotor flow and the stator flow can be done almost independent of each other. For applications to end-suction volute centrifugal pumps, only one blade channel of the impeller has to be calculated. The circumferentially averaged flow quantities at the outlet of the impeller form then the inlet variables for the calculation of the volute. The technique is widely used in axial flow machines. Examples of calculations of this type in low specific speed and medium specific speed mixed flow pumps with axial diffusers are given by Takamara and Goto [1] and Goto [2]. Sedlar and Mensik [3] give examples of calculations of radial flow pump stages with semi-axial and radial diffusers. In the cited references, it is reported that the mixing plane method leads to accurate performance predictions. This is not completely surprising since the method is known to perform well for axial flow machines. In the cited examples there is no volute and as a consequence, the flow character is rather similar to the flow character in axial flow machines. Sedlar and Mensik also performed calculations with the frozen rotor technique. They also report good results with this technique. In volute pumps, the circumferential variation of pressure imposed by the volute on the impeller for flows different from design flow can be quite large, especially for low flow rates. It is also well known that the inhomogeneous pressure distribution imposed on the impeller exit is felt inside the impeller, as shown by Sideris and Van den Braembussche [4], Miner *et al.* [5] and Liu *et al.* [6]. In principle, the frozen rotor technique allows one to take into account the influence of the pressure variation on the impeller flow. Therefore it is tempting to use this technique for the flow analysis of volute centrifugal pumps, although the unsteadiness caused by the rotor/stator interaction is neglected. Good results have been obtained with the frozen rotor technique by Cugal and Baché [7] for a volute centrifugal pump and by Gugau *et al.* [8] for a volute low-speed centrifugal compressor. The results obtained in the first reference might be not completely representative since the pump has a double tongue volute. This causes that the circumferential pressure

distortion for low flow rates is less pronounced than for a single tongue volute. The authors of the second reference observe non-real flow patterns for low flow rates. As a consequence, they recommend the use of the mixing plane method. They found that the quality of the prediction of the pressure ratio as function of the mass flow is equally good for both steady methods. The discrepancy between calculated and measured values for the pressure ratio was only about 1–2%. This is a surprisingly good result.

In the present study, flow analysis is done on two test pumps of end-suction volute type, well known from the literature. The first pump is a plexiglas research pump from the University of Virginia [5]. It has low specific speed and a two-dimensional form. The second pump is a medium specific speed pump from the British Hydraulic Research Association [9]. The pump has doubly curved vanes. For both pumps, the performance characteristics are known from experiments. We analyse here how far the CFD-methods typically available in a commercial package can be used for the prediction of the head as function of flow rate for constant rotational speed. We take the frozen rotor technique as the central technique of the investigation. In particular, we want to determine how far this technique can take into account the influence of the circumferential pressure variation, caused by the volute for flows different from best efficiency flow, on the impeller flow. The sliding mesh method is used as a reference technique. It is only used for three flow rates on pump 1. These calculations are meant to demonstrate that this technique is able to reproduce the performance parameters correctly. The mixing plane method is used for one flow much different from best efficiency flow in order to verify the quality of the predictions of this method.

2. Methodology

A hybrid mesh is created using FLUENT'S PREPROCESSOR GAMBIT. The geometry is generated with the aim of subdivision into multiple blocks. First, the impeller is generated. One impeller channel is meshed and is then rotationally copied the necessary number of times. For the first pump, the impeller is completely two-dimensional. The impeller mesh is made with hexahedra and wedge cells. For the second pump, the impeller channel is much more complex. The mesh is made in a completely unstructured way, mainly using tetrahedra, but other cell forms like pyramids, hexahedra and wedges also occur. The inlet channel is meshed for both pumps with prisms. The volute in both pumps is too complex for a structured grid. The meshing is done in an unstructured way, mainly using tetrahedra. For the first pump, the cross section is rectangular. The precise cross section shape of the volute for the second pump cannot be obtained from [9]. It was approximated by a trapezoid. For the first pump, the outlet channel has a constant rectangular section. This channel is meshed in a structured way using hexahedra. In the second pump, a diffuser that changes section from trapezoid to circle follows the volute. This diffuser is meshed in an unstructured way, mainly using tetrahedra. A constant section circular tube meshed with prisms follows the diffuser. Figure 1 shows the complete mesh for pump 1 and the mesh for pump 2 up to the outlet of the volute. In pump 1, the interface between the impeller and the volute is split into two surfaces for the sliding mesh calculation. Since these surfaces are represented in a piecewise manner, a small gap is necessary between them in order to allow the rotation of the impeller. Data were interpolated between these surfaces, neglecting the gap. For pump 2, no sliding mesh calculations have been done. A particular aspect of the mesh for this pump is that the mesh is split into two parts at the interface between the volute outlet

and diffuser inlet. The interface plane is treated as a mixing plane, as explained later. For the frozen rotor calculations, two positions of the impeller have been used. In position 1, the mid of an impeller channel corresponds to the tongue of the volute. In position 2, an impeller blade corresponds to the tongue. The impeller of pump 1 is of closed type. In the calculations, the axial gap between the impeller and the pump house has been set to zero. So, leakage flow is neglected. The impeller of pump 2 is of open type and there is a small gap in axial direction between the impeller blades and the shroud. The gap has been set to a small value so that exactly one cell is obtained in the gap. The gap at the backside of the impeller has been set to zero. This has as a consequence that in the calculations the leakage flow is not correctly calculated. The total number of cells for pump 1 is about 300 000. For pump 2, it is about 550 000.

Table 1 summarises the main geometrical parameters of the two pumps. Full details on the geometry can be found in [5, 9]. Both pumps have an annular space following the impeller with a larger width than the outlet width of the impeller. The annular space defines the volute tongue gap.

Table 1. Geometric properties of the test pumps

| | Pump 1 | Pump 2 |
|---------------------------------|----------------------|---|
| Diameter inlet tube | 38.3 mm | 138 mm |
| Inlet diameter impeller | 50.8 mm | 67.8 mm (hub), 102.8 (mean), 139.9 mm (shroud) |
| Inlet angle impeller | 16° | 30° (hub), 19.5° (mean), 14° (hub) |
| Inlet width impeller | 24.6 mm | 49.2 mm |
| Outlet diameter impeller | 203.2 mm | 203.2 mm |
| Outlet angle impeller | 16° | 23° |
| Outlet width impeller | 24.6 mm | 31.75 mm |
| Number of blades | 4 | 5 |
| Thickness of the blades | 3 mm | 5.5 mm |
| Outlet diameter annular space | 215.9 mm | 223.6 mm |
| Width of the annular space | 25.8 mm | 44.5 mm |
| Width volute | 25.8 mm (constant) | 44.5 mm (base width) |
| Outlet section volute | 2785 mm ² | 3890 mm ² |
| Outlet section diffuser | constant section | 8171 mm ² |
| Rotational speed used in test | 620 rpm | 2910 rpm |
| Corresponding nominal flow rate | 6.3 l/s | 60 l/s |

The equations solved are the three-dimensional Reynolds-averaged Navier-Stokes equations. The fluid is incompressible. The segregated solver with implicit formulation is used. For the convective terms second order upwinding is used. For the pressure correction, the SIMPLE scheme is used.

As a turbulence model, the realizable k - ε model is used with non-equilibrium wall functions. The mass flow rate and the flow direction normal to the boundary are imposed at the inlet. The turbulence intensity is determined from the formula for a fully developed pipe flow. A uniform turbulence length scale is imposed calculated from the diameter of the inlet pipe. At the outlet, flow parameters are extrapolated from the interior.

Calculations are typically started from a no-flow initial condition, with the impeller at standstill. The rotational speed is increased gradually until the final speed is reached. The first flow rate calculated has always been the nominal flow rate. Often the calculations for

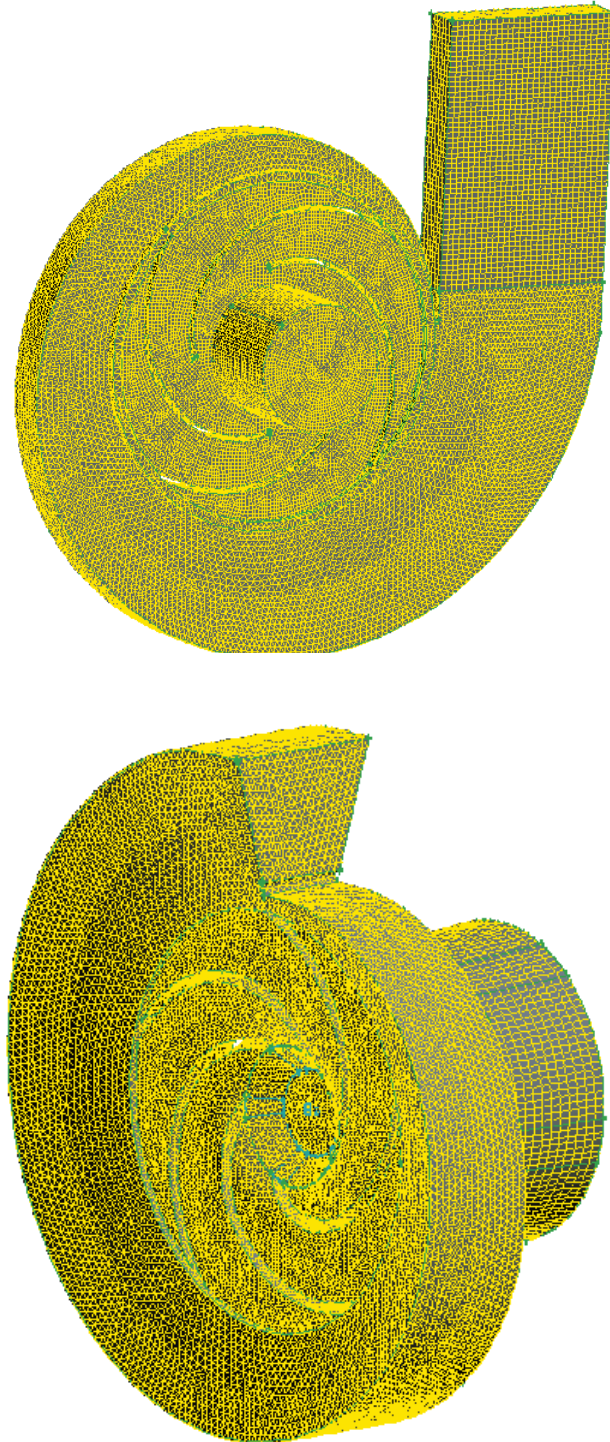


Figure 1. Geometry and mesh of the test pumps

some flow rate can be started from the result of a neighbouring flow rate, without going through the procedure of gradual increase of the rotational speed. For the very low flow rates this is not possible since the general flow pattern changes significantly when lowering the flow rate. Calculations for low flow rates have to be started from no-flow conditions.

3. Results of frozen rotor calculations

Figure 2 shows the calculated head as function of flow rate for pump 1, obtained by the frozen rotor technique for the two positions of the impeller, in comparison with the experimental data. For a flow rate 7 l/s, *i.e.* somewhat above the nominal flow rate (6.3 l/s), the predictions are quite good. The quality of the predictions deteriorates for flow rates different from the nominal flow rate. In particular for low flow rates, the calculations give a serious underprediction of the head. Figure 3 shows the corresponding result for pump 2. The same observations are made here. The calculations for pump 2 have been done in two parts. As already said, a mixing plane is introduced at the outlet of the volute. This turned out to be necessary for the calculation of the diffuser. Since in the frozen rotor technique, the rotor stands still, wakes shed from the blades of the impeller stay localised in space. For either of the two positions of the impeller, a wake causes an unphysical flow separation on the upper or lower wall of the diffuser. In reality, this does not happen since wakes are largely mixed in the volute and are unsteady. No localised separation occurs in the diffuser. To mimic this physical mixing process, a mixing plane was introduced at the end of the volute. The consequence is that the calculation of the diffuser and the outlet pipe becomes independent of the calculation of the rest of the pump.

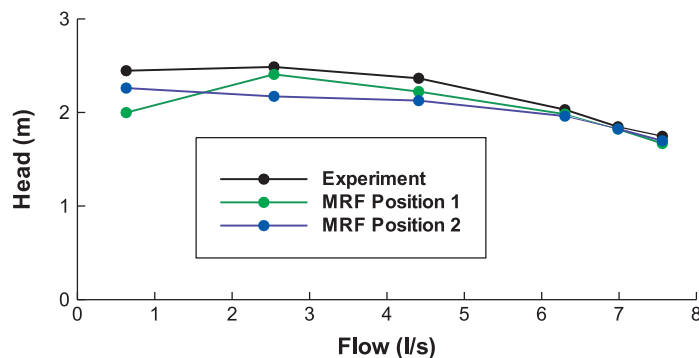


Figure 2. Pump 1: head as function of flow rate. Frozen Rotor predictions

Figure 4 shows the distribution of static pressure in an orthogonal plane at the mid of the impeller and the volute for pump 1, in position 1, for the nominal flow rate (6.3 l/s). The distribution of the pressure is very similar for all channels in the impeller. The pressure gradient due to the centrifugal force can be seen in the volute. There is basically no pressure gradient in the circumferential direction in the volute. Figure 4 shows also the distribution of the static pressure for flow rate 7.5 l/s (120% of the nominal flow rate). The distribution of the pressure in the impeller channels is still very similar for all channels. There is now a slight decrease of the pressure in the circumferential direction in the volute, in the sense of the flow.

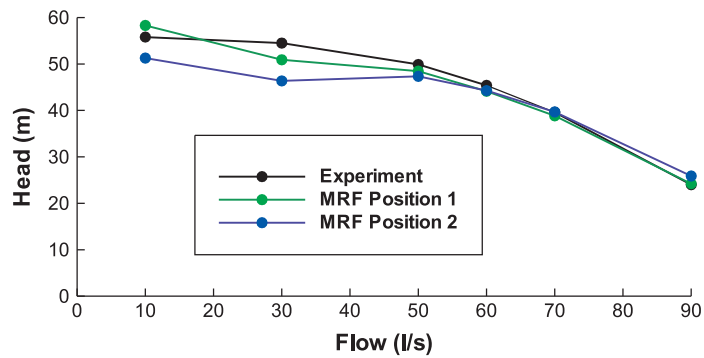


Figure 3. Pump 2: head as function of flow rate. Frozen Rotor predictions

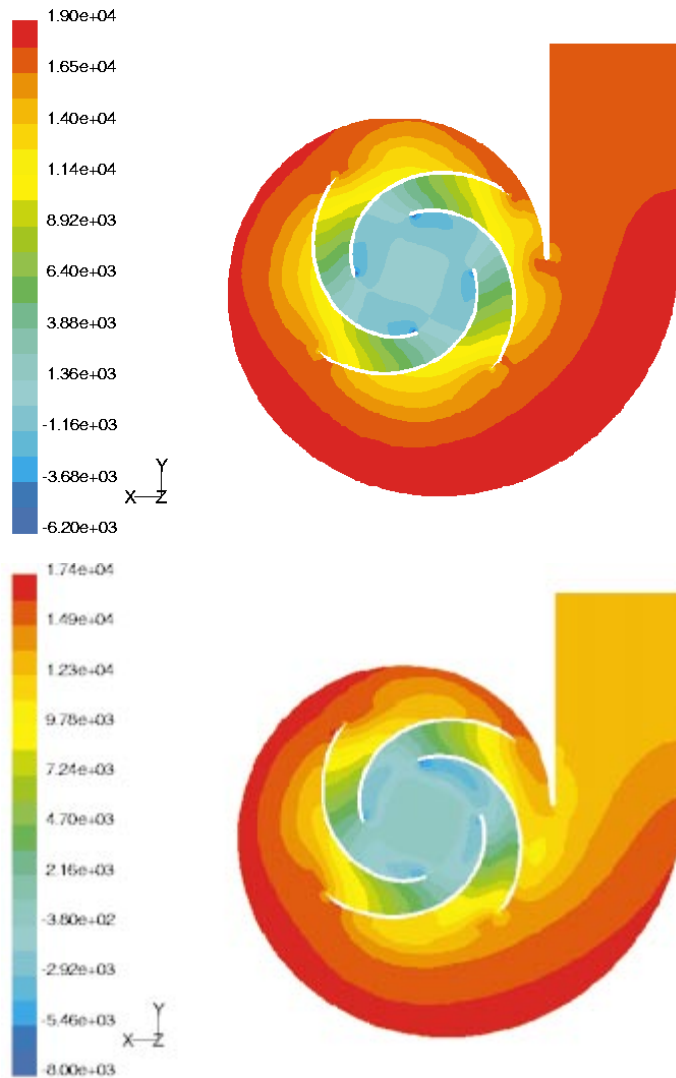


Figure 4. Distribution of static pressure in the mid plane for pump 1. Top: nominal flow rate (6.3 l/s). Bottom: high flow rate (7.5 l/s)

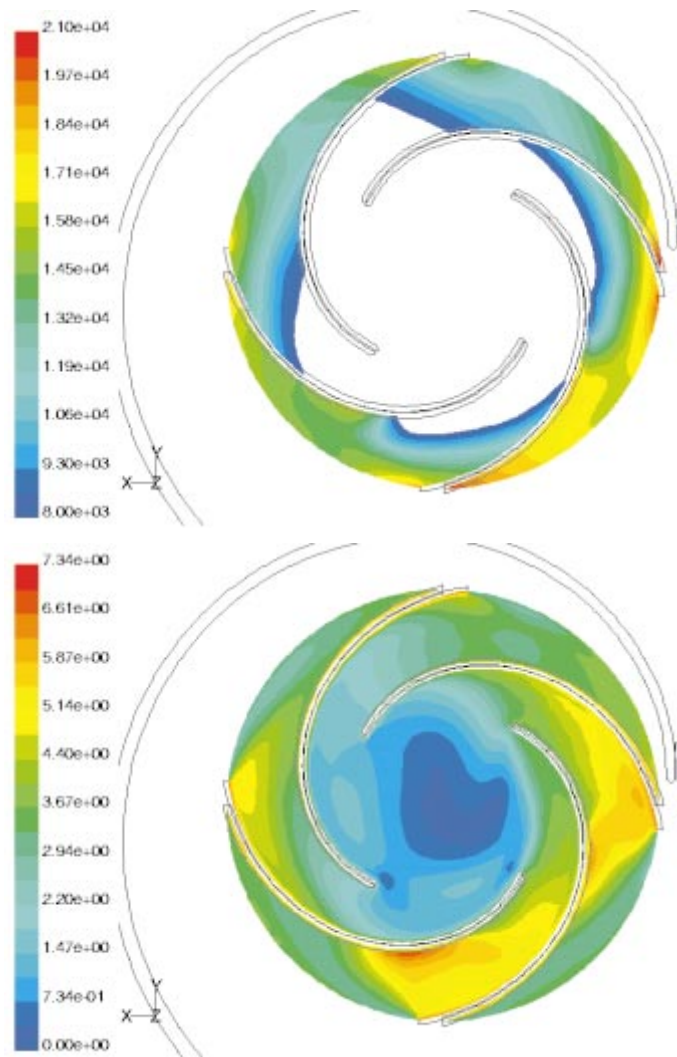


Figure 5. Distribution of static pressure (top) and absolute velocity (bottom) in the mid plane for pump 1. Low flow rate (2.54 l/s)

Figure 5 shows the distribution of the static pressure and absolute velocity for flow rate 2.54 l/s (40% of nominal flow rate) in the mid plane of the impeller for position 2. Only the pressure distribution in the impeller is shown in order to better illustrate the difference between the different impeller channels. The increase in pressure in the volute in circumferential direction in the sense of the flow is felt at the exit of the impeller channel, causing the unequal pressure distributions in the channels. At first sight, one would consider that this behavior corresponds to reality. Figure 6 shows a vector plot of the relative and absolute velocity in the mid plane of the impeller, projected into this plane, corresponding to the pressure plot of Figure 5. In the vector plot of relative velocity, large recirculation zones can be seen in the impeller channels exposed to the largest volute pressure. Flow velocities are very low in these channels. The flow vectors at the periphery of the impeller point inwards, so in the mean there is reversed flow in these channels. The impeller channels

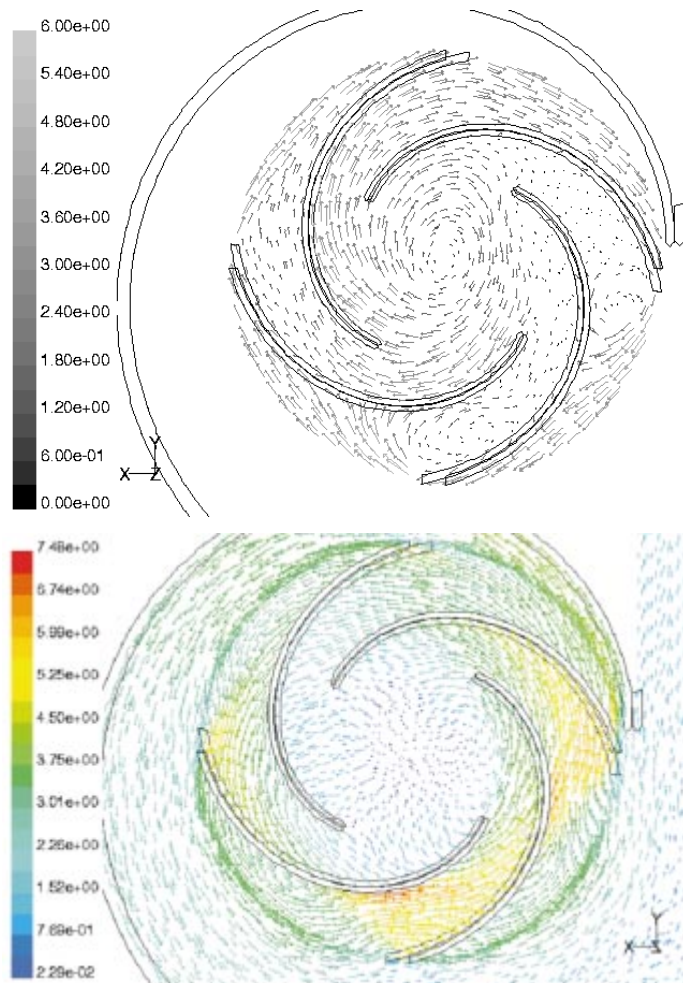


Figure 6. Vector plot of relative velocity (top) and absolute velocity (bottom) for low flow rate for pump 1

exposed to the lowest volute pressure have a flow that is well aligned with the impeller blades. The inflow at the periphery in the impeller channel exposed to the largest counter pressure can clearly be seen in a vector plot of the absolute velocity. At this stage, one can already doubt about the physical correctness of the obtained flow pattern by verifying the work exchange in the impeller. On the basis of an integral of the moment of momentum flux through a cylindrical surface around the impeller, it could be verified that the work exchanged by the impeller and the flow is lower than the energy increase in the fluid. This has as consequence that the calculated hydraulic efficiency of the pump is larger than one.

Figures 7 and 8 show the distribution of the static pressure on a mean stream surface in pump 2, projected on an orthogonal plane for nominal flow rate (60 l/s), high flow rate (90 l/s) and low flow rate (30 l/s). The same observations apply as for pump 1, *i.e.* the pressure distribution in the impeller adjusts to the pressure distribution in the volute. As such, this is as expected since precise continuity of pressure is imposed as interface

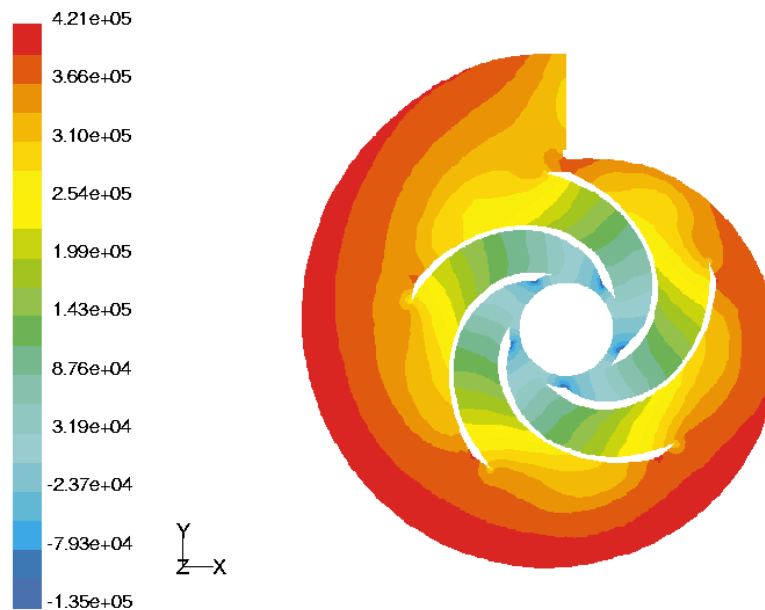


Figure 7. Distribution of static pressure in the mean stream surface of pump 2: nominal flow rate (60 l/s)

condition between the impeller and the volute. Again the observation is that for low flow rate the velocity distribution (not shown) is not likely to be physically correct.

4. Comparison between frozen rotor and sliding mesh results

For three flow rates of pump 1, sliding mesh calculations have been performed. In Figure 9, the calculated head is shown in comparison to the experimental result and to the results of the frozen rotor calculation. The sliding mesh calculations lead to some overprediction of the head. The overprediction increases with decreasing flow rate. A slight overprediction of the head is certainly a good feature. We have to remember that the leakage flow of the pump cannot be obtained by the present calculations, since the gap between the impeller and the shroud is set to zero. Taking into account a correction for leakage losses would certainly bring the calculated results for the sliding mesh technique very close to the experimental data.

Figure 10 shows the vector plot of absolute velocity in the mid plane of the pump, projected into this plane for flow rate 2.54 l/s. This figure should be compared to Figure 6. Obviously, there is no reversed flow detected by the sliding mesh calculation.

Figure 11 compares the radial velocity components for the frozen rotor and sliding mesh calculations for flow rate 2.54 l/s. From this comparison, the conclusion is that the prediction of reversed flow by the frozen rotor method is completely erroneous. The sliding mesh calculation shows that the distribution of the flow over the impeller is almost homogeneous. The observation is that the impeller flow almost does not react to the pressure variation in the volute. Figure 12 shows the comparison for pressure. The pressure distributions obtained from both calculations are not identical but the general behavior is the same. The sliding mesh calculation shows that the pressure distribution in the impeller

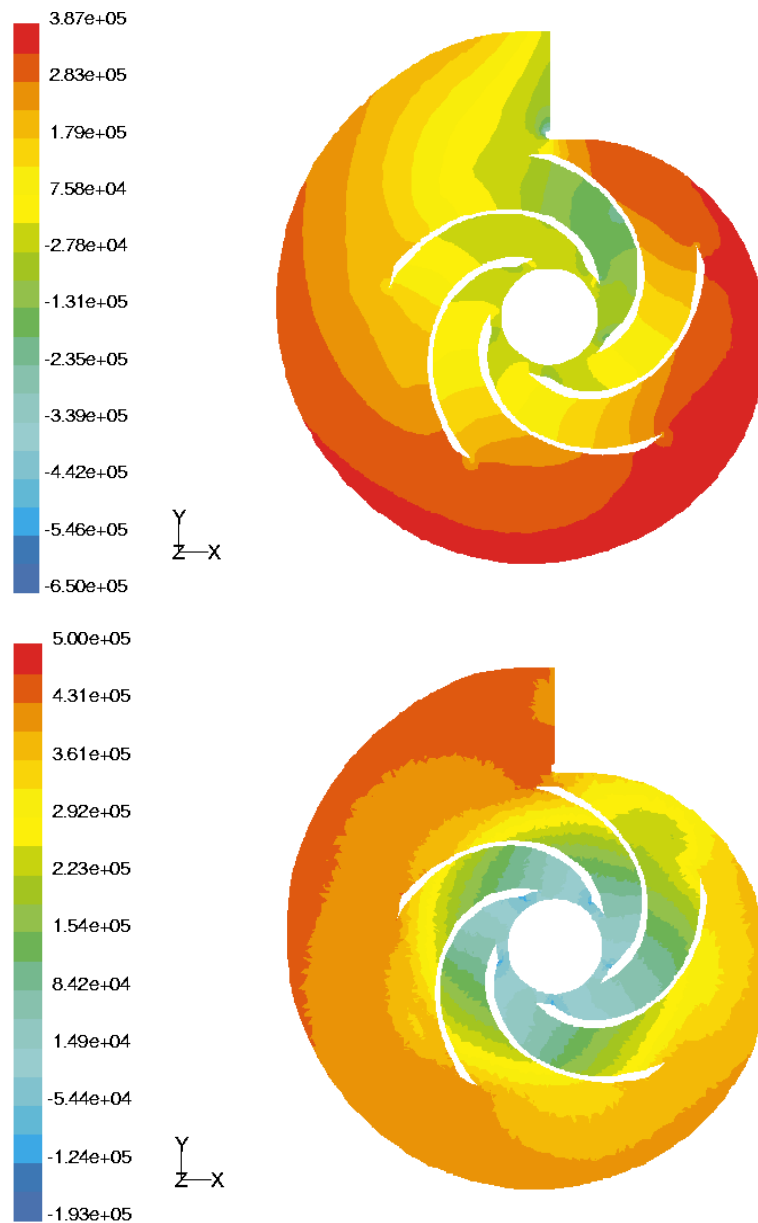


Figure 8. Distribution of static pressure in the mean stream surface of pump 2.
Top: high flow rate (90 l/s). Bottom: low flow rate (30 l/s)

adjusts to the pressure distribution in the volute. This does, however, not lead, as is shown by Figure 10, to a velocity distribution that follows the pressure distribution.

The explanation of this phenomenon is to be found in the inertia of the fluid. For a flow rate of 2.54 l/s for pump 1, it can be verified that the time needed for a fluid particle to flow through the impeller is approximately equal to the time needed for one rotation of the impeller. This means that the flow through an impeller channel is in reality exposed to a complete cycle of pressure variation in the volute and, as a consequence,

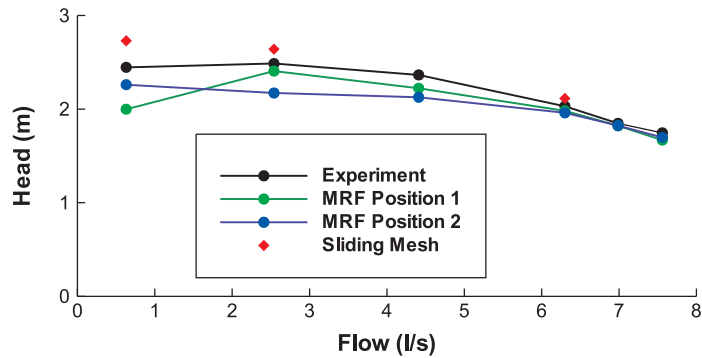


Figure 9. Head as function of flow rate for pump 1: comparison of Frozen Rotor and Sliding Mesh calculations

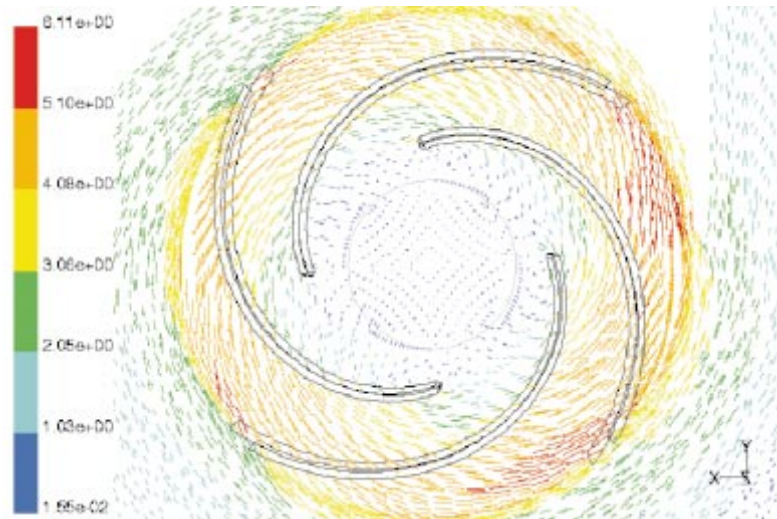


Figure 10. Vector plot of absolute velocity in the mid plane of pump 1 for flow rate 2.54 l/s, obtained by the Sliding Mesh method

reacts approximately in the same way as if it were exposed to the mean pressure in the volute. By this interpretation, it can now be understood why the frozen rotor calculation is fundamentally incorrect.

5. Mixing plane calculation

The conclusion formulated in the previous section leads to the expectation that mixing plane calculations might perform quite well as a performance prediction method. Figure 13 shows the distribution of radial velocity and static pressure obtained by the mixing plane method for flow rate 2.54 l/s for pump 1. Reversed flow is observed. So, the flow field prediction is, like for the frozen rotor calculation, not realistic. Remark that there is circumferential variation of flow properties in the impeller. The reason is that for reversed flow, the static pressure in the volute becomes the inflow condition for the impeller. Calculations have also been done with an isolated rotor and a constant counter pressure. Backflow was also obtained.

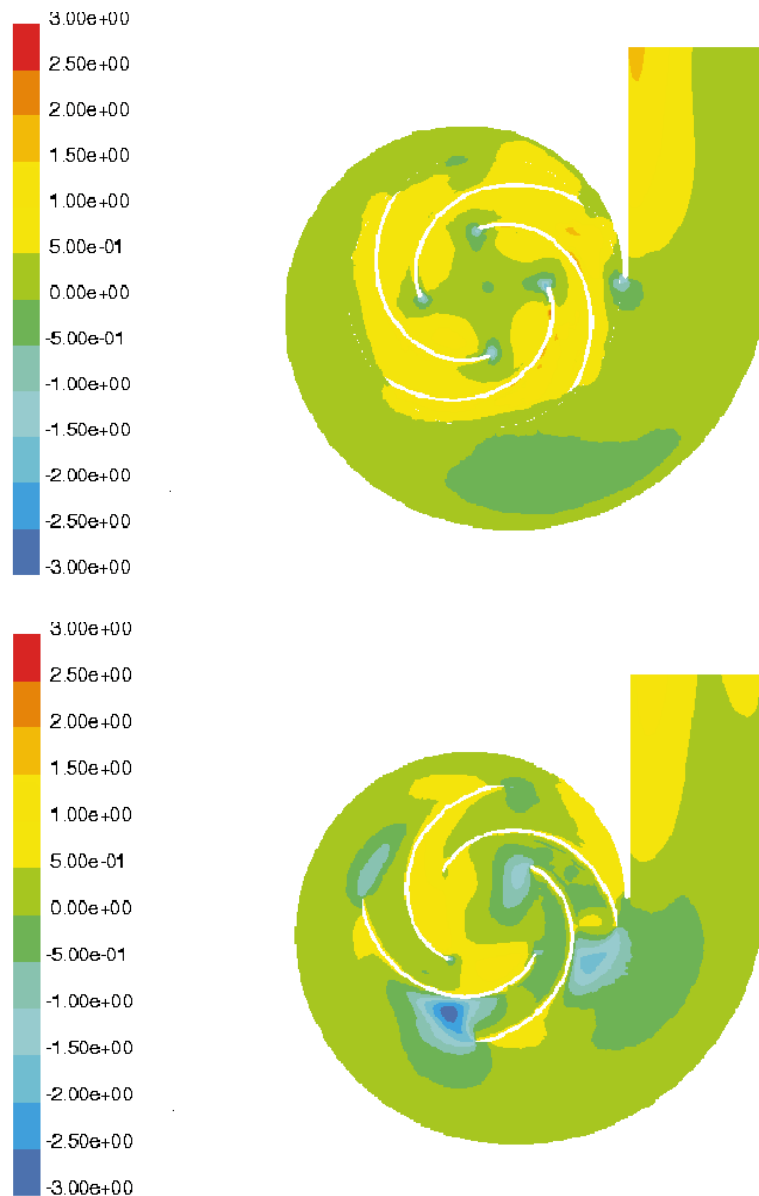


Figure 11. Distribution of radial velocity components for Sliding Mesh (top) and Frozen Rotor (bottom), pump 1, 2.54 l/s

The flow pattern obtained in Figure 13 is clearly not physical. The consequence is that the mixing plane method gives an erroneous prediction of head. The head obtained for the flow rate 2.54 l/s is about 2.03 m. This is much too low. The explanation for the erroneous behaviour of the mixing plane method is again to be found in the inertia of the fluid. Apparently, as the sliding mesh calculation shows, inertia prevents backflow. By neglecting this fundamentally unsteady effect, the mixing plane method predicts unphysical behaviour.

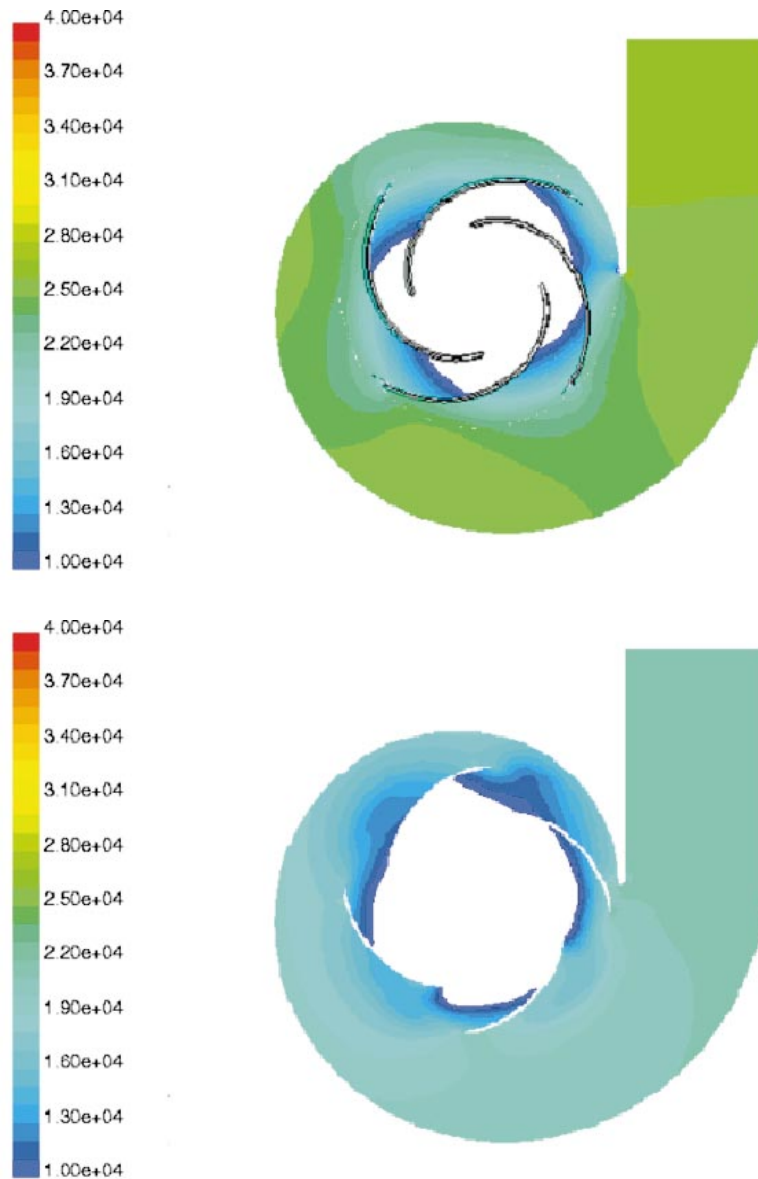


Figure 12. Distribution of static pressure for Sliding Mesh (top) and Frozen Rotor (bottom) calculations, pump 1, 2.54 l/s

6. Conclusion

Steady calculation methods like the Frozen Rotor method and the Mixing Plane method cannot be used with confidence to analyse the performance of volute centrifugal pumps. This is in contrast to applications to turbomachines with circumferentially periodic flow character. The basic reason for the erroneous behaviour of the steady methods is the inability to represent the effect of the fluid leaving the impeller. This inertia effect causes that the pressure variation imposed by the volute for flows much different from best efficiency flow is seen in a largely filtered way by the fluid in the impeller. The real flow is much more

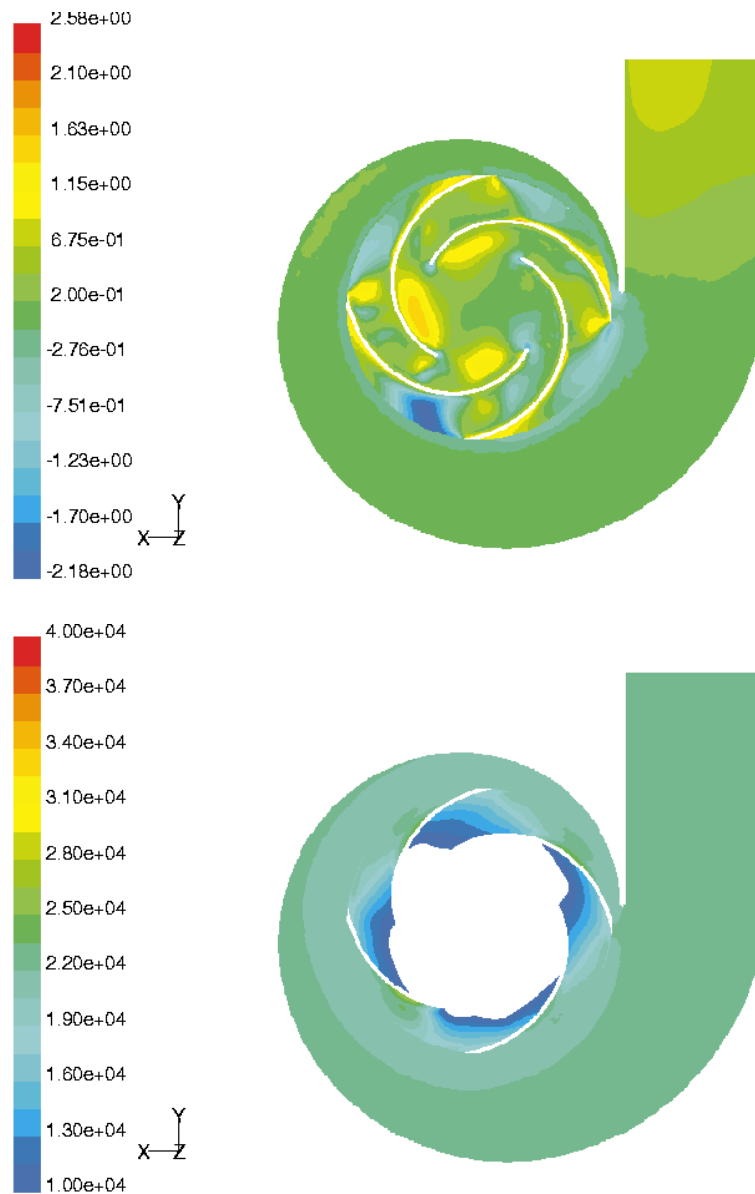


Figure 13. Distribution of radial velocity (top) and static pressure (bottom) for Mixing Plane calculations, pump 1, 2.54 l/s

homogeneous than predicted with the Frozen Rotor and the Mixing Plane methods. Only a truly unsteady method like the Sliding Mesh technique is able to correctly reproduce this flow behaviour. The predicted performance by the Sliding Mesh method can be used with confidence.

References

- [1] Takemura T and Goto A 1996 *Journal of Turbomachinery* **118** 552
- [2] Goto A 1997 *Prediction of diffuser pump performance using 3D viscous stage calculation* FEDSM97-3340 ASME Fluids Engineering Division Summer Meeting

- [3] Sedlar M and Mensik P 1999 *Investigation of rotor/stator interaction influence on flow fields in radial flow pumps* Proc. 3rd European Conference on Turbomachinery, IMechE, pp. 1017–1025
- [4] Sideris M and Van den Braembussche R 1987 *Journal of Turbomachinery* **109** 48
- [5] Miner S, Beaudoin R and Flack R 1989 *Journal of Turbomachinery* **111** 205
- [6] Liu C, Vafidis C and Whitelaw J 1994 *Journal of Fluids Engineering* **116** 303
- [7] Cugal M and Baché G 1997 *Performance prediction from shutoff to runout flows for a complete stage of a boiler feed pump using computational fluid dynamics* FEDSM97-3334 ASME Fluids Engineering Division Summer Meeting
- [8] Gugau M, Matyschok B and Stoffel B 2001 *Experimental and 3D numerical analysis of the flow field in a turbocharger compressor* Proceedings: Fourth European Turbomachinery Conference SGE, pp. 297–306
- [9] Anonymous 1982 *Design specifications. IMechE Conference Publications C184/82*; and Anderson H, Salisbury A, Dziewulski T and Burgoyne D 1982 *Discussion. IMechE Conference Publications C186/82*