Application of Fluid/Structure Interaction Methods to Determine the Impeller Orbit Curves of a Centrifugal Pump

FRIEDRICH-KARL BENRA
Faculty of Engineering Sciences
Department of Mechanical Engineering, Turbomachinery
University of Duisburg-Essen
Lotharstr. 1, 47048 Duisburg, Germany
http://www.uni-due.de/tm/

Abstract: - Sewage water pumps very often are equipped with impellers featuring a small number of blades which therefore provide big cross sections of the flow channel. Single-blade impellers comprise the smallest possible number of blades and are known as non-clogging pumps. But severe disadvantages occur during operation, because the time-variant flow in such pumps results in periodic unsteady flow forces which affect the impeller producing strong radial deflections of it. In order to determine the oscillations of the pump rotor induced by the exciting forces a coupled solution of the flow field in the pump and the structural mechanics of the pump rotor (FSI method) is required. In the present paper simulations which accomplish different coupling methods of the fluid dynamics and the structural dynamics have been carried out for a single-blade pump. In a first approach an explicit partitioned coupling method with simple interchange of boundary conditions has been applied. This approach does not take the feedback of the structural analysis to the flow into account. The results comply only roughly with the measurements. In a second attempt a full coupling of fluid and structural dynamics has been carried out. The simulations were coupled by transferring the computed flow forces as a load for the structural mechanics and the calculated impeller deflection acts as a deformation of the computational grid for the solution of the fluid dynamics. The impeller orbit curves obtained by this method compare much better to the measurements.

Key-Words: - Fluid/structure interaction, centrifugal pump, impeller orbit curves

1 Introduction

In general, Fluid-structure interaction (FSI) can be described as the mutual impact of different physical phenomena on a flow system. The impact on the system is recordable only by simultaneous applying the laws of the involved physical disciplines. So FSI is a subset of multi-physics applications which typically involve solving coupled systems of partial differential equations from multiple physical models. For example it combines the following simulations: fluid dynamics, structure dynamics, thermal, acoustic, magnetic, electromagnetic simulations and so on. A very common description of FSI can be obtained from [1]: *Coupled systems and formulations are those applicable to multiple domains and dependent variables which usually describe different physical phenomena and in which neither domain can be solved while separated from the other and neither set of dependent variables can be explicitly eliminated at the differential equation level.*

Fluid-structure interaction is an effect which can be watched in the nature and in the technique as well. For instance, bending a palm tree by the storm is a kind of fluid-structure interaction. Or in a more technical sense, the swinging of a bridge as the consequence of the wind is another example. For both examples the feedback from the structure to the fluid is only very small and it is not even important for the system. So in such cases this feedback can be neglected. Nevertheless the flow forces acting at the structure can introduce severe problems to the structure. The determination of the behavior of the structure is only possible with the knowledge of the forces and moments, which are exerted by the fluid on it. The probably most famous example of such a Fluid-Structure Interaction is the collapse of the Tacoma-Narrows Bridge due to exciting flow forces in November 1940. In other cases the feedback from the structure which is affected by the flow changes the flow behavior in a considerable way. Exact solutions of such fluid-mechanical problems are only feasible by considering the behavior of common boundaries for both fluid and structure domains in order to take the interaction between fluid and structure into account. In figure 1 the different numerical disciplines which are involved for FSI-solutions are shown in a schematic view.

Various strategies for numerical solutions solving FSI-problems are described by [2]:

A) Monolithic method (fully implicit)

Simultaneous solution for all unknowns of the coupled overall system: all interaction effects between the dependent equations are covered

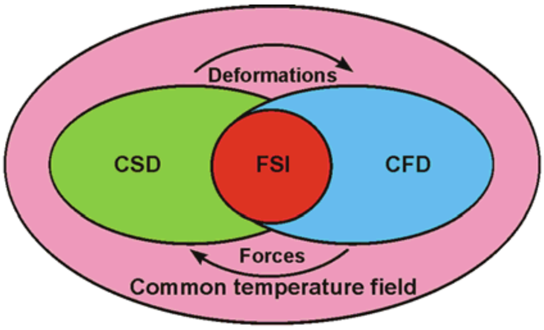
B) Partitioned methods (explicit, implicit/explicit)

Separate solution for the single physical fields:

consideration of interaction effects by exchange of variables at the common interface (surface or volume coupling)

C) Field elimination method

Elimination of field variables at the level of differential equations: limited to elementary linear problems



CFD: Computational Fluid Dynamics: calculates the flow features
CSD: Computational Structure Dynamics: behavior of structures
FSI : Fluid/Structure Interaction

Figure 1: Numerical disciplines used for FSI solutions

Depending on the physical nature of the interaction different coupling methods for the involved physical fields are required:

1) Explicit coupling

- alternating solution of solid and fluid problems with simple interchange of boundary conditions
- flexible, concerning the choice of the solvers for the individual fluid and solid sub-tasks (software modularity)
- often poor convergence behavior

2) Implicit coupling

- solves the equations simultaneously
- good convergence behavior, concerning the coupling process
- voluminous equations, hard to solve

3) Intermediate strategy

- combining the advantages of 1) and 2) for example by:
- predictor-corrector iteration technique
- combining a monolithic solver with a partitioned scheme

At least three solution strategies for FSI-problems are proposed in the available literature [3 - 9]:

a) Weak coupled method (one-way)

Information interchange between sub-tasks is provided only once per time step. No iteration for overall solution within time step is available – no time exactness of solution available. This strategy is appropriate only for little interaction.

b) Strong coupled method (two-way)

Iteration for overall solution within each time step gives a time accurate solution. This method is applicable for strong interaction. Under-relaxation improves convergence behavior.

c) Simultaneous solution

The sub-fields are solved within only one iteration using a consistent discretization in space and time. The method provides a time exact coupling.

2 Application of FSI-methods to a certain technical problem

Non-clogging sewage water pumps normally are equipped with impellers featuring only one blade. The disadvantage which occurs during operation of such pumps is an unequal pressure field at the perimeter of the volute. The time-variant flow in such pumps provides periodic unsteady flow forces which affect the impeller and which produce strong radial deflections of it [10 - 12]. The periodic bending of the pump shaft aligned with the deflections arouses extensive alternating load to the shaft material and often leads to a failure. So it should be a valuable issue to predict the deflections during the pump design.

The numerical effort for different FSI-methods is shown in figure 2 in a schematic view. The effort is increasing from model a) to c). The need for information interchange between the flow and the structural solution increases with increasing strength of the coupling mechanism. In cases, where the changes of the structure have a more or less strong impact on the flow which is responsible for the structural uncertainties, a one-way coupling can not produce the right answer. These cases very often can be found in internal flow systems.

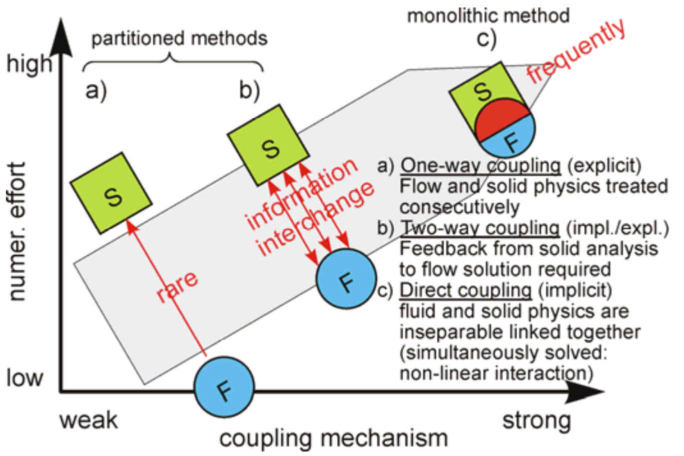


Figure 2: Numerical effort for different FSI-models

Taking the numerical effort into account, the best choice for a complex application like determining the orbit curves of a single-blade pump seems to be a one-way or a two way-coupling method. These solution strategies provide an explicit coupling of two independent software codes, one for the fluid and the other for the structure. The numerical methods typically used for both the flow calculation and the structural analysis are well established and reliable. So the solution of the complex problem can benefit from the advanced features of the single methods. Commercial available solvers for the Navier-Stokes equations can be used for the simulation of the 3-dimensional transient flow field inside the pump and for the structural analysis of the pump rotor several commercial FEM-software codes are available. Both calculations can be coupled by interchanging parameters resulting from the individual software codes.

3 Computing impeller orbits using a one-way coupling

Assuming that the impact of the impeller deflection on the flow in the pump is very small, the oscillations of a single-blade impeller during pump operation were determined by a one-way coupling of the flow and the structure physics. In figure 3 a scheme of the simulation model in which the fluid and the solid problem are solved consecutively is shown. From the results of the numerical simulation of the transient flow field in the pump the hydrodynamic forces were calculated. These forces were used as the load acting at the impeller during the investigation of the pump rotor dynamics [13, 14].

Fluid
Numerical simulation of transient flow in the pump:
TASCflow 2.12

Calculation of hydrodynamic forces

Solid
Analysis of structural dynamics of pump rotor:
ProMecanica
Result: impeller deflections

Figure 3: Scheme of the two-way coupling model

The calculated hydrodynamic forces for three operating points of the sewage water pump are shown in figure 4 for one impeller revolution in the absolute frame of reference. The hydrodynamic forces increase with increasing flow capacity. Using these transient forces as the load acting at half the blade height of the pump impeller, a calculation of the structure dynamics of the rotor with the commercial software tool ProMechanica has been accomplished.

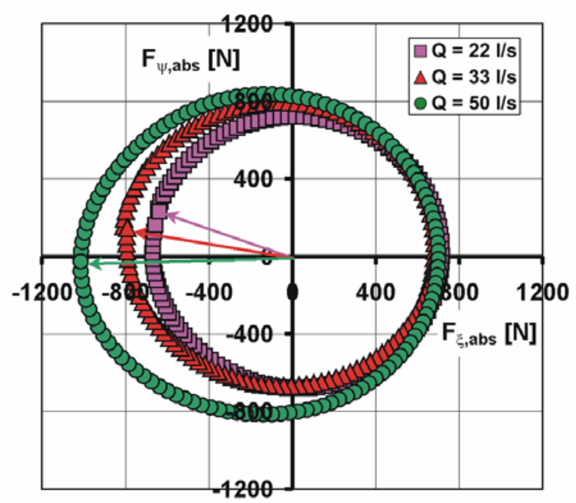


Figure 4: Hydrodynamic forces from num. simulation

In figure 5 the volume model of the pump rotor including the bearings which was used for a structural analysis with ProMechanica is depicted. Furthermore the most important geometrical data are given in the figure.

The deflections at the outer side of the suction mouth of the impeller (see the red bullet in figure 5) attained with this approach are shown in figure 6 in the absolute coordinate system for three investigated operating points

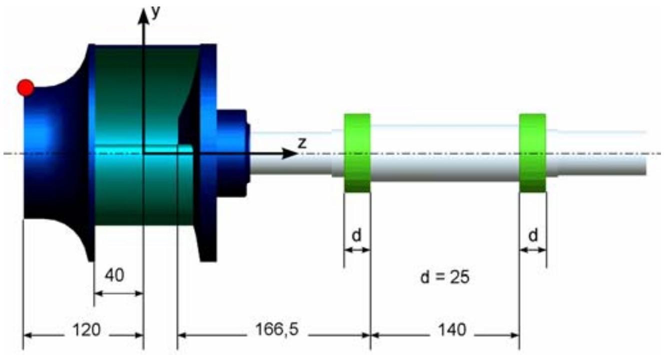


Figure 5: Scheme of the investigated pump rotor

of the pump. The shapes of the impeller orbit curves follow the shapes of the force orbits, because the bending of the pump shaft is in the elastic regime of the material. The impeller deflections increase with increasing flow capacity of the pump. For the smallest investigated flow rate the calculated impeller orbit is nearly circular. For zero deflection in the ψ-direction the amounts of the positive ξ-deflections are nearly the same for all flow rates. The origin of the impeller orbit curve is shifted for the overload operating point in the direction of the positive ξ-axis. So the deflections in the negative ξ-axis are much stronger than in the positive direction. For the ψ-axis a nearly symmetric behavior can be obtained.

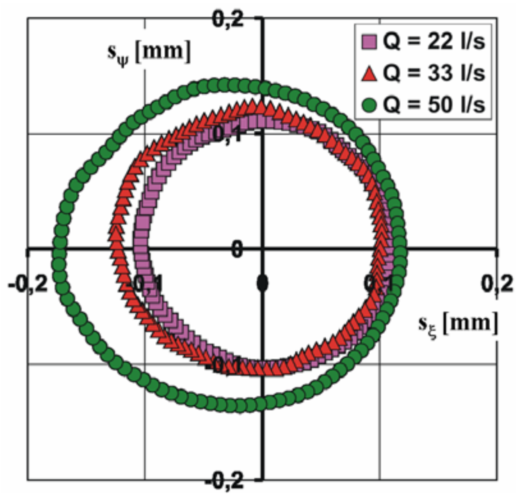


Figure 6: Calculated impeller orbit curves (one-way)

4 Computing impeller orbits using a two-way coupling

When the flow induced deformation of the structure has an impact on the flow behavior in a manner which is not negligible, a two-way coupling of the flow (CFD) and of the solid calculation (CSD) is required to capture the complete phenomenon. Applied to the single-blade pump under investigation here, this approach has to take the impact of the impeller deflections on the flow field into consideration. In figure 7 a scheme of this simulation model is depicted in which the problems for the flow and for the solid are solved alternately.

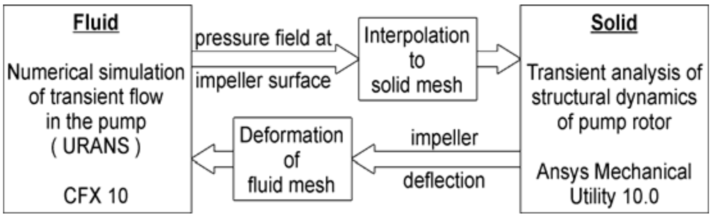


Figure 7: Scheme of the two-way coupling model

The transient flow calculation for the single-blade sewage water pump has been accomplished with the Navier-Stokes solver CFX10.0 using a multi-bloc structured hexahedral grid. First of all the flow in the pump has been simulated with a rigid rotor. No impeller deflections have been taken into account for 10 complete impeller revolutions in order to achieve a periodic flow field. The time step for the simulation of the flow field was equivalent to an impeller rotation angle of $\Delta\varphi = 3$ deg. After the 10^{th} impeller revolution the flow calculation was coupled to the structural analysis of the pump rotor which was carried out with Ansys Mechanical Utility 10.0 using an unstructured tetrahedral mesh for the pump rotor. The information transfer between both numerical schemes has been arranged as follows: For every time step the pressure field at the surface of the impeller was transferred from the flow solver to the FEM solver. The pressure field which was known from the flow simulation acted as a physical load on the fluid wetted impeller surface (hexahedral surface grid of impeller) and was converted to forces using the tetrahedral surface mesh of the impeller volume model. This procedure implies an extensive interpolation. Then, in a transient analysis of the structural dynamics the resulting rotor deflection for the actual force loading has been evaluated for the actual impeller position. As the calculated impeller deflection is a function of the physical load it strongly follows the direction and the magnitude of the amplifying hydrodynamic forces. The original nodes of the flow mesh were corrected by displacing them using the calculated surface displacement of the impeller in a certain procedure.

For the flow mesh which has been corrected for the impeller displacement a new flow simulation has been accomplished. The maximum impeller deflections were only some tenth of a millimeter. So, in a global view the new flow field did not vary very much from the preceding flow field and the new predicted hydrodynamic forces were nearly the same than before. The impeller deflection computed in a second loop was nearly the same than predicted in the first loop. Because of this reason the coupling procedure has been stopped after only one stagger iteration.

The convergence history for the amount of deflection at the outer diameter of the suction mouth of the impeller is plotted in figure 8 for the first 3 impeller revolutions after coupling both simulations for an overload operating point of the pump (Q = 50 l/s). When starting the coupled simulation (impeller turning angle $\varphi = 0$ deg) the initial deflection of the impeller is zero. During the first 25 time steps an increase of the deflection including a strong over-deflection can be recognized. After about one impeller revolution (120 time steps) the calculated impeller deflections became nearly periodic and were repeated every impeller revolution.

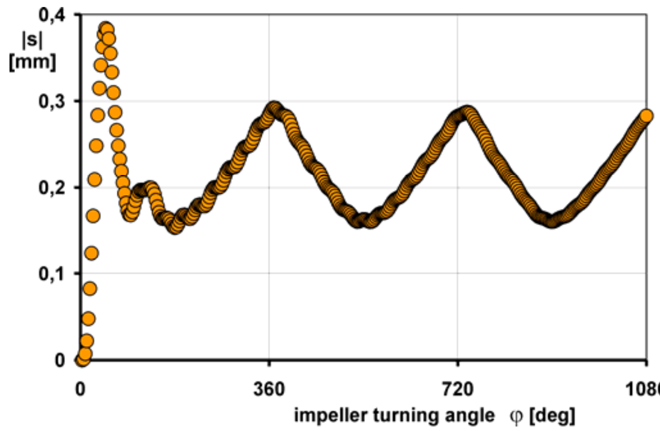
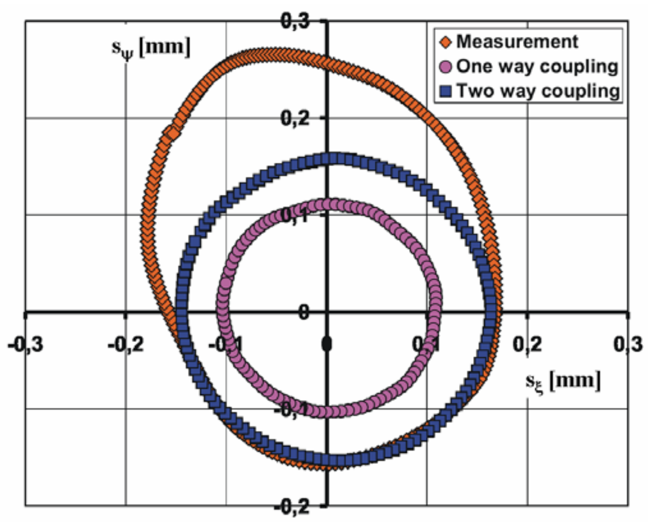
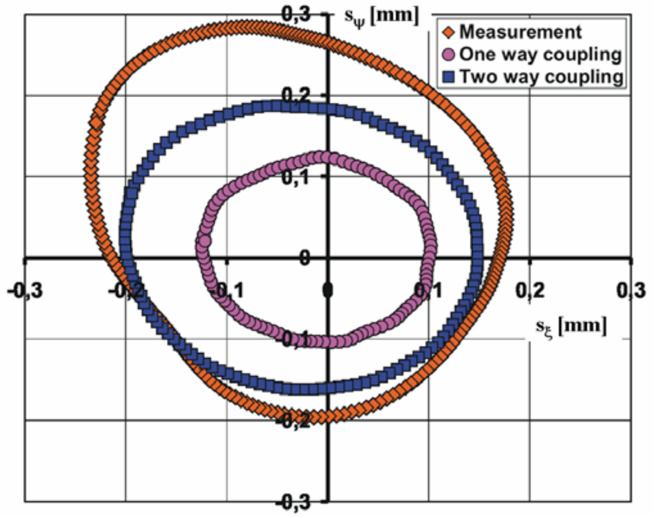


Figure 8: Convergence history of a coupled simulation

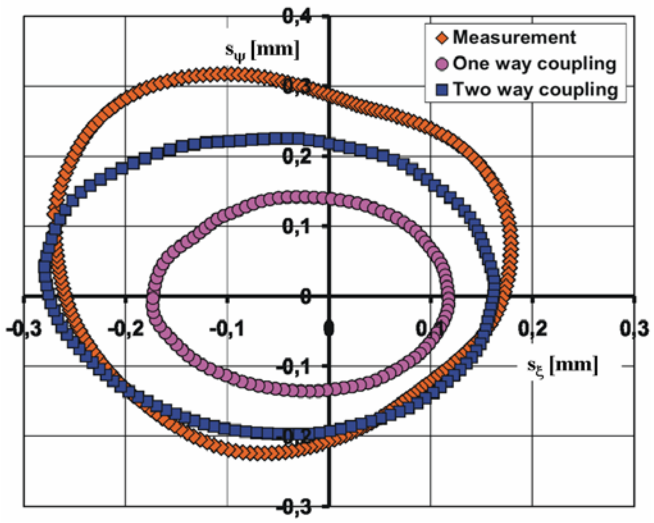
The third impeller revolution after coupling the CFD and the CSD simulation has been regarded worth to be evaluated. A comparison of the numerical obtained impeller deflections to the measurements is shown in figures 9a to 9c for three operating points of the pump (part load, design, overload). All results show the impeller deflections at the outermost point of the suction mouth (see the red bullet in figure 5) as orbit curve presentations. The presented results in figure 9a depict the comparison of measured and calculated orbit curves for a part load operation point (Q = 22 l/s) of the pump. The orbit curve which has been calculated by a simple one-way coupling has a nearly circular shape. There is nearly no accordance with the measurements which show a much stronger impeller deflection for the complete impeller revolution. The agreement for the results of the two-way FSI simulation to the measurements is perfect for about one half of the impeller revolution. For the other half impeller revolution the impeller deflection is strongly underestimated.



a)



b)



c)

Figure 9: Comparison of impeller orbit curves obtained by measurement and by FSI simulation:
a) part load operation: Q/Q_{des} = 67 %
b) design point operation: Q/Q_{des} = 100 %
c) overload operation: Q/Q_{des} = 150 %

In figure 9b the impeller orbit curves are presented for the design operating point of the pump (Q = 33 l/s). Both calculated impeller orbit curves are slightly elliptic. The impeller deflections calculated with the one-way approach are only slightly stronger than the calculated deflections for the part load operation point. So again a strong discrepancy between these results and the measurements can be obtained, because the measured deflections increased with increasing flow rate. The agreement between the two-way simulation and the measurement again is very good for about one half of the impeller revolution. For the other half impeller revolution again a strong underestimation of the deflections can be asserted.

In figure 9c the pikes of the deflection vectors are shown as orbit curves for an overload operating point of the pump (Q = 50 l/s). The same behavior as discussed before can be stated for this duty point. Even though the one-way coupling method shows slightly increased impeller deflections compared to the measurement again a strong underestimation of the impeller deflections for the complete impeller revolution must be stated for this operating point. As seen before for the other operating points a good agreement between the measurement and the two-way simulation is available for about half an impeller revolution. For the rest of the impeller revolution an underestimation of the impeller deflections has been achieved also for the overload operating point.

The discrepancies between measurement and simulation begin to emerge when the trailing edge of the blade approaches the discharge socket of the pump. The reason may be a backlash of the pressure in the pipe to the impeller. The high pressure in the pipe comes back to the pump casing and acts at the blade suction side of the impeller which is now opposite to the discharge socket. This leads to strong deflections in the ψ-direction which could not be obtained by the numerical simulation because of the steady boundary condition at the pump outlet.

5 Conclusions

The present paper is addressed to the evaluation of flow induced deflections of centrifugal pump impellers comprising a single blade. Following a FSI solution strategy with a one-way coupling the hydrodynamic forces and the impeller deflections have been obtained from single numerical solutions for the flow field (CFD) and for the structural dynamics (CSD). The simple one-way coupling method underestimates the measured impeller deflections for the complete impeller revolution in a tremendous manner. This may have different reasons:

1. The hydrodynamic forces were calculated from the transient flow field and then applied as a load on the impeller at half the blade height. This lever arm has been chosen somewhat arbitrarily. The assumption that the

forces act at half the blade height makes the loading for the structural analysis in some way inaccurate.

2. The rotation of the rotor has not been considered during the dynamic structural analysis. So the gyroscopic forces were not included in this investigation.
3. The bending moments produced by asymmetric axial forces have not been taken into account during the calculation.

Beside the above noted reasons another inducement for the strong deviations is imaginable: The measured rotor deflections are in the range of about ± 0.3 mm depending on the operating conditions. Such substantial deflections will have reactions on the flow which have not been considered in the one-way coupling simulation. As the impeller deflections resulting from the transient pressure field in turn affect the flow in the pump, the numerical effort which is required to predict the impeller orbit curves must be higher. The two-way coupled solution which takes the backlash of the impeller deflections to the flow into account gives excellent results only for about one half of an impeller revolution. For the other half of the impeller revolution an underestimation of the impeller deflections has been obtained for all investigated operating points. The reason for this disagreement is unexplainable at this time and needs some further investigations.

Nomenclature:

Arabic and Greek letters

F	N	force
p	bar	static pressure
Q	l/s	volume flow rate
s	mm	deflection
x, y, z	m	Cartesian coordinates in relative frame
Δ	-	difference
φ	deg	angle of impeller position
ω	s^{-1}	angular velocity of impeller
ξ, ψ, ζ	m	Cartesian coordinates in absolute frame

Subscripts

des	design

Abbreviations

3D	**3**-**D**imensional
CFD	**C**omputational **F**luid **D**ynamics
CSD	**C**omputational **S**tructural **D**ynamics
F	Fluid
FEM	**F**inite **E**lement **M**ethod
FSI	**F**luid-**S**tructure **I**nteraction
S	Solid
URANS	**U**nsteady **R**eynolds **A**veraged **N**avier-**S**tokes equations

References:

[1] O. C. Zienkiewicz; R. L. Taylor: *The Finite Element Method*, 5th Ed., Vol. 1, Butterworth-Heinemann, Oxford (2000)

[2] Schäfer, M.; Sieber, G.; Sieber, R.; Teschauer, I.: *Coupled Fluid-Solid Problems: Examples and Reliable Numerical Simulation*, W. A. Wall, K.-U. Bletzinger and K. Schweizerhof (eds.), *Trends in Computational Structural Mechanics*, CIMNE, Barcelona, (2001)

[3] Felippa, C., Park, K., de Runtz, J.: *Stabilization of staggered solution procedures for fluid-structure interaction analysis*, T. Belytschko and T.L. Geers (Eds.), Computational Methods for Fluid-Structure Interaction Problems, AMD Vol. 26, American Society of Mechanical Engineers, New York, pp. 95-124, (1977)

[4] Felippa, C., Park, K.: *Staggered transient analysis procedures for coupled mechanical systems: Formulation*, Computer Methods in Applied Mechanics and Engineering **24**, pp. 61-111, (1980)

[5] S. Piperno, C. Farhat, and B. Larrouturou: *Partitioned procedures for the transient solution of coupled aeroelastic problems*, *Part I: Model problem, theory and two-dimensional application*, Comp. Meth. Appl. Mech. Engrg., 124, pp. 79-112, (1995)

[6] S. Piperno: *Explicit/implicit fluid/structure staggered procedures with a structural predictor and fluid subcycling for 2D inviscid aeroelastic simulations*. Internat, Journal of Numerical Methods in Fluids, 25(10), pp. 1207-1226, (1997)

[7] D. P. Mok, W. A.Wall, and E. Ramm: *Accelerated iterative substructuring schemes for instationary fluid-structure interaction*. K.J. Bathe, editor, Computational Fluid and Solid Mechanics, pp. 1325-1328, Elsevier, (2001)

[8] F. Nobile: *Numerical approximation of fluid-structure interaction problems with application to haemodynamics*, PhD thesis, EPFL, Switzerland, (2001).

[9] D. Peric and W. G. Dettmer: *A computational strategy for interaction of fluid flow with spatial structures*, 5th international Conference on Computational of Shell & Spatial Structures, IASS-IACM, (2005)

[10] Okamura, T.: *Radial Thrust in Centrifugal Pumps with a Single-Vane-Impeller*, Bulletin of the JSME 23, No. 180, (1980)

[11] Aoki, M.: *Instantanous Interblade Pressure Distributions and Fluctuating Radial Thrust in a Single-Blade Centrifugal Pump*, Bulletin of the JSME 27, No. 233, (1984)

[12] Agostinelli, A.; Nobles, D.; Mockridge, C. R.: *An Experimental Investigation of Radial Thrust in Centrifugal Pumps*, Transactions of the ASME, Journal of Engineering for Power, Vol. 82, No. 2, (1960)

[13] Benra, F.-K.: *Numerical and Experimental Investigation on the Flow Induced Oscillations of a Single-Blade Pump Impeller,* ASME Journal of Fluids Engineering, Vol. 128, pp. 783–793, (2006)

[14] Benra, F.-K.; Dohmen, H. J.: *Comparison of Pump Impeller Orbit Curves Obtained by Measurement and FSI Simulation*, Proceedings of ASME Pressure Vessels and Piping Division Conference 2007, Paper PVP2007-26149, San Antonio, Texas, USA, (2007)