IV International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2011
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

NUMERICAL STUDY OF THE FLUID – STRUCTURE INTERACTION IN THE DIFFUSER PASSAGE OF A CENTRIFUGAL PUMP

ALFRED FONTANALS[*], ALFREDO GUARDO[†*], MIGUEL COUSSIRAT[††] AND EDUARD EGUSQUIZA[†]

*Escola Universitaria d'Enginyeria Tècnica Industrial de Barcelona.
C\ Compte d'Urgell, 187. 08036. Barcelona, Spain.

[†] Centre de Diagnòstic Industrial i Fluidodinàmica (CDIF), Universitat Politècnica de Catalunya
Av. Diagonal, 647. ETSEIB. – 08028 Barcelona, Spain
e-mail: alfredo.guardo-zabaleta@upc.edu – Web page: http://www.upc.edu/cdif/

[††] Electromechanic Department. FRM, Universidad Tecnológica Nacional
Coronel Rodríguez 273 – 5500. Mendoza, Argentina

Key words: Fluid-structure interaction.

Abstract. Reliable design of turbomachinery is a complex task. In order to gain overall efficiency in the machine performance, designers may have to reduce the gap between the impeller and the diffuser, forcing them to be as closely spaced as possible. In these situations, there may be a strong interaction between them that influences both the aerodynamics and the structural performance of blades and vanes. This phenomenon is called rotor-stator interaction (RSI), and it has a strong influence on the machine behavior. These interactions can have a significant impact on the vibrational and acoustical characteristics of the machine [1-2]. Sometimes, this interaction has led to blade or vane failure [3].

Unsteadiness and turbulence play a fundamental role in RSI [4-5], and the use of computational fluid dynamics (CFD) is becoming a usual requirement in turbomachinery design due to the difficulties and elevated cost of the experimentation required to identify RSI phenomena. Nowadays, a CFD analysis based on Reynolds–averaged Navier–Stokes equations (RANS) and a coupled eddy viscosity turbulence model (EVM) is commonly applied in turbomachinery design. Therefore, the choice of an appropriate turbulence model and the boundary layer treatment is far from trivial, and a suitable turbulence modeling plays an important role for successful CFD results.

In this work, an entire stage of a diffuser pump was modeled by means of a commercial CFD code in order to study the pressure fluctuations due to the interaction between the impeller and the diffuser of the pump. The obtained numerical results were compared against the experimental results of Tsukamoto et al., [6]. Full RANS equations coupled with several EVM were solved for a diffuser pump stage in order to establish the most accurate modeling strategy for a diffuser pump. Boundary layer sensitivity tests were performed, and numerical discretization influence on results was also tested and established. Frequencies of the pressure fluctuations in the diffuser passage are also obtained with several EVM and compared against experimental results.

1 INTRODUCTION

In turbomachinery design, the Rotor-Stator Interaction (RSI) is an important phenomenon, with a strong influence on the machine behavior. These interactions can have a significant impact on the vibrational and acoustical characteristics of the machine [2, 3]. Unsteadiness and turbulence play a fundamental role in complex flow structure [7, 8], and the use of Computational Fluid Dynamics (CFD) is becoming a usual requirement in design in turbomachinery due to the difficulties and high cost of the necessary experiments needed to identify RSI phenomena.

The RSI can be divided into two different mechanisms: potential flow interaction and wake interaction [1] The nature of the flow due to the wake interaction is unsteady and turbulent and, in the case of flow in turbomachines, there also appear three-dimensional boundary-layer, curvature and system rotation effects.

Nowadays, a CFD analysis based on Reynolds-Averaged Navier-Stokes equations (RANS) and a coupled eddy viscosity turbulence model (EVM) is commonly applied in turbomachinery design. Therefore, the choice of an appropriate turbulence model and the boundary layer treatment is far from trivial, and a suitable turbulence modeling plays an important role for successful CFD results [4]

In this work, an entire stage of a diffuser pump was modeled, by means of a commercial CFD code, in order to study the pressure fluctuations due to the interaction between the impeller and the diffuser of the pump, and the obtained numerical results were compared against the experimental results of Tsukamoto et al. [6] Full RANS equations coupled with several EVM were solved for a diffuser pump stage in order to establish the most accurate modeling strategy for a diffuser pump. Boundary layer sensitivity tests were performed, and numerical discretization influence on results was also tested and established.

2 NUMERICAL MODELING

2.1 Geometry and Grid Generation

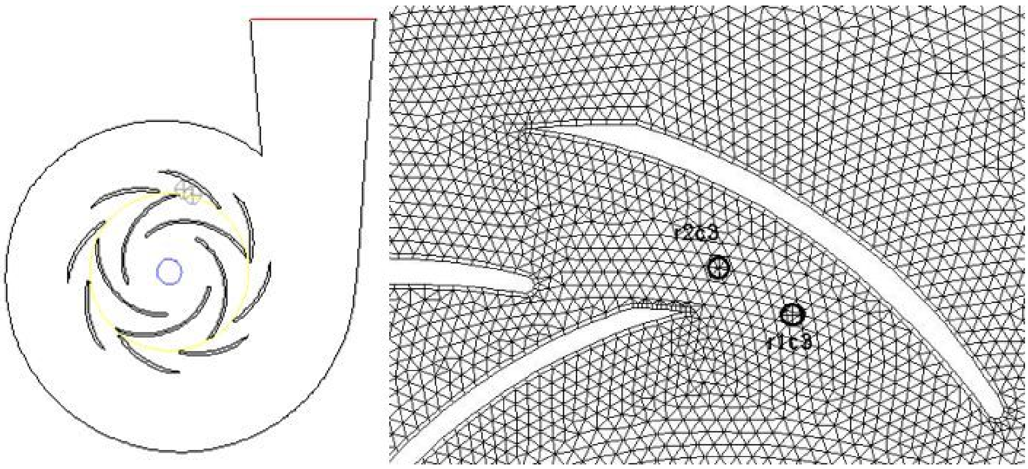
The modeled test pump was a single-stage diffuser pump with five impeller blades, N_i=5, eight diffuser vanes, N_v=8, and volute casing (Figure 1). A detailed description of the pump is given by Tsukamoto et al. [6].

Table 1: Mesh sensitivity test

Mesh	Number of cells	y^+
1	$4{,}8 \times 10^4$	20-200
2	$9{,}7 \times 10^4$	20-200
3	$3{,}2 \times 10^5$	20-200
4	$8{,}1 \times 10^4$	1-10
5	$1{,}4 \times 10^5$	1-10
6	$3{,}0 \times 10^5$	1-10

A mesh sensitivity test was performed in order to evaluate the independency of the numerical results from the mesh density. Also, the effect of the boundary layer treatment on the results was performed. For accomplishing the aforementioned, for 3 different mesh densities the boundary layer around the blades, vanes and walls was modeled using a wall function (WF), with a y^+ ranging from 20 to 200, and another three mesh densities were tested using a two layer model (TLM), with a y^+ ranging from 1 to 10. For further details on the meshes used see Table 1.

Figure 1: Geometry and studied point position in the vaned diffuser stage.



2.2 Unsteady Calculations Setup

Two dimensional, unsteady incompressible Reynolds-averaged Navier-Stokes equations were solved by means of the commercial CFD code Ansys Fluent 12.1 [9]. A constant pressure value was imposed at the fluid inlet and a constant pressure value was imposed at the pump outlet. A non slip boundary condition was imposed at the runner blades, diffuser vanes and volute casing wall. A rotational speed of 2066 rpm was imposed to the blade impeller.

One of the goals of this work was to benchmark the ability of different turbulence models in capturing the pressure fluctuations and the characteristic frequencies in the diffuser according to the experimental results reported by Tsukamoto et al. [6]. Turbulence models employed in its work are listed in Table 2.

In order to accomplish the proposed turbulence model performance study, pressure fluctuations in two points of the vaned diffuser passage, which correspond to the reported measurement points for the experimental data set, were recorded for every numerical simulation developed. Location of the monitor points r1c3 and r2c3 can be seen in Figure 1.

The unsteady formulation used was a second-order implicit velocity formulation and a pressure-based solver was chosen. The SIMPLE pressure-velocity coupling algorithm was used, and second order scheme discretization was selected for the numerical experiments.

Table 2: Turbulence models used

#	Turbulence model	Reference
1	Spalart – Allmaras	Spalart and Allmaras [10]
2	Standard k-ε	Launder and Spalding [11]
3	Realizable k-ε	Shi et al. [12]
4	RNG k-ε	Yakhot and Orszag [13]
5	Standard k-ω	Wilcox [14]
6	Shear Stress Transport (SST) k-ω	Menter[15]
7	Reynolds Stress Model (RSM)	Launder et al. [16]

The maximum number of iterations for each time step was set to 40 in order to reduce all computed normalized numerical residuals to $1x10^{-5}$. The interface between the rotor blade and the diffuser vane was set to a sliding mesh, in which the relative position between the rotor and the stator was updated every time step. The adopted computational time step was about 1/360 of the rotor revolution time.

3 RESULTS AND DISCUSSION

Due to the unsteady nature of the flow, it is required that the whole flow domain is affected by the unsteady fluctuations. In order to check the aforementioned situation, a flow rate monitoring was made at the domain outlet. Uniform unsteady flow behavior was reached after 10 revolutions.

3.1 Pressure Fluctuations

For the two monitored points, r1c3 and r2c3, in the diffuser vane the unsteady static pressure was obtained, in order to determine the effect of the RSI in the diffuser vane. The results obtained for several turbulence models of the non-dimensional unsteady pressure, $\Delta\Psi$, are shown in Figures 2 and 3.

Results obtained show that the highest pressure value in the diffuser is due to the potential effect generated by the movement of the rotor vane in front of the diffuser vane. The wakes at the rotor outlet enter the diffuser and interact with the vanes producing high pressure peaks.

When comparing the numerical results obtained with each one of the turbulence models tested, it can be observed that there’s no significant difference in the results of pressure fluctuation. Only the Standard k-ω turbulence model diverge from others turbulence models results and experimental results.

When analyzing the effect of the boundary layer treatment on the results, it can be observed that WF are able to accurately reproduce the pressure fluctuations due to RSI, while a TLM does not lead to an improve of the results, even when using a turbulence model that takes into account the boundary layer transition from laminar to turbulent flow, as the Standard k-ω or the SST k-ω.

A detailed analysis of the obtained results allow to notice that using a WF, the Spalart-Allmaras, Realizable k-ε and SST k-ω turbulence models show the best fitting for the experimental pressure fluctuations (Figure 4).

When a TLM is used as near-wall treatment, the best fitting obtained for pressure fluctuation correspond also to Spalart-Allmaras, Realizable k-ε and SST k-ω turbulence models.

Previous studies carried out by our research group [4] showed that the SST k-ω turbulence model presented a good performance when determining the velocities in the boundary layer and the wake vortex shedding when studying flow around an isolated foil, and also a low computational cost if compared with more accurate (but more resource-consuming) turbulence models as RSM or Large Eddy Simulation (LES).

Figure 2: Computed non dimensional unsteady pressure, ΔΨ, at point r1c3. Boundary layer treatment: left, wall functions; right, two layer model.

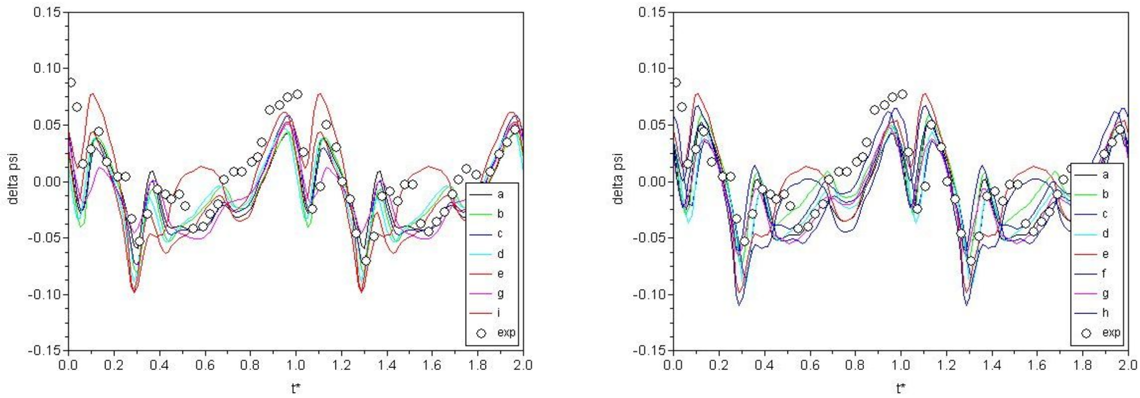


Figure 3: Computed non dimensional unsteady pressure, ΔΨ, at point r2c3. Boundary layer treatment: left, wall functions; right, two layer model

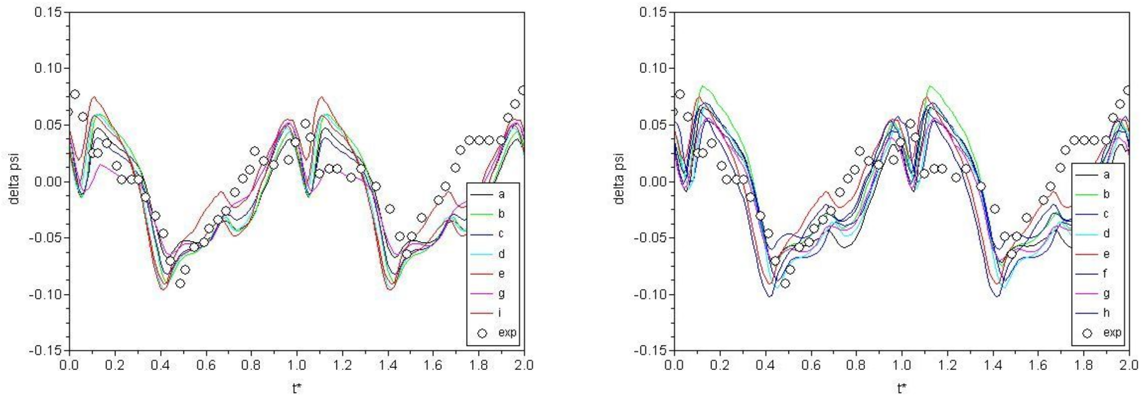


Figure 4: Computed non dimensional unsteady pressure, ΔΨ. Wall function boundary layer treatment: left, point r1c3; right, point r2c3.

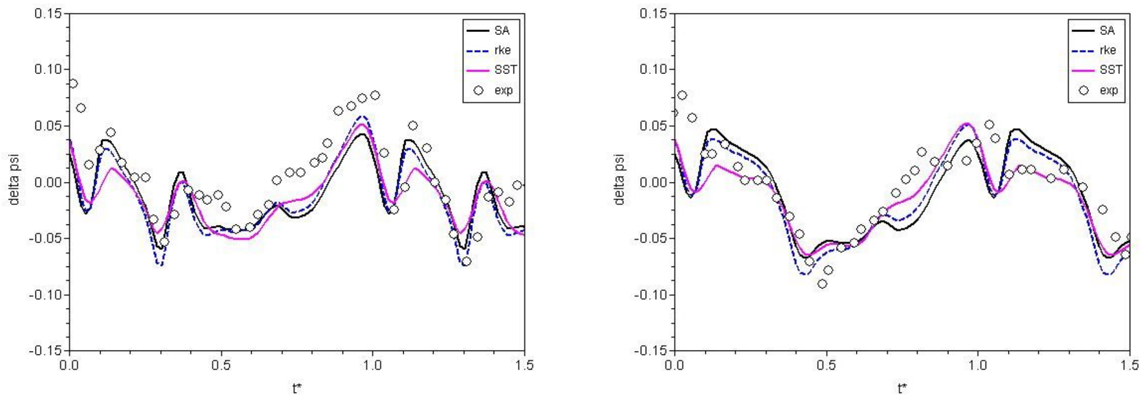


Table 3: Pressure fluctuations in frequency domain

	Frequencies		1Z_iN	2Z_iN	3Z_iN	4Z_iN	5Z_iN
	Experimental		172	344	517	689	861
	Turbulence model	Pressure Point					
a	Spalart-Allmaras	r1c3	171	343	518	689	861
		r2c3	171	343	518	689	861
b	Standard k-ε	r1c3	171	346	517	689	860
		r2c3	171	346	517	689	860
c	Realizable k-ε	r1c3	171	343	518	689	861
		r2c3	172	343	518	689	861
d	RNG k-ε	r1c3	171	343	517	689	861
		r2c3	171	343	517	689	861
e-f	Standard k-ω	r1c3	-	345	516	688	862
		r2c3	171	-	516	-	861
g-h	SST k-ω	r1c3	171	343	-	689	862
		r2c3	171	-	515	689	862
i	RSM	r1c3	172	343	517	689	861
		r2c3	172	343	517	689	861

3.1 Frequencies

In order to capture the RSI effects, the relationship between the pressure fluctuations and the movement of the rotor vanes in front of the diffuser vanes was determined. Using a Fourier transform, the characteristic frequencies of the pressure fluctuations have been obtained at points r1c3 and r2c3, resulting that the pressure fluctuates with the impeller blade passing frequency Z_iN and its higher harmonics. Table 3 shows the obtained frequencies with different turbulence models and the experimental results. It can be noticed that the k-ω family turbulence models are not able to capture the representative frequencies while all the other models benchmarked accurately capture the characteristic frequencies of the phenomena.

Figure 5: Spalart-Almaras model. Left: calculated unsteady pressure at point r1c3; right: frequency domain.

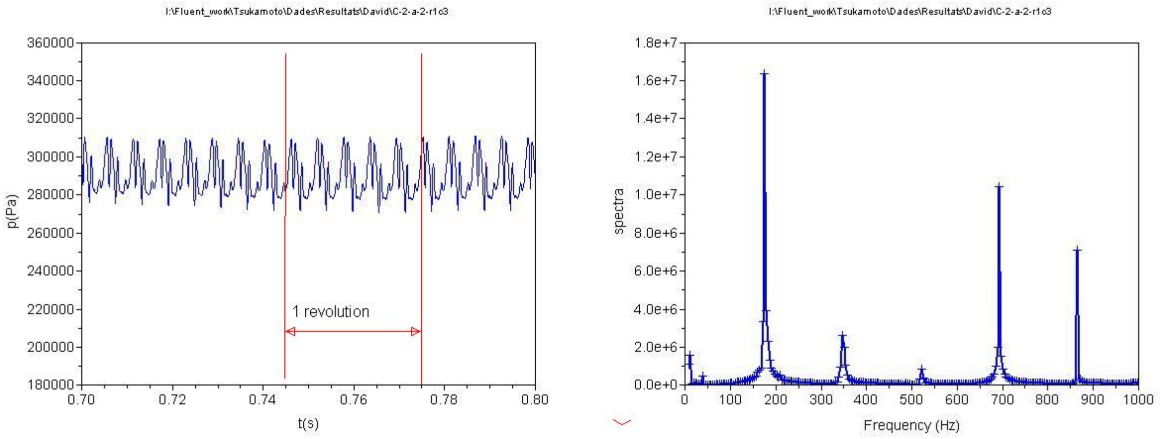


Figure 6: Realizable k-ε model. Left: calculated unsteady pressure at point r1c3; right: frequency domain.

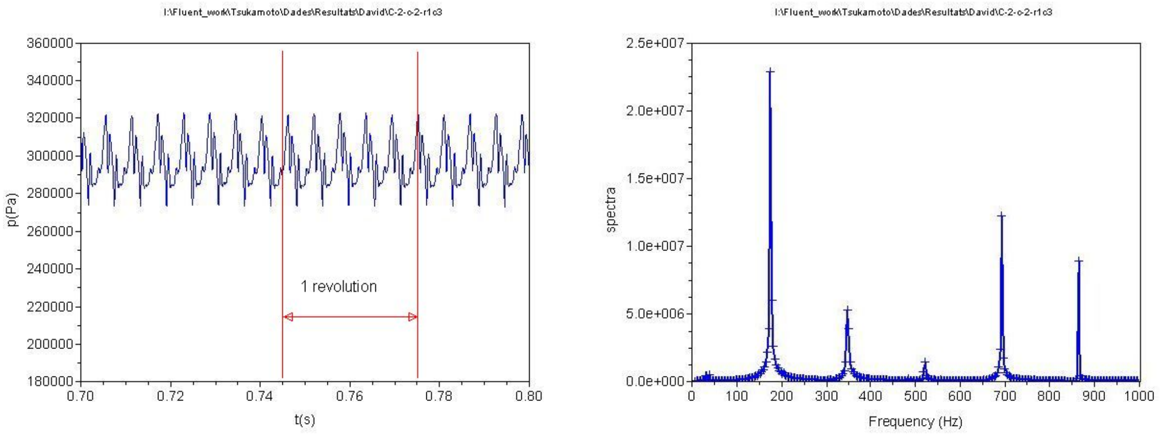
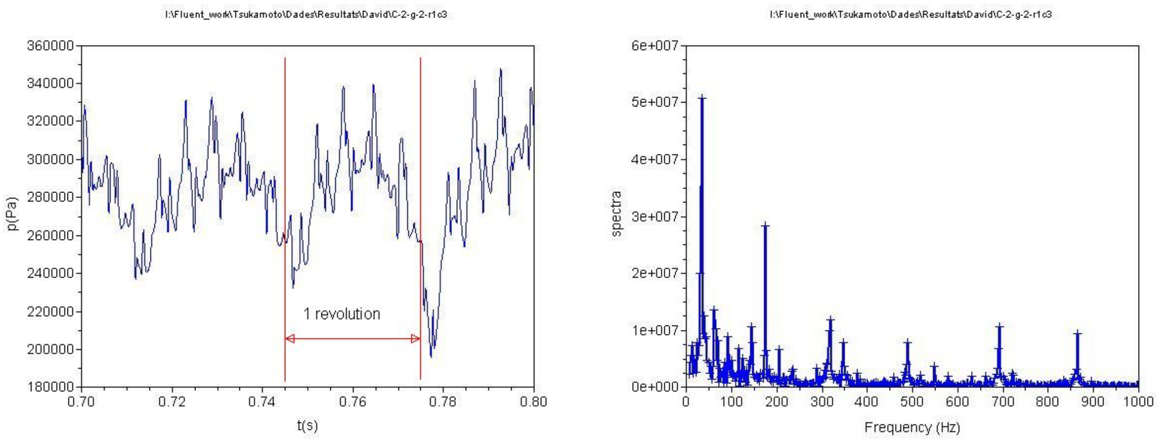


Figure 7: SST k-ω model. Left: calculated unsteady pressure at point r1c3; right: frequency domain.



Figures 5, 6 and 7 show in detail the pressure fluctuations at point r1c3 and its correspondent Z_iN frequencies (after treating the obtained pressure data with a Fourier transform) for 3 different turbulence models: Spalart-Allmaras, Realizable k-ε and SST k-ω.

For the Spalart-Allmaras and Realizable k-ε turbulence model (Figures 5 and 6), it can be noticed that for 1 revolution of the rotor, the pressure fluctuation generated by the movement of the rotor vanes follows a periodic pattern, and the frequency domain accurately registers the data of the impeller blade passing frequency Z_iN.

For the SST k-ω turbulence model (Figure 7), pressure fluctuation show different values along 1 revolution of the rotor for each passing of the impeller blade in front of the diffuser vanes, showing also a cyclic pattern after each revolution. The presence of all this 'noise' in the numerical results obtained makes it difficult to determine the impeller blade passing frequency Z_iN, as it translates into non-physical peaks in the frequency domain. This behavior pattern was also noticed for the Standard k-ω turbulence model.

4 CONCLUSIONS

A 2-D CFD has been applied to the study of RSI in the vaned diffuser of pump. Reynolds-averaged Navier-Stokes equations with several turbulence models were tested in order to check their capability to capture RSI phenomena in the pump. Models tested showed good results of pressure fluctuation in the vaned diffuser when comparing against experimental results, except the Standard k-ω model.

A detailed analysis of the obtained results allow to notice that the Spalart-Allmaras, Realizable k-ε and SST k-ω turbulence models show the best fitting for the experimental pressure fluctuations

When analyzing the effect of the boundary layer treatment on the results, it can be observed that WF are able to accurately reproduce the pressure fluctuations due to RSI, while a TLM does not lead to an improve of the results, even when using a turbulence model that takes into account the boundary layer transition from laminar to turbulent flow, as the Standard k-ω or the SST k-ω.

The relationship between the pressure fluctuations and the movement of the rotor vanes in front of the diffuser vanes was determined. The characteristic frequencies of the pressure fluctuations were obtained, resulting that the pressure fluctuates with the impeller blade passing frequency Z_iN and its higher harmonics. It was noticed that the k-ω family turbulence models were not able to capture the representative frequencies while all the other turbulence models benchmarked accurately captured the characteristic frequencies of the phenomena.

All the turbulence models tested shoed a periodic pattern in the pressure fluctuation in which each cycle is produced by the movement of a rotor blade in front of a diffuser vane, except for the k-ω turbulence models family. For these models, pressure fluctuations value was different for each pass of an impeller blade in front of a diffuser blade, but the overall behavior for 1 revolution of the impeller followed a regular pattern.

Spalart-Allmaras and Realizable k-ε turbulence models show the most accurate results of the benchmarked models, making them a suitable option for CFD modeling of RSI in turbomachinery. It is important to remark that Spalart-Allmaras turbulence model has a smaller computational cost, as it is a 1-equation model.

Previous work of our research group showed that SST k-ω turbulence models was a good choice for estimating boundary layer velocities and wake vortex shedding around isolated foils, but results obtained in present work show that when RSI potential effect is present, the numerical response of the model generates background noise in the results, making it difficult to clearly differentiate the characteristic frequencies of the phenomena..

In order to obtain a better understanding of the RSI phenomena it would be necessary to determine the influence of the discretization schemes, the time step value. These numerical studies are to be performed in the near future.

5 NOMENCLATURE

ψ = instantaneous pressure coefficient = $(p - P_s)/\left(\rho U_2^2/2\right)$

Δψ = non-dimensional unsteady pressure = $\tilde{p}^* /\left(\rho U_2^2/2\right)$

p = instantaneous static pressure = $\bar{p} + \tilde{p}$

$\bar{p}$ = time average p

$\tilde{p}$ = unsteady component of p

p^* = relative pressure = $p - P_s$

P_s = total pressure at pump suction port

U_2 = peripheral speed of impeller = $\pi D_2 N$

N = rotational speed

ρ = density

t^* = non-dimensional time = t/T_i

t = time

T_i = time required to traverse one pitch of impeller blade $= 1/(N Z_i)$

r = radius

c = symbol of pressure traverse line

Z = number of blades

ACKNOWLEDGEMENTS

Funding from the Spanish Ministry of Science and Innovation (Grant No. DPI 2009 – 12827) is appreciated. The authors also appreciate the contribution from EUETIB - UPC in the development of this work.

REFERENCES

[1] Ardnt, Acosta, Brennen and Caughey. Rotor-Stator Interaction in a Diffuser Pump. *J. of Turbomachinery* (1989) **111**:213-221.

[2] Chow, Y., Uzol, O. and Katz, J. Flow nonuniformities and turbulent “hot spots” due to wake-blade and wake-wake interaction in a multi-stage turbomachine. *J. of Turbomachinery* (2002) **124**:553-563.

[3] Uzol, O., Chow, Y., Katz, J. and Meneveau, C. Experimental investigation of unsteady flow field within a two-stage axial turbomachine using particle image velocimetry. *J. of Turbomachinery* (2002) **124**:542-552.
[4] Coussirat, M., Fontanals, A., Grau, J., Guardo, A. and Egusquiza. E. CFD study of the boundary layer influence on the wake for turbulent unsteady flow in rotor-stator interaction. *IAHR 4th. Symposium on Hydraulic Machinery and Systems*. Foz do Iguassu, Brazil (2008).
[5] Dring, Joslyn, Hardin And Wagner H. Turbine Rotor-Stator Interaction. *J. Eng. for Power* (1982) **104**:729-742.
[6] Tsukamoto, H., Uno, M., Hamafuku, N., And Okamura, T. Pressure fluctuations downstream of a diffuser pump impeller. *The 2nd Joint ASME/JSME Fluids Engineering Conference, Forum of unsteady flow, FED* (1995) **216**:133-138.
[7] Henderson, A., Walker, G. and Hughes, J. The influence of turbulence on wake dispersion and blade row interaction in an axial compressor. *J. of Turbomachinery* (2006) **128**:150-165.
[8] Soranna, F., Chow, Y., Uzol, O. and Katz, J. The effect of inlet guide vanes wake impingement on the flow structure and turbulence around a rotor blade. *J. of Turbomachinery* (2000) **128**:82-95.
[9] Fluent Inc. Fluent 6.3. *User's guide*, (2006).
[10] Spalart, P.R. and Allmaras, S.R. A one equation turbulence model for aerodynamic flow. *La Recherche Aérospatiale* (1994) **1**:5-21.
[11] Launder, B.E. and Spalding, D.B. *Lectures in mathematical models of turbulence*, Academic Press, London (1972).
[12] Yakhot, V. and Orszag, S. Renormalization group analysis of turbulence: I Basic theory, *J Sci Comput* (1986) **1**:3-51.
[13] Shih, T. Liou, W. Shabbir, A. Yang, Z. and Zhu, J. A new k-ε eddy viscosity model for high Re turbulent flow – Model development and validation, *Comput Fluids* (1995) **24**:227-238.
[14] Wilcox, D.C. *Turbulence model for CFD*, DCW Industries Inc., California (1998).
[15] Menter, F.R., Two equations eddy-viscosity turbulence models for engineering applications, *AIAA J* (1994) **32**:1598-1605.
[16] Launder, B.E. Reece, G.J. and Rodi, W. Progress in the development of a Reynolds stress turbulence closure, *J Fluid Mech* (1975) **68**:537-566.