NUMERICAL FLOW ANALYSIS OF THE GAMM TURBINE AT NOMINAL AND OFF-DESIGN OPERATING CONDITIONS

Laurent Gros, Jean Louis Kueny, François Avellan
IMHEF-EPFL
33, av. de Cour, CH–1007 Lausanne, Switzerland
e-mail : laurent.gros@epfl.ch

Laurent Bellet
EDF-CNEH
Savoie Technolac, F-73373, Le Bourget-du-Lac, France
e-mail : laurent.bellet@edfgdf.fr

Abstract

The flow in a Francis turbine runner (GAMM Turbine) is analysed numerically. Different operating points are calculated using two industrial software packages based respectively on a finite element method (N3S) and a finite volume method (TASCflow®) and compared to experimental results. The numerical results allow to observe physical phenomena in the runner that are important in the process of hydraulic turbomachinery design. Values of C_u and C_m velocity components, blade pressure distribution and recirculation in the flow are compared to experimental results at nominal and off-design flow conditions. The experimental and numerical results show a similar efficiency evolution in function of flow rate and head, however the absolute level of energetic losses are overestimated by the two numerical codes.

1 INTRODUCTION

Many manufacters use commercial Navier-Stokes codes in their design process to analyse the flow in Francis turbines. Computer resources involved in the flow analysis should be compatible with the needs of design process of a runner. Therefore 8 hours of CPU time can be considered as acceptable for each operating point on a medium size power computer workstation. The aim of this paper is to compare calculations with experimental data for the case of the 500 GAMM Francis turbine for which a lot of experimental data are available at IMHEF, such as blade pressure measurements.
In the present study, both N3S and TASCflow® software, using a k-ε standard turbulent model, are used to analyse the behavior of the Francis runner. The numerical calculations are performed for five operating points corresponding to the experimental conditions and two extra conditions close to the best efficiency value (Figure 1). The mesh size is one criterion to perform calculations in an acceptable computation time. Coupled calculations taking into account rotor stator interaction (using the frozen rotor or average stage technique) are now available on commercial codes with acceptable computing efforts. This coupled calculation is performed at the best efficiency point and compared to experimental data and to the single stage calculation.

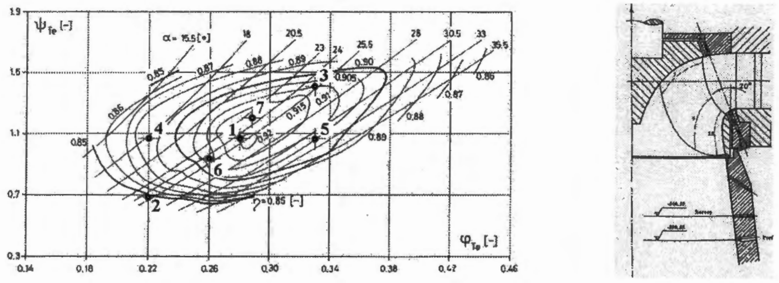


Figure 1: Operating conditions and Experimental surveys.

2 EXPERIMENTAL AND NUMERICAL ANALYSIS

2.1 Description of experimental and numerical investigations

The 500 GAMM Francis runner, designed at IMHEF, is a model used for experimental research study in the laboratory [1]. Flow surveys have been carried out with a five-holes pressure probe at the inlet and outlet sections of the runner for five different operating conditions (1,2,3,4 and 5) as shown in Figure 1. Pressure transducers mounted on the pressure and suction sides of the runner blades provide the pressure distribution along streamlines 2, 9 and 15 of the blade.

The absolute and relative flow angles α and β are used for the comparison, as well as the non-dimensional velocity and pressure values, defined as:

$$C_u = \frac{c_u}{\sqrt{2E}} \qquad C_m = \frac{c_m}{\sqrt{2E}} \qquad C_p^* = \frac{p^* - p^*_{ref}}{\rho E} \qquad \phi = \frac{Q}{\pi \omega R^3} \qquad \psi = \frac{2E}{\omega^2 R^2} \qquad \tan\alpha = \frac{C_u}{C_m}$$

where C_u, C_m and C_p^* are respectively related to the circumferential velocity, the meridian velocity and the specific potential energy.

The numerical calculations are performed for the five experimental points (1,2,3,4,5) and two other points (6 and 7), in order to understand better the behavior of the flow close to the B.E.P.

	Numerical Calculation - GAMM turbine		
	Single Stage Calculation		Coupled Calculation
Numerical Software	N3S [5] Electricité de France Finite Element	CFX-TASCflow [6] AEA-Technology Finite Volume	CFX-TASCflow AEA-Technology Finite Volume
Computer Workstation	IBM Risc6000	UltraSparc SUN	UltraSparc SUN
Geometry	CATIA CAD	ICEM-CFD HEXA	ICEM-CFD HEXA
Mesh	SIMAIL Unstructured Tetrahedron Element	ICEM-CFD HEXA Block Structured Hexahedron Element	TASCgrid Block Structured Hexahedron Element
Mesh size	93'500 Nodes	81'152 Nodes	59'500 Nodes
Analysis Software	CEI-Ensight	CEI-Ensight TASCtool	CEI-Ensight TASCtool
Inlet Section	$R = 239mm$	$R = 239mm$	$R = 370mm$
Oulet Section	$Z = -240mm$	$Z = -400mm$	$Z = -346mm$
Operating Points	1,2,3,4,5,6,7	1,2,3,4,5,6,7	1

Table 1: Characteristics of numerical calculations.

Figure 2: View of the mesh calculation domain : N3S and TASCflow.

2.2 Boundary conditions

The following boundary conditions are applied in the calculation for the 500 GAMM runner.
- At the outlet, we assume a free exit, i.e. no normal stress.
- On lateral faces periodic conditions are assumed in order to reduce calculation domain.
- Wall conditions are imposed on the shroud, the hub and the blade.
- At the inlet, three different inlet boundary conditions are used in order to compare the results and to define the best boundary conditions to impose:

- Uniform conditions defined by the flow rate and the guide vane opening corresponding to the experimental operating points.

- Boundary conditions for the runner-guide vane interaction:

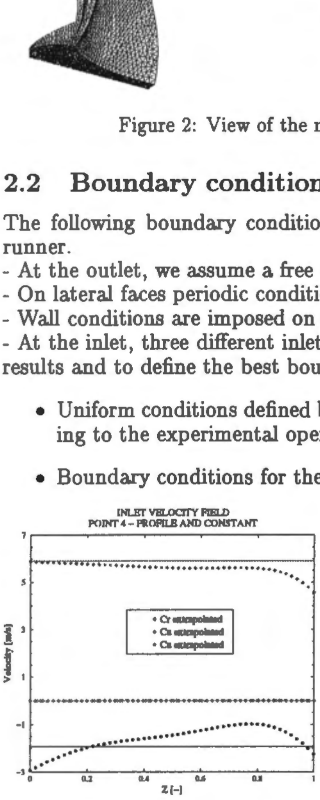


Figure 3: Inlet boundary conditions.

- Inlet conditions extrapolated from the flow survey at the inlet mesh domain. This extrapolation is calculated along streamlines in the hypothesis of potential flow. Figure 3 shows the difference between constant inlet conditions and those which are imposed with the extrapolation of experimental surveys.

- In the case of coupled runner / guide vane / stay vane calculations a constant α flow angle is imposed according to the α angle of the skeleton line at the leading edge of the stay vane.

3 RESULTS AND DISCUSSION

After control of numerical convergence of the flow rate, the specific energy and momentum, comparison between experimental data and numerical results obtained with N3S and TASCflow are proposed. The physical coherence is shown with the three following criteria: pathological cavitation behavior, recirculation and efficiency. Other comparisons are made on the experimental values at the inlet and outlet sections like C_u, C_m, α, β and pressure distibution on the blade. The analysis of the numerical results is fully in agreement with the results already described during the GAMM and Ercoftac workshop [4].

The coupled calculation is only done for the B.E.P, with constant inlet conditions imposed by the flow rate and the opening guide of stay vanes. The interest of this method although more important meshes efforts is to provide compatible runner inlet boundary conditions with guide stay vanes stage, during the design process.
This calculation gives correct results. In particular the inlet runner conditions (velocity components) are very well predicted at the best efficiency point (Figure 4). No signifiant differences are observed between single and coupled calculation. The calculation is therefore performed only on the runner in order to decrease the mesh size and the calculation time.

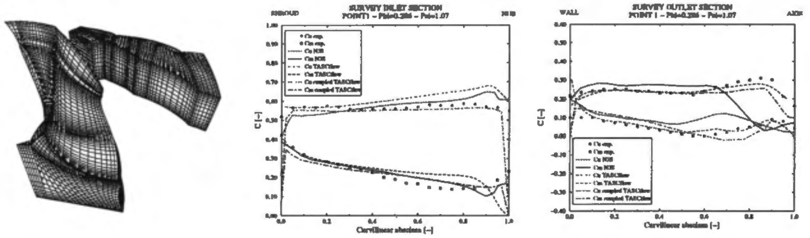


Figure 4: Coupled calculation : mesh, inlet and outlet survey sections.

For the single stage calculation, the pathological cavitation behavior is clearly illustrated by low pressure distribution on the blade suction side which corresponds to the top of the hillchart (Point 6,1 and 7 on Figure 5 and 6).

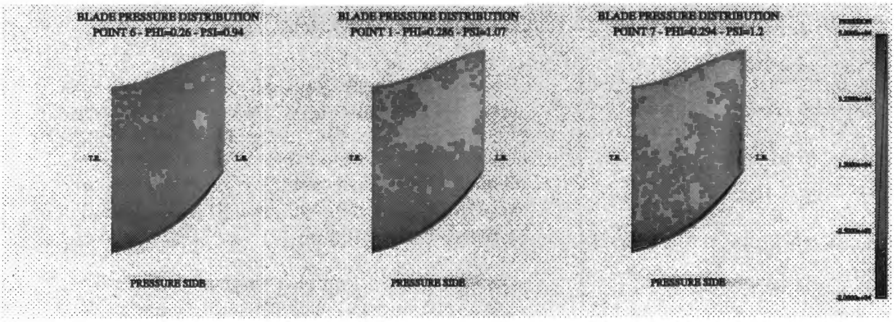


Figure 5: Blade pressure distribution : Point 6, 1 and 7 (N3S).

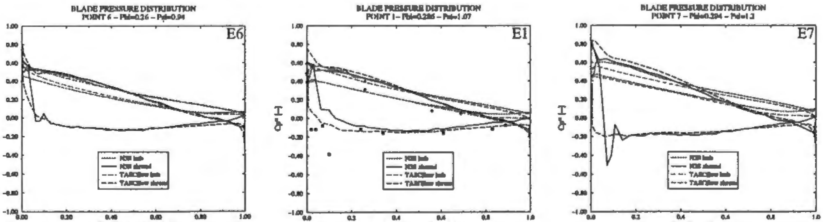
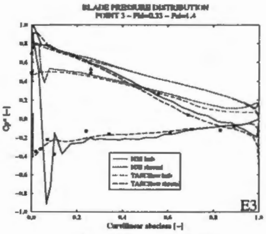
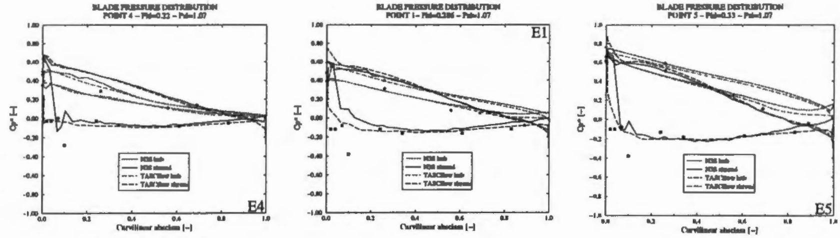


Figure 6: Blade pressure distibution on streamlines 2 and 15 : Point 6, 1 and 7.

There are no significant differences between blade pressure distributions given by N3S, TASCflow® and the experimental results (Figure 7) except for the pressure distribution at high ψ (Figures 6 E7 and 7 E3) where N3S predicts better the low pressure distribution on the suction side near the leading edge. The blade loading is well calculated. This result is very important especially for cavitation prediction with industrial codes.





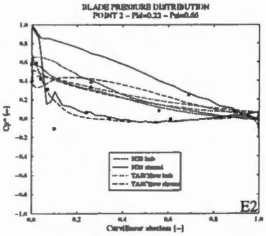


Figure 7: Blade pressure distribution on streamlines 2 and 15.

The flow recirculation below the runner hub corresponds to the partial load ($\psi = 0.66$). The meridian velocity factor is given for the same guide vane opening (Point 2, 1 and 3 in Figure 8). For point 2, we can clearly see the recirculation close to the axis of the runner which prevents from obtaining experimental data in this area (Figure 11 C2 and C4).

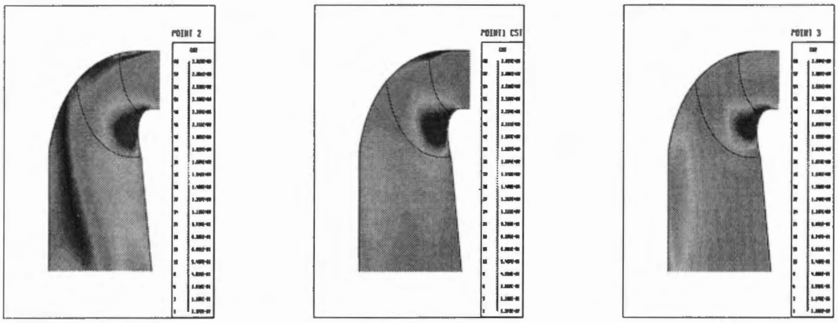


Figure 8: Meridional view in Cm/Cm_{inl} for points 2, 1 and 3.

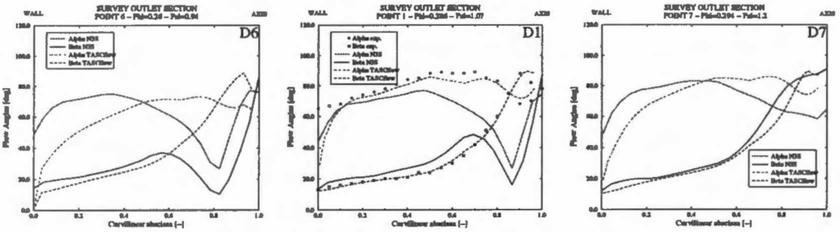


Figure 9: Outlet survey section : Point 6, 1 and 7.

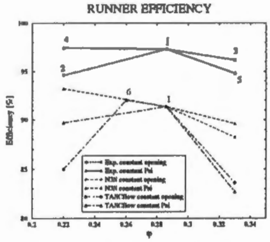


Figure 10: Runner efficiency.

Concerning the efficiency of the runner, energetic losses are overestimated with numerical simulations due to the turbulence model used for calculations and the log-law wall treatment. However the shape of the efficiency curve for constant opening angle and constant ψ is comparable with experimental data and the localisation of the B.E.P is well estimated (Figure 10).

For the inlet the numerical results coincide well with experimental flow surveys for the both codes. We only compare the results at outlet section.

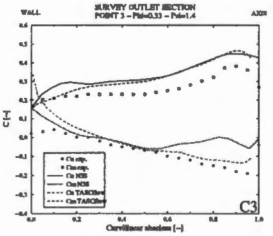
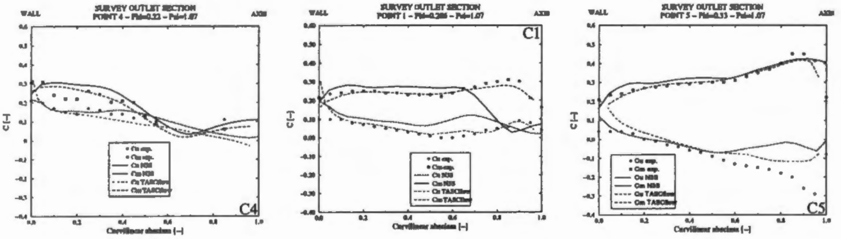
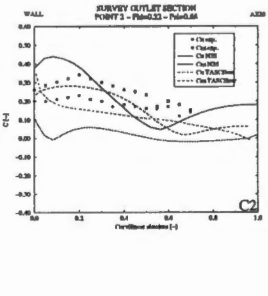


Figure 11: Outlet survey section.





The evolution of the velocity components C_u and C_m is especially well calculated by TASCflow® (Figure 11 and 12) for all operating points. At constant opening (24 [deg]) the prediction of the stall below the hub with N3S is shifted higher ϕ and this phenomenon explains the differences observed on Figure 9 D6, D1 and D7 for points 6, 1 and 7. However the evolution of the flow pattern is therefore well predicted with a ϕ delay.

For low flow rate (point 2), the finite element software N3S gives very bad results. Values of velocity components C_u and C_m and flow angle α and β (Figure 11 and 12) are very different for experimental values due to the recirculation at the outlet. And it is because this problem was partended by old N3S calculations that the outlet of the calculation domain is limited at $Z = -240mm$.

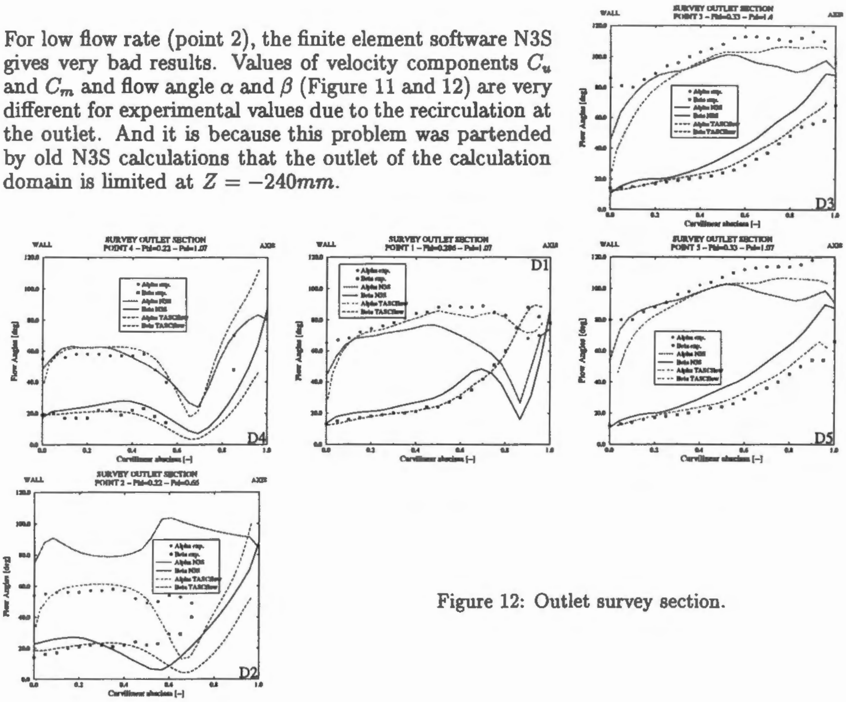


Figure 12: Outlet survey section.

4 CONCLUSION

The single runner computation in a Francis turbine for nominal and off-design points is performed with two industrial Navier-Stokes codes. The mesh size is intentionally limited to 100′000 nodes in order to obtain acceptable CPU times on a classical workstation. The most relevant information for the design process is well predicted : blade incidence, B.E.P localisation, pressure evolution on blade, ... The global evolution of main flow patterns are well predicted by calculations (occurence of stall, torche behavior at runner outlet, ...) . Single stage calculations provide good results if the inlet boundary conditions are well defined. In our case these conditions are extrapolated from flow surveys but adapted boundary conditions can be obtained using coupled runner/stay vanes calculations. The coupled computation gives correct results. However to validate this coupling it is necessary to perform calculations for the off-design operating conditions.

Acknowledgment

The authors wish to thank Electricité de France Company and the IMHEF staff for its support.

Bibliography

[1] Avellan F. & al - Flow survey and blade pressure measurements in a Francis turbine model - XV IARH, 1990 - Belgrade, Yugoslavia.
[2] Proceedings of the GAMM workshop, 1989, Lausanne, to be published by Vieweg Verlag in the series Notes on Numerical Fluid Mechanics.
[3] Parkinson E. - Turbomachinery workshop Ercoftac II- Test case 8- Francis turbine - 1995 -IMHEF, Lausanne, Switzerland.
[4] Notes on Numerical Fluid Mechanics - Vol 39 - 3D Computation of Incompressible Internal Flows - Edited by Gabriel Sottas and Inge L. Ryhming.
[5] EDF - SIMULOG - N3S Turbomachines, Manuel Utilisateur.
[6] AEA - Technology - TASCflow3d. User Documentation.