

FEDSM200-11162

NUMERICAL SIMULATION OF CENTRIFUGAL PUMPS

Matthew T Stickland; Tom J. Scanlon
University of Strathclyde, Department of Mechanical
Engineering, 75 Montrose Street, Glasgow, G1 1XJ

Eduardo Blanco-Marigorta; Joaquín Fernández-Francos
Jorge L. Parrondo-Gayo; Carlos Santolaria-Morros
Universidad de Oviedo, Área de Mecánica de Fluidos
Campus de Viesques, 33271-Gijón (Asturias), Spain
jff@correo.uniovi.es

ABSTRACT

The power computers increase and the specific calculation software development have made possible, nowadays, the numerical simulation of flow and energy transfer inside the turbomachinery.

To teach Fluid Mechanics is not easy not only for the professors but also for the students because the theoretical part must be complemented with a technical part where students can see the phenomena. However, specially in hydraulic turbomachinery, we can't see the phenomena except if we have a specific material, for example a PIV. Even if we would have this material, the access to specific parts of turbomachinery is not possible due to its constructive layout.

The use of numerical simulation tools allows us to obtain data in inaccessible positions for the experimentation, as well as the study of unusual or dangerous performances.

With the numerical simulation, the pressure fluctuation at any point of the pump can be easily obtained. Other important results are the radial forces on the impeller, which have a significant variation with the working points.

One of the advantages of this kind of modelling is the ease to carry out changes in the geometry, parametric studies and analysis of anomalous operation conditions.

INTRODUCTION

Flow description in turbomachinery teaching become difficult due to the complexity of the geometry and the fact that these machines have both moving and fixed parts. The equations that describe the flow, the Navier-Stokes equations, have to be determined in two coordinate systems. In the moving part, the impeller, the momentum equations include the centrifugal and Coriolis forces, which is more challenging for the student to comprehend. In general, turbomachinery teaching is carried on starting with one dimensional theoretical models, complemented with laboratory sessions. The problem is that undertaking laboratory experiments is complicated because it is necessary to have expensive equipment and assistants helping the students in the task realization. Generally, only one study may be carried out in each equipment at one time. All this is a limit to the number of students that can be placed together in the laboratory. On the other hand, the practices of turbomachinery, especially in pumps, consist of obtaining only the characteristic curves, head, torque and efficiency versus flow rate. If we wish to visualize the interior flow in a turbomachine we must use special equipment, like PIV techniques. It is possible then, that the laboratory budget be excessive.

One of the advantage of CFD techniques in the realization of simulation practices is that we can have more students because an assistant can teach several groups. Furthermore, we can simulate a great deal of machines and obtain not only the performance, but also the flow details.

How far can we go with CFD? There are no apparent limits. Computers and commercial codes, year on year, advance with more power and the price continues coming to fall. Solving simulations up to 50000 cells poses no problems for the modern computers. The main problem with CFD software is that the students must have a minimum knowledge of numerical methods.

The numerical simulation of flows in turbomachines is not easy because of their moving parts. 2D simulation of axial machines is a little easier, because we can simplify the study putting the blades like an straight cascade. We consider that the flow leaving the stator goes inside the rotor with a velocity equal to the absolute velocity minus the rotor velocity. Simple CFD codes, without solving the flow in moving and fixed parts simultaneously, can simulate this process with good agreement.

In centrifugal turbomachinery the previous simplification is not possible. Even in the easiest case, to simulate the flow in a impeller channel, it is necessary to solve the equations taking into account the centrifugal effect.

Besides, the flow in a centrifugal pump, for example, is strongly three-dimensional and unsteady. Also, the interaction volute-impeller, due to the axial asymmetry of the volute, and the disposition of the tongue cause some effects difficult to simulate. The first effect generates an steady pressure distribution that is no-uniform at the impeller outlet causing the presence of a radial force there. The second effect, owing to the flow impact leaving the channel against the tongue, causes pressure pulsations that produce dynamical forces added to the previous.

The simplifications when we are simulating the flow in a pump have to take into account the importance of the effects that we do not include. The simulation of an isolated impeller or, by applying symmetry, in a channel, would not take account the two previous effects. Even the simulation of the entire machine in steady mode considers that the impeller always stays in the same position with respect to the volute. A quasi-steady solution could be made, repeating the calculation with different rotor positions, but even then, the time derivatives would not be correctly calculated. The two effects above mentioned could be seriously misinterpreted.

Advanced CFD codes could consider the full case, with sliding mesh in the mobile part. We introduce a time step and the program will characterize the flow in consecutive time instants separated a time step. In each time step the moving part rotates, changing its position. In this way we take into account the change of relative position between the volute and the impeller, and the unsteadiness of the flow.

TRAINING

One of the drawbacks of the use of CFD in complex fluid dynamic subjects as turbomachinery is that it requires a deep knowledge of both subjects. In our case, during the undergraduate engineering courses, the students only have ten hours of training. In this time, they solve two or three basic fluid mechanics problems with two-dimensional flow. Only during the postgraduate studies they can undertake a serious turbomachinery training.

The students who go on to make their final project in our department must have a supplementary instruction. We give them further fundamental references about CFD (Shaw, 1992; Versteeg & Malalasekera, 1995) and turbomachinery (Dixon, 1978; Stepanoff, 1957; Newmann, 1991), complemented with some basic numerical cases. All this provides the necessary help for them to face the project with only the usual guidance.

A reasonable time scale to undertake the project is about six months. In the first month, the student is dedicated full time to improve these kind of techniques, solving several cases and, step by step, he gets closer to the specific work that he had to solve. The next three months is dedicated to modelling and to solving his case taking account of all possible variables. While the computer is solving the simulation, the student prepares a literature survey. This documentation will be of great use in the analysis phase. In the last two months the student carries out the analysis and prepares the presentation of results.

CENTRIFUGAL PUMP SIMULATION

The numerical study of each kind of turbomachine has its own characteristics. Below there are commented some of the assumptions and effects that could be taken into account for the simulation of a centrifugal pump at the postgraduate level.

In these machines the geometry is highly three-dimensional. A two-dimensional model will eliminate the axial flow direction, with it we can not simulate the secondary flows. But due to the transfer of energy is brought about through the centrifugal forces that the impeller passes on to fluid, and this forces have radial and tangential directions, it is possible to consider that the model captures the fundamental phenomena that occur in the pump. Another consideration is that a three-dimensional model could be a little too much for an student.

With the two-dimensional model there are a problem with the volute because it is usually wider than the impeller outlet. If the geometry is strictly maintained, velocities in the volute are higher than what is desirable. In order that the curve of numerical head would better correspond with the experimental curve, the volute dimensions should be increased so that the volumetric relations are kept.

The blade number and shape has great importance. A pump with few backward blades gives less distortion in the flow and

it is possible to simulate it with a relatively rough mesh. A large number of blades requires a great number of cells to correctly simulate the flow passages. Forward blades, because of the severe stall that is usual in the suction side, needs a fine mesh in this area if we want to capture this phenomenon.

Although, most of CFD codes can do automatic meshing, obtaining a good grid is one of the tough challenges that student has to face. In turbomachinery, especially in the centrifugal devices, the best grid use to be an unstructured one.

It is necessary to select appropriate boundary conditions at the inlet and outlet. A fixed flow rate condition at the inlet and a static pressure condition at the outlet use to be a good choice, at least from the code convergence point of view. Boundary conditions of total pressure at the inlet and static pressure at the outlet is a more realistic situation. Inlet and outlet should be as far from the impeller as possible in order to avoid influence in boundary conditions.

About the turbulence, commercial CFD codes usually incorporate some models varying from the simple one equation to the more complex of Reynolds Stresses or eddy simulation. The existent literature contains information about which turbulence model describes better the flow. The student should made tests with some of them to investigate their influence.

EXAMPLE

Next a typical case is briefly described. It have been solved with the software FLUENT. Figure 1 is an sketch of the real pump. The simulation performed have been incompressible, two-dimensional, unsteady, with sliding mesh in the impeller, with the k-ε turbulence model. Figure 2 shows one of the grids tested. It has about 10.000 unstructured cells.

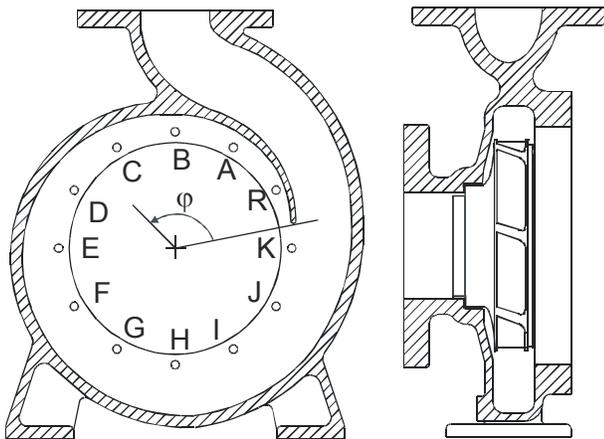


Figure 1: Sketch of real pump

As has been commented, there are great differences in performance with small changes in volute geometry. In figure 3 it can be observed the characteristic curve obtained with various models. It seems that the best choice is the geometry that keeps the real distance at the tongue and the volumes in the rest of the volute.

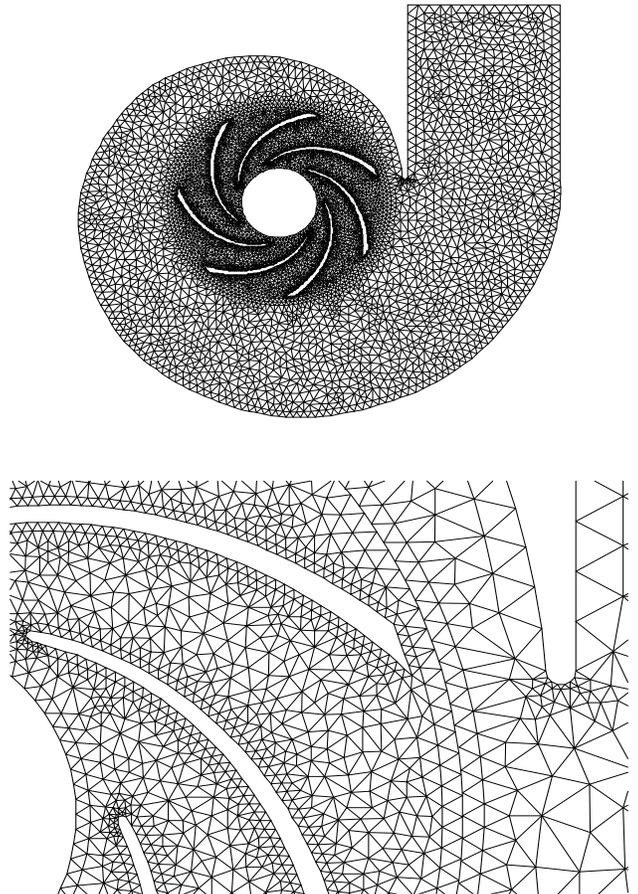


Figure 2: Grid tested and mesh detail

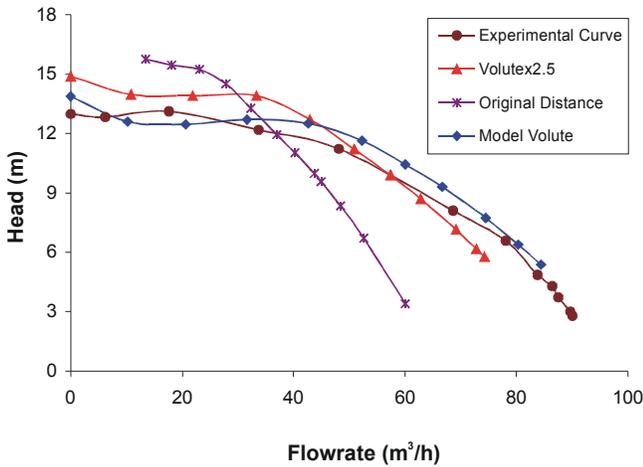


Figure 3: Characteristic curve

The solution gives a great number of data. Pressure and velocity could be obtained at any point of the geometry. For example, in figure 4 the pressure distribution around the impeller is plotted for several flow rates in a fixed time step. Figure 5 shows the radial forces over the impeller (time averaged) compared with the experimental ones.

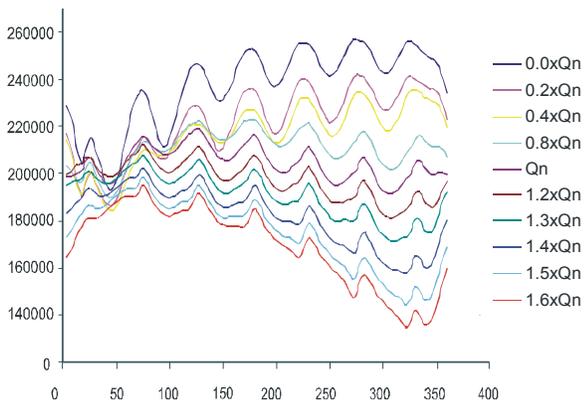


Figure 4: Pressure distribution around the impeller for several flow rates

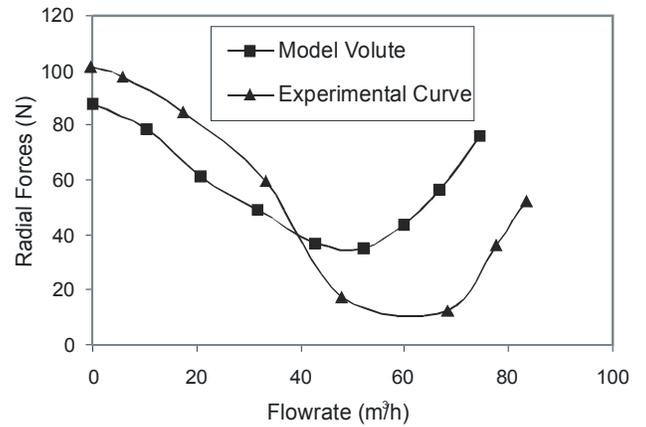


Figure 5: Experimental and numerical radial forces

Efficiencies and pressure fluctuations do not match exactly the experimental measurements. This can be attributed mainly to the geometry and two-dimensional assumptions. Nevertheless, the essential phenomena are properly reported. Pressure and velocity distributions allow students to understand how energy is transferred to the fluid, the difference between absolute and relative velocities, the effects arising for flow rates far away from the nominal point, etc.

The next figures are a sample of the analysis that can be done with the numerical data. Figure 6 represents the pressure contours of a simulation with the same boundary conditions in two different time steps. The unsteady simulation with the sliding mesh technique allows us to capture the existing pressure pulsations and their propagation along the volute.

The relative velocities inside the impeller are shown in the figure 7. The flow rate is lower than the nominal one and we could see the trend to separation on the suction side of the trailing edge.

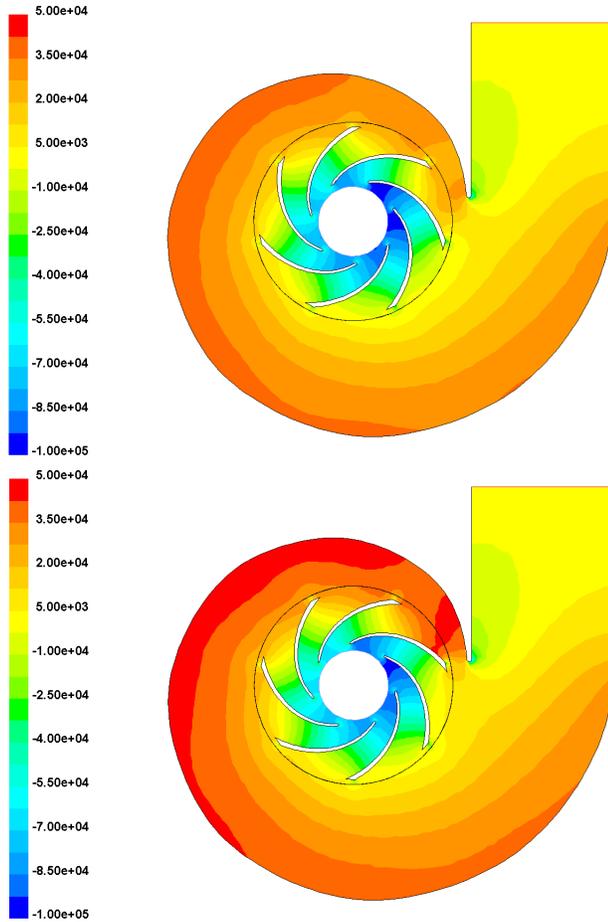


Figure 6: Numerical pressure contours in two different time steps

The absolute velocity vectors near the tongue for a flow rate greater than the nominal, are represented in figure 8. Here is clearly visible the separation on the outlet side and the blockage between the tongue and the impeller.

As a result of this work a short video film was created with the aim to clearly visualize the unsteady phenomena. To simulate the pump rotation, different frames from unsteady simulations were captured, one for each time step, and with commercial animation programs were edited into short clips. These sequences have arisen as a good tool for teaching.

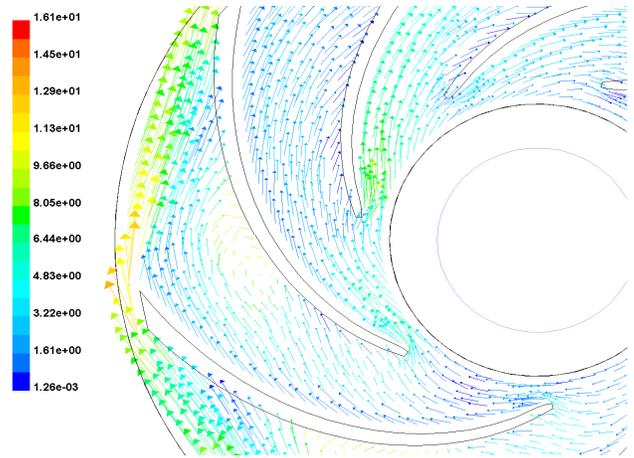


Figure 7: Relative velocities inside the impeller

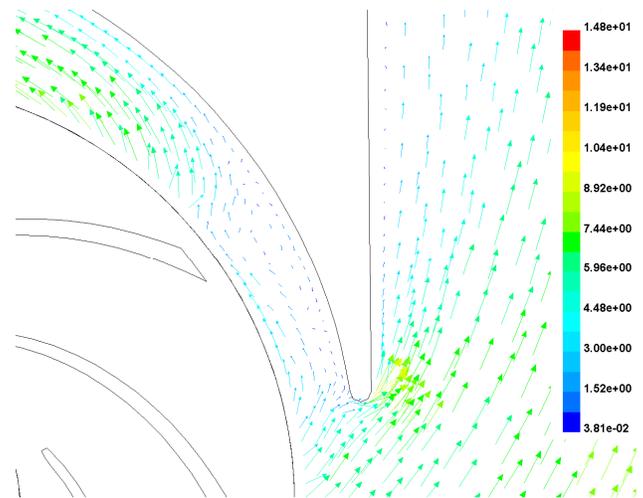


Figure 8: Absolute velocities near the tongue

CONCLUSIONS

Commercial CFD codes have reached such development that their study in engineering courses is almost necessary. Besides they are an useful teaching tool, not only in Fluid Dynamics but also in specialized subjects as turbomachinery.

Although students require some specific training to use properly this kind of software, benefits are relevant.

Some painful phenomena for a theoretical teaching: the energy transfer process, performance outside the nominal point, volute-impeller interaction, etc. are clearly shown with the help of unsteady numerical simulation.

ACKNOWLEDGMENTS

The authors gratefully acknowledge the financial support of the COMISIÓN INTERMINISTERIAL DE CIENCIA Y

TECNOLOGÍA under Project TAP-1199-CO2-02 titled “Sistema informático para la detección en línea de daños en turbomáquinas hidráulicas”.

REFERENCES

Dixon, S.L., 1978, “Fluid Mechanics, Thermodynamics of Turbomachinery”, Pergamon Press Ltd. Oxford.

Neumann, B., 1991, “The interaction Between geometry and Performance of a Centrifugal Pump”, Mechanical Engineering Publications Ltd.

Shaw, C.T., 1992, “Using Computational Fluid Dynamics”, Prentice Hall International (UK) Ltd.

Stepanoff, A. J., 1957, "Centrifugal and Axial Flow Pumps", J. Wiley & Sons, New York.

Versteeg, H.K. and Malalasekera, W., 1995, “An Introduction to Computational Fluid Dynamics”, Longman Scientific & Technical, Essex.