MESHLESS INTERPOLATION APPLIED TO FLUID-STRUCTURE INTERACTION

Rafael de Mello Pereira
Universidade de Brasília-Departamento de Engenharia Mecânica – Laboratório de Energia e Ambiente –LEA
CEP 70.910-900 – Brasília –DF – BRASIL
+55(61) 307-2314 R236/238
rpereira_01@bol.com.br

Antônio C. P. Brasil Júnior
Universidade de Brasília-Departamento de Engenharia Mecânica – Laboratório de Energia e Ambiente –LEA
CEP 70.910-900 – Brasília –DF – BRASIL
+55(61) 307-2314 R236/238
brasil@enm.unb.br

Abstract. *The present work provides a multidimensioanl interpolation method that will be applied in the interface fliud-structure for hydroturnines blades. The methodology proposed in this work firstly computes the fluid flow by unstructured finite volume method providing the pressure field in the boundary of the solid problem. This field is interpolated to a finite element mesh to determine the load applied and finally the structural problem is solved. The interpalation procedure should work with two meshes with different refinement levels and avoid the use of a gridiron layout, to attend this requisites a meshless interpolation approach will be adopted.*

Keywords. *Interpolation, Meshless, Hydroturbines, Fluid-Structure Interaction*

1. Introduction

Hydro machinery industry has used in the last decades more intensely numerical analysis for the development of even more efficient turbines (Wichström, 1997; Carstens, 2002). This industry saw the application of finite volume method as the main tool for the flow simulation and finite elements for structural analysis, each one of these problems has its proper peculiarities. Turbulence models based on the Reynold mean equation like κ-ε and κ-ω are commonly applied in the study of the behavior of the fluid and fatigue model used for fail criteria for the structural analysis. Both the problems when applied for realistic cases with complex geometries are very computational expensive. With the constant and fast increase in the computer capacity more refined model are proposed, at this time hundreds of thousands or even millions of degrees of freedom problems are normally solved.

A new problem begins to be explored last years with the consideration of the interaction between the fluid and the structure (Crolet, 1992). The level of sophistication of this interaction can vary, in the most expensive case only one problem is solved with the fluid and structure domains together. This is called the coupled case and since the fluid and the structural models differ in their formulation and discretization, an interface model has to be introduced that represents the connectivity and physical interaction. Normally this interface model makes possible integration in time of the governing equations (Beckert, 1999).

A cheaper and simple technique adopted is solve the fluid and structural problem separately and use the solution of one problem as the boundary condition for the other problem. The load applied in the structural problem is interpolated from the pressure field calculated by the CFD code and using an iterative process the mesh used for the flow simulation can be updated by the displacements calculated in the solid analysis.

For turbo machinery structural analysis fluid-structure interaction techniques are essential because the boundary condition for the solid problem comes from a very complex pressure field and the surfaces of the blades are irregulars. This problem can be found also in aeronautical application for airplane wings. This paper aims to use an interpolation process to solve a uncoupled fluid structure interaction. The full uncoupled process can by visualized in Fig. (1).

The interpolation main task is determined the load applied in the solid mesh with the CFD nodes solution. For some situations this problem is contouring adjusting a function over the geometry with the boundary information needed, the difficult with this boarding arise from the restriction of the data fitting approximation and the laborious process involved. For a most automatic and robust process a mesh to mesh interpolation is desired. The interpolation process can be reached in many ways according to the mesh information available, the simplest case occurs when the connectivity information is know and the value of a node inside an element can be evaluated using conventional shape functions (Löhner, 2001). For the worse case where the connectivity matrix is not available and the only information are the nodes coordinates and the values to be interpolated a meshless approach is useful. A good interpolation algorithm must also deal with meshes with different refinement and local concentration of nodes, what is very common in fluid-structure interaction. An alternative surface meshless approach based of boundary elements method can be found in (Ochiai, 2001).

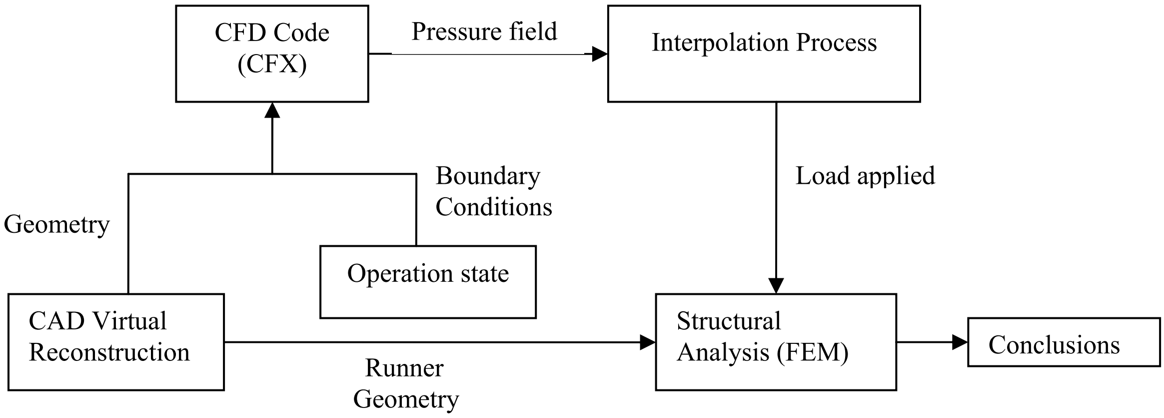


Figure 1. Uncoupled fluid-structure interaction project sequence.

2. Theory

2.1. Turbulence Formulation

For incompressible turbulent flows, the conservation of mass, and momentum can be expressed by the classical Reynolds averaged equations given by:

$$\nabla \cdot \mathbf{U} = 0 \tag{1}$$

$$\frac{\partial \mathbf{U}}{\partial t} + (\nabla \mathbf{U})\mathbf{U} = -\frac{1}{\rho}\nabla P + \nabla \cdot \left(\nu \nabla \mathbf{U} - \overline{\mathbf{u} \otimes \mathbf{u}}\right) \tag{2}$$

In those equations **U**, P are the mean velocity and pressure fields; ρ , ν are the density and kinematics viscosity of the fluid; and $\overline{\mathbf{u} \otimes \mathbf{u}}$ is the Reynolds stress tensor modeled by the Boussinesq eddy viscosity assumption:

$$\overline{\mathbf{u} \otimes \mathbf{u}} = \frac{2}{3}k\mathbf{I} - 2\nu_T \mathbf{D}(\mathbf{U}) \tag{3}$$

where ν_T is the turbulent eddy viscosity, $\mathbf{D}(\mathbf{U})$ is the mean rate-of-strain tensor, $\mathbf{I}$ is the identity tensor and κ the kinetic energy of turbulence.

The turbulent eddy viscosity is modeled by the Prandtl-Kolmogorov relation written as:

$$\nu_T = C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

where ε is the dissipation rate of kinetic energy. It is the basis of the classical $\kappa - \varepsilon$ model of turbulence. This model requires the use of two additional transport equations for κ and ε given by:

$$\frac{\partial k}{\partial \varepsilon} + \mathbf{U} \cdot \nabla k = \nabla \cdot \left[\left(\nu + \frac{\nu_T}{\sigma_k}\right)\nabla k\right] + P_k - \varepsilon \tag{5}$$

$$\frac{\partial \varepsilon}{\partial t} + \mathbf{U} \cdot \nabla \varepsilon = \nabla \cdot \left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon}\right)\nabla \varepsilon\right] + C_{\varepsilon 1} P_k \frac{\varepsilon}{k} - C_{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{6}$$

In those equations P_k is the production of k which is written as:

$$P_k = 2\nu_T\left[\mathbf{D}(\mathbf{U}):\mathbf{D}(\mathbf{U})\right] \tag{7}$$

The standard values of the constants are:

$$C_\mu = 0.09\,; C_{\varepsilon 1} = 1.44\,; C_{\varepsilon 2} = 1.92\,; \sigma_k = 1.0\,; \sigma_\varepsilon = 1.3$$

2.2. Solid Formulation

Turbines runner are normaly subjected to load fluctuation, the operational regime due diferent power generation requirements is the main source of this time fluctuation. Then to realistically predict structural integrity of turbine componets is necesary the application of fatigue models. These models could be very sofisticated and a entiry load period hystory necessary to predict the life of the component subject to fatigue. This work will consider the simplest cases of linear elasticity (Bathe, 1996; Ottosen, 1992), a sequence of linear problems are normaly used to determine stress and strain which are used in fatigue models.

Equation (8) express the *equilibrium condition* (i.e. the balance priciple) for a infinitesimal peace of the body, the stress tensor **S** (9) is symmetric and represent stresses components inside de body, and b is due body forces.

$$\nabla \bullet \mathbf{S} + b = 0 \tag{8}$$

$$\mathbf{S} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \tag{9}$$

For a arbitrari surface inside of over the body with normal **n** (10) represent the stress conponents acting in this surface.

$$\boxed{\begin{aligned} t_x &= \sigma_{xx}n_x + \sigma_{xy}n_y + \sigma_{xz}n_z \\ t_y &= \sigma_{yx}n_x + \sigma_{yy}n_y + \sigma_{yz}n_z \\ t_z &= \sigma_{zx}n_x + \sigma_{zy}n_y + \sigma_{zz}n_z \end{aligned}} \tag{10}$$

For an isotropic and homogenius material with Young's modulus E and Poisson's ratio ν the constitutive relation expressed by *Hooke's law* is given by (11-12).

$$\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon} \tag{11}$$

where

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yyx} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{bmatrix}; \mathbf{D} = \frac{E}{(1+\nu)(1-2\upsilon)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.5(1-2\upsilon) & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5(1-2\upsilon) & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5(1-2\upsilon) \end{bmatrix}; \boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yyx} \\ \varepsilon_{zz} \\ \varepsilon_{xy} \\ \varepsilon_{xz} \\ \varepsilon_{yz} \end{bmatrix} \tag{12}$$

The solid problem boundary conditions comes for a pressure component interpolated from the fluid solution, these pressure component is a scalar value equal to the modulus of the compression vector acting normal to the surface. To determine this vector the pressure component is multiplied by the surface unit normal vector (13).

$$\mathbf{t}_{Boundary} = -p\mathbf{n} \tag{13}$$

2.3. Numerical fluid and solid solver.

A finite volume commercial code (CFX) was used for solution of fluid flow, the implemented $\kappa - \varepsilon$ model was used, and the boundary condition for the turbine problem was taken from index test flux and spiral casing geometry.

For solid solution an in-house developed code using three dimensional linear elasticity was used. Linear tetrahedral elements with four nodes are used and an interactive conjugated gradient solver used.

2.4. Interpolation Procedure.

The interpolation $\overline{f}(x)$ of a function $f(x)$ in a considered domain Ω can be defined by (14) where $p_j(\mathbf{x})$ is an independent vector with n basis functions (Gavete, 2002; Krysl, 2001), usually the polynomials. For example in a three dimensional space different degrees of approximation basis are given by (15)

$$f(x) \approx \overline{f}(x) = \sum_{i=1}^{n} p_j(\mathbf{x}) a_j(\mathbf{x}) \tag{14}$$

$$\boxed{\begin{array}{l} \mathbf{p}(\mathbf{x}) = [1] \\ \mathbf{p}(\mathbf{x}) = [1, x, y, z] \\ \mathbf{p}(\mathbf{x}) = [1, x, y, z, xy, xz, yx, x^2, y^2, z^2] \end{array}} \tag{15}$$

The coefficients $\mathbf{a}(\mathbf{x})$ are determined by minimizing the sum of the square distances of the error at each point with respect to $\mathbf{a}(\mathbf{x})$:

$$J = \sum_{i=1}^{n} w(\mathbf{x} - \mathbf{x}_i)[\mathbf{p}(\mathbf{x}_i)\mathbf{a}(\mathbf{x}) - f_i]^2 \tag{16}$$

If the existence condition $|\mathbf{A}| \neq 0$ is satisfied then:

$$\mathbf{a}(x) = \mathbf{A}^{-1}(\mathbf{x})\mathbf{B}(\mathbf{x})\mathbf{f}, \tag{17}$$

where

$$\mathbf{A}(\mathbf{x}) = \sum_{i=1}^{n} w_i(\mathbf{x})\mathbf{p}^T(\mathbf{x}_i)\mathbf{p}(\mathbf{x}_i) \tag{18}$$

$$\mathbf{B}(\mathbf{x}) = \left[w_1(\mathbf{x})\mathbf{p}^T(\mathbf{x}_1), w_2(\mathbf{x})\mathbf{p}^T(\mathbf{x}_2), \ldots, w_n(\mathbf{x})\mathbf{p}^T(\mathbf{x}_n)\right] \tag{19}$$

$$\mathbf{p}(\mathbf{x}_i) = \left[1, x_i, y_i, z_i, \ldots\right] \qquad (i = 1, 2, \ldots, n) \tag{20}$$

$$f = \left[f_1, f_2, \ldots, f_n\right]^T \tag{21}$$

In the above equations f_i is the nodal independent variable at the node $\mathbf{P}_i(\mathbf{x}_i)$, $w_i(\mathbf{x}) = w_i(\mathbf{x} - \mathbf{x}_i)$ is a weight function, m the number of the basis functions, n the number of nodes in the influence domain $\Omega_l \in \Omega$. The matrix $\mathbf{A}$ and its inverse $\mathbf{A}^{-1}$ are symmetrical. This property can be utilized in the computation to reduce the computing time choosing an apropriete symmetrical solver like cholesk.

The weight function w used has a inportant fuction and must satisfy the following requirements:

- It must be positive, continuous and differentiable in the influence domains.
- It shoould have a relatively larger value for a node which is closer to the point being interpolated in the influence domain.

For PDE numerical solver that use meshless methods weight functions have stabily importace (Du, 1998), for the interpolation process it increases the quality when gives a large weight for nodes close to the interpolation point.

An Example of weight function are show below:

$$w(r)=\begin{cases}1 & (d=0)\\ \left[e^{-(d/c)^{2k}}-e^{-(dm/c)^{2k}}\right]\Big/\left[1-e^{-(dm/c)^{2k}}\right] & (0<d\le d_m)\\ 0 & (d>d_m)\end{cases} \tag{22}$$

where d_m denotes the size of the support for the weight function, c is a positive parameter which control the relative weights. When c decreases, the nodes closer to the evaluation point have higher weights than those of far nodes. K is a constant ≥ 0.5 (Du, 1998).

An spherical influence domain are choose classically, but for boundary values interpolation it could induce some problems, the main happen when try to interpolate values for two surfaces very close that have very diferent pressure value. Unfortunately this case is standart for turbines blades and airfoil see Fig. (2). Fig (3) show a semi-sphere domain and C_p profile over an airfoil.

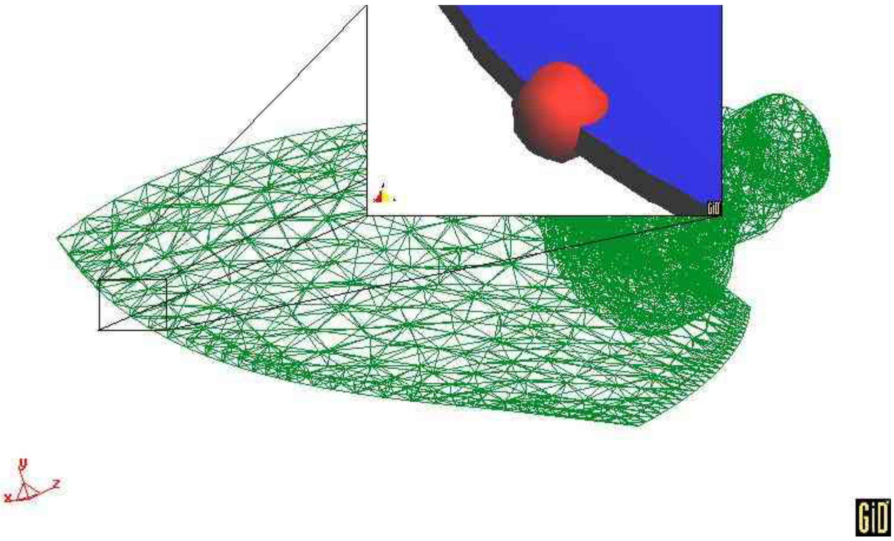
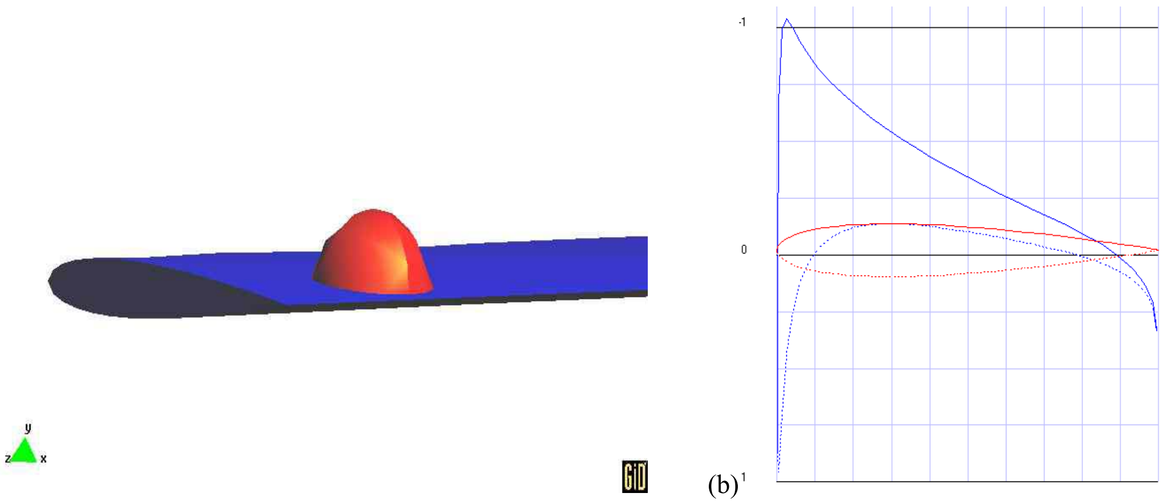


Figure 2. Sphere interpolation domain crosing thin trubine blade.

A simple way to avoid this proble is choose a semi sphere influence domain that will take count only the points in the righ side of the blade fig. 3.



(a) (b)[1]

Figure 3. Semi-sphere interpolation domain over airfoil (a) and Cp distribution for this airfoil (b).

3. Numerical results.

3.1. Wedge numerical example.

In the first example a pure numerical case is explored, an imposed function is attributed to a solid mesh nodes with a wedge on its base Fig. (4a), than a secondary surface mesh that represents the solid boundary Fig. (4b) is interpolated using the values imposed for the first mesh. Finally the values find are compared with the analytical function.

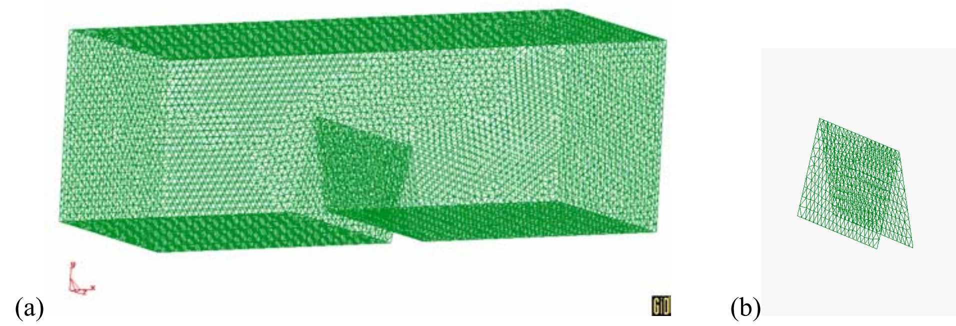


Figure 4. Solid mesh representing flow problem and surface mesh representing structural boundary conditions.

Both meshes have different grade of refinement and the nodes in the surface are not coincident, the solid mesh height and depth is 1 and the width is 3 the wedge has a 78.7° angle. A sine trigonometrical function was used because this function can't be perfectly approximated by a polynomial function.

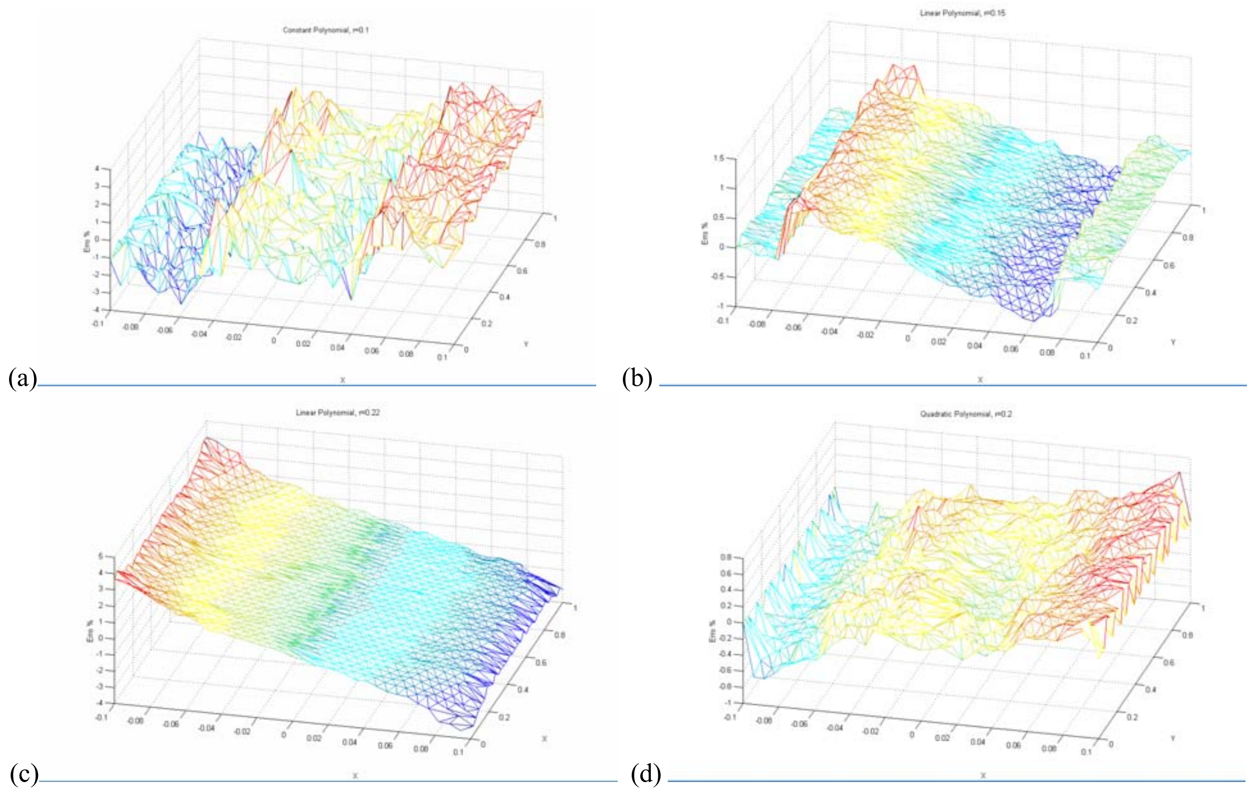


Figure 5. Percentual error associated for four different polynomial approximation grades.

The four figures above show a considerable reduction in the error ratio when the polinomial degree is increased, except that Fig. (5b) and Fig. (5c) although have the same polynomial degree have diferent errors, this is because the interpolation domain Ω is bigger inducing the consideration of points too far from the desired point. The choose of r has a great inportace, if r is choose small the condition of existence $N\_points >= Poly\_degree$ can't be satisfacted, if r is choose biger the influence domain could considerate points too far from the desired point. This problem is even worse if the mesh has diferent refiniment grades. For this reason a diferent approach will be used in the next two examples, althought use a fixed radius a fixed number of neighborhood point is used.

3.2. Airfoil numerical example.

The next example explores a more realistic case, an airfoil subjected to frontal water flow. Figure 6a shows the geometry of the airfoil, the tunnel has 3 meter height and 8 meter width; the airfoil has a 1m cord length. For the fluid domain a 3 m/s and relative 0 Pa boundary conditions are admitted. The solid proprieties are 2.1E9 Mpa Young Modulus and 0.3 Poisson's ratio. The airfoil is constrained in the both sides.

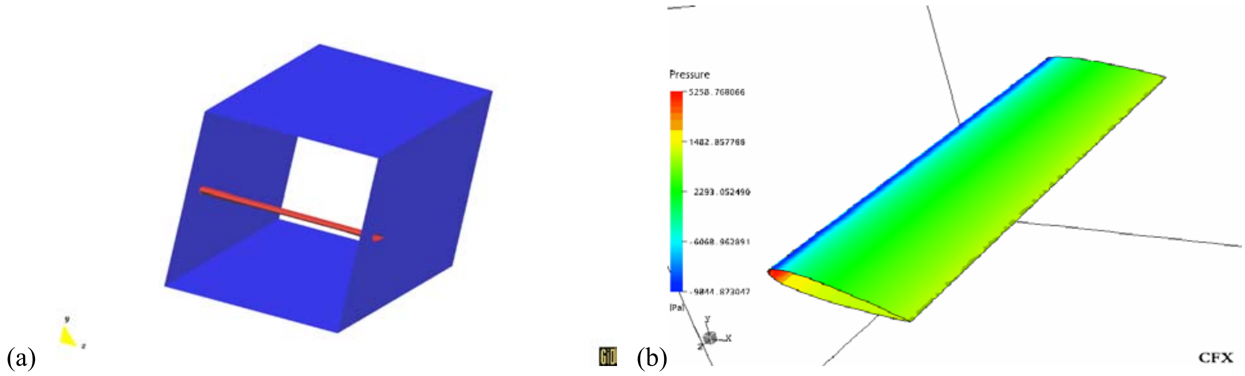


Figure 6. Geometry and pressure distribution over airfoil.

The fluid problem was simulated using the commercial code CFX using a 120000 nodes tetrahedral mesh with $\kappa - \varepsilon$ model. Fig. (6b) shows the pressure field found. The solid problem was solved with a 30000 nodes mesh and linear elasticity model. Fig. (7b-7d) shows the structural analysis results for displacement and critical stress. Interpolation was done with a fixed number of nodes in the interpolation domain because the meshes used have local refinements. A linear polynomial was used with 15 neighbor nodes in the influence domain.

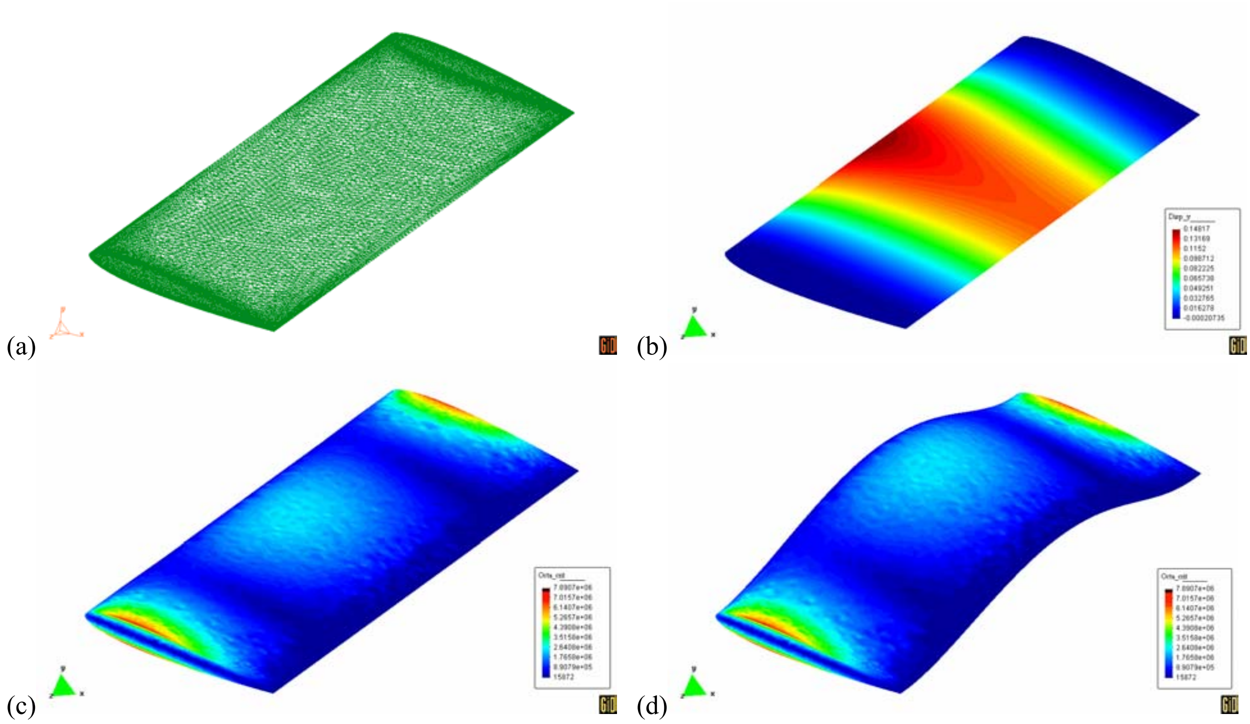


Figure 7. Mesh used, displacements and critical stresses over the airfoil.

To evaluate the interpolation results the three force components are calculated by integration of the pressure over the faces and compared with the results forces given by CFX, Tab (1).

Table 1. Comparison between total forces interpolated and calculated by CFX.

	F_X component	F_y component	F_z component
CFX	1450.25 N	10975.7 N	-0.25 N
Interpolated	1413.31 N	10985.7 N	21.56 N
Error	< 1%	< 1%	-

The results show a good agreement between the interpolation and CFX code. The F_z component should be zero because the airfoil is symmetric so the value found is numerical garbage and can't be compared.

3.3. Kaplan Turbine Blade Example.

This final example compares the quality of interpolation for a most realistically case. A complete turbine blade was simulated using CFX, Fig. (8) shows the values of pressure over the blade and Tab (2) shows the compared results for the total sustentation force.

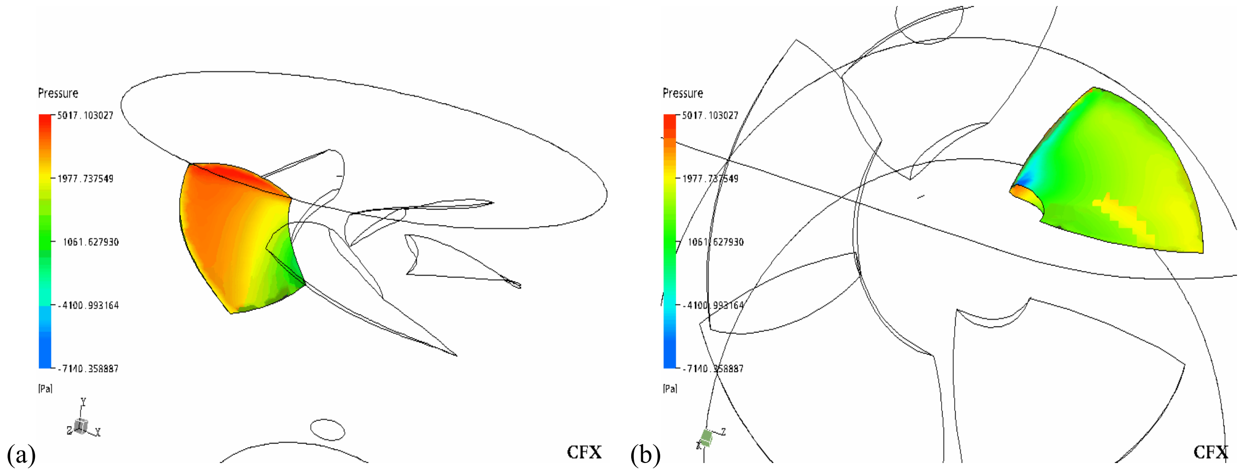


Figure 8. Simulated pressure distribution for a turbine blade.

Table 2. Comparison for different polynomial degree, nodes neighbors and interpolation domain.

Polynomial degree	Nodes Number	Half sphere	F_Y[N]	Error %
CFX	-	-	-5345.73	-
0	1	Y	-5342.39	< 1%
0	3	Y	-5319.66	< 1%
0	1	N	-3781.45	29%
1	10	Y	-5319.22	< 1%
1	20	Y	-5288.19	< 2%
1	5	Y	-14520.1	171.6%
1	4	Y	Overflow	-
1	10	N	-2418.55	54%
2	20	Y	-5345.29	< 1%

Table (2) shows the limitation for the interpolation process, for the geometries with thin parts like blades problems the half sphere interpolation domain is essential, and when the number of interpolation points is close to the number of polynomial terms the results can be very bad or the interpolation is impossible.

The mesh used for the solid simulation and the critical stress is show in Fig (9).

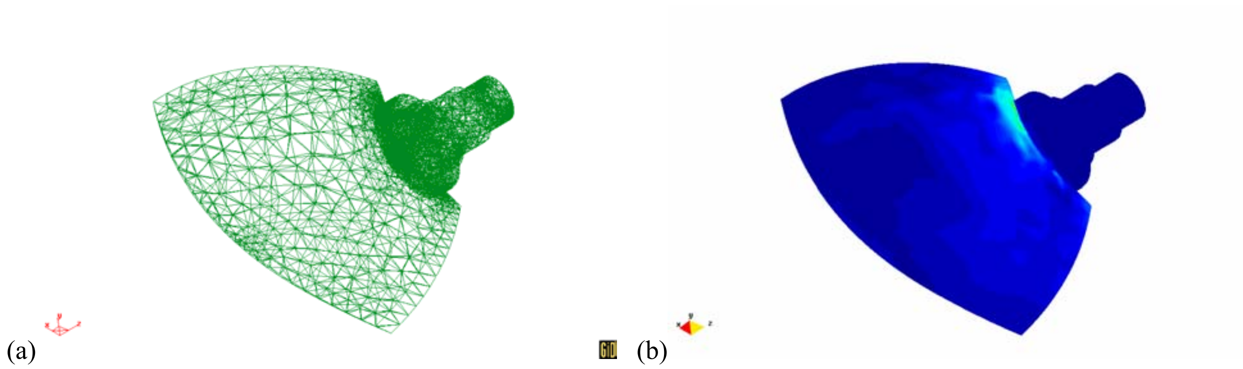


Figure 9. Solid mesh and critical stresses for the blade simulation.

4. Comclusion

The problem of interpolation from a tree dimensional mesh for a surface boundary mesh can be satisfactory and flexible solved using meshless methods, some attention need be give for the interpolation domain when applied for a thin body but this problem is perfectly turned around by using a semi sphere domain. The other problem observed is the radius choose when the mesh have different refinement level, this problem are turned around using a fixed number of interpolation points. A flexible algorithm for interpolation has great value for fluid-structure application and mesh less approach can handle real world problems.

5. Acknowledgments

The authors gratefully acknowledge the financial support provided, respectively, by the agency CNPq and by the agency ELETRONORTE S/A.

6. References

Bathe Klaus-Jürgen, 1996, "Finite Element Procedures", Prentice-Hall , Upper Saddle River, New Jersey

Beckert A., 2000, "Coupling fluid (CFD) and structural (FE) models using finite interpolation elements", Aerospace Science Technologi, 4 (2000) pp. 13-22

Carstens Volker, Kemme Ralf, Schmit Stefan, 2003, "Coupled simulation of flow-structure interaction in turbomachinery", Aerospace Science and Technology 02/2003, v 1.131 article in press.

Crolet J. M., Ohayon R., 1992, "Computational methods for fluid-structure interaction", Longman Scientific & Technical, Harlow, Essex, UK

Du Chongjiang , 2000, "An element-free Galerkin method for simulaation of stationary two-dimensional shallow water flows in rivers", Comput. Methods Appl. Mech. Engrg. 182 (2000) 89-107

Gavete L., Cuesta J.L., Ruiz A., 2002, "A numerical comparison of two different approximations of the error in a meshless method", European Journal of Mechanics A/Solids 21 (2002) 1037-1054.

Krysl Petr, Belytschko Ted, 2001, "ESFLIB: A library to compute the elemnent free Galerkin shape function", Comput. Methods Appl. Mech Engrg. 190 (2001) 2181-2205

Löhner Rainald, 2001, "Applied Computational Fluid Dynamics Techniques", Jown Wiley & Sons, inc. New York, NY USA, pp. 239-260.

Ochiai Yoshiro, 2001, "Multidimensional interpolation and numerical integration by boundary element method", Engineering Analysis with Boundary Elements 25 (2001) 697-704.

Ottosen Niels, Petersson Hans, 1992, "Finite Element Method", Prentice Hall International (UK), Maylands Avenue, Hemel Hempstead Hertford, UK.

Wickström A.,1997, "Structural Analysis of Francis Turbines Runners Using Adina", Computers & Structures Vol. 64, No 5/6, pp 1087-1095, 1997.

Wilcox D. C., 1993, "Turbulence Modeling for CFD", DCW Industries, 5354 Palm Drive, La Cañada, California, USA, pp. 87-91