Report of activities at TU Braunschweig

Gustavo A. Ríos Rodriguez
Assistant Researcher at CONICET

July 30, 2014

1 Introduction

The present report aims to show the research activities that were carried out during the stay of three months at the Institut für Konstruktionstechnik (IK) of the Technische Universität at the city of Braunschweig, from 01/04/14 up to 26/06/14. These activities were developed within the framework of the International Research Staff Exchange Scheme (IRSES), project "NumSim" PIRSES - GA2009_246977 Marie Curie Actions, funded under the 7 framework program of the European Commission. The final goal of this work plan is to use High Performance Computing (HPC) resources to obtain a parallel multiphysics code by combining the advantages of specialized codes such as elPaSo [2] and PETSc-FEM [7], in order to deal with the numerical solution of intrincate Fluid-Structure Interaction (FSI) problems. The activities comprise the colaborative work between Dr.Ing.Marco Schauer from the German Institute with Dr.Ing.Luciano Garelli and Dr.Ing.Gustavo Ríos Rodriguez, both researchers at CIMEC-CONICET.

As it was already mentioned, the final aim of the visit is the development and implementation of a coupling strategy to solve FSI problems using both parallel codes elPaso and PETSc-FEM, which are developed by the TU Braunschweig and CIMEC respectively. The underlying idea is to combine both codes using a partitioned scheme to achieve a parallel FSI solver for problems with a *weak-coupling* between the fluid dynamics and the structure dynamics. The fluid dynamic equations are solved using an Arbitrary Lagrangian Eulerian (ALE) scheme and the mesh movement to adapt the flow domain to the displacement of the structure are computed with PETSc-FEM. Meanwhile, the structural problem is solved by elPaSo code. Also, the adaptive refinement capabilities of PETSc-FEM will try to be exploited in the context of FSI problems.

2 Working stages

The work that was carried out during these months consisted of continuing with the activities that were initiated during the three months visit of Dr.Marco Schauer to the CIMEC in 2013, which were reported in [4]. Indeed, they can be grouped and described as in the following sections.

2.1 Installation of PETSc-FEM software at the Institut für Konstruktionstechnik computer center

The first activity was to install the PETSc-FEM software in the computer center of the IK. This activity was carried out by the system administrator of the computer center together with Dr.Ríos

Rodriguez and took three weeks aproximately, since most of the libraries required by PETSc-FEM were not installed in the server. Finally, PETSc-FEM software was installed and tested in the server account of Dr.Ríos Rodriguez, so that it can be used by the researchers of the German Institute.

2.2 Code development for the information transfer between non matching meshes.

The final goal of the work is the coupling of both codes PETSc-FEM and elPaSo using a partitioned scheme. It was discussed and finally agreed between the researchers of both institutes that some kind of strategy was needed to transfer the solution (the loads or pressures) from the fluid to the structure mesh, but considering that both meshes do not match at their common interface, i.e. that their vertices do not coincide at the interface. This is a primary requirement since the setup of the problem (mainly mesh generation) can be greately simplified and the computing time can be reduced because of the decoupling of the meshes at the interface, which would allow to use different kind of elements and discretizations for the fluid and the structure. Different discretizations is a common situation in FSI problems, since the finite element mesh for the fluid domain usually requires to be much finer than that for the structure one. Also, it was agreed that adaptive refinement capabilities of PETSc-FEM could not be applied to solve the fluid dynamic equations unless such a projection strategy were developed and implemented as a software layer between PETSc-FEM and elPaSo, because of the non-matching meshes at the fluid-structure boundaries generated by modifications introduced to the fluid mesh.

The strategies for information transfer that were developed during the work follow the ideas mentioned in [3] and [6], from which they can be classified as: *i*) direct or geometric interpolation, *ii*) conservative projection and *iii*) monotone conservative projection, ordered these by increasing complexity, computational cost and numerical accuracy.

The direct or geometric interpolation algorithm was implemented as a first step and the conservative algorithm as the second one. Both codes are written in the C++ language so that they can be easily integrated within the PETSc-FEM software and also with the aim of taking advantage of previously developed C++ functions of the adaptive refinement code. By the time of writting this report, both are test codes in the sense that they are thought for testing the projection strategies but without focusing on the performance side. Future revisions will aim to improve this aspect.

3 A brief description of the load projection strategies

The following sections briefly introduce both the direct and the conservative projection strategies that were implemented. A detailed description and analysis of them can be found in [3] and [6].

3.1 Direct interpolation

This is the simplest and less expensive of both strategies, being its main drawback that it is not conservative [6]. This means that from an integral point of view, not all the load from the fluid side is transferred to the structure side. Considering that u is the quantity or variable to be projected, let p be a vertex of the mesh structure with coordinates $\overline{x}_p$. The value of variable u at point p can be computed by direct interpolation with the shape functions of the nodal values corresponding to the vertices of the fluid element which "contains" vertex p. For doing this, it is necessary to

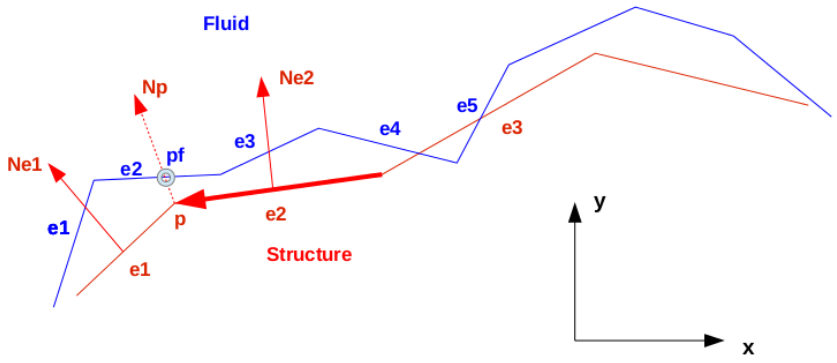

Figure 3.1: Direct projection scheme in 2-D

identify the element on the fluid domain boundary where the projection of vertex p falls within. This problem is efficiently resolved in this work by using the ANN library [1], which allows to solve the Approximate Nearest Neighbour search problem. In this case, a list is built with the nearest fluid elements to vertex p by considering the distance from this vertex to the centroids of the fluid elements. Subsequent iteration is performed on this list in order to effectively determine the fluid element which contains vertex p by solving, in 2-D the intersection problem between the straight line that passes through both vertices of the current iteration element (or the plane that passes through all the vertices of the element for the 3-D case) and the straight line normal to the structure domain boundary at vertex p. If intersection exists, then it has to be checked that it falls within the current fluid element by applying the following condition,

$$\min(N^i(pf), 1 - N^i(pf)) \geq 0, \quad \forall i \tag{3.1}$$

where $N^i(pf)$ is the shape function associated to local i-vertex of the fluid element evaluated at the normal projection of point p, which is denoted as pf (see figure 3.1). The normals to the structure boundary mesh vertices are computed as the average of the normals to the boundary mesh elements that share each vertex. If it is finally found that pf is within the current fluid element, the corresponding shape functions are used to interpolate the variable u,

$$u(p) = \sum_{i}^{nbe} N^i(pf) u_i \tag{3.2}$$

being u_i the nodal values of variable u at the vertices of the fluid element and nbe is the number of vertices per element.

3.2 Conservative projection

The conservative strategy propose that the integral of the variable of interest (e.g. the pressure) along the fluid domain boundary p_f be equal to the integral along the structure domain boundary p_s. Assuming a Finite Element Method (FEM) approximation for pressure the p and a weigthed residual approach on the fluid-structure boundary interface Γ_{fs}, and particularly a Galerkin method for the weighted residual approach, it can be stated that

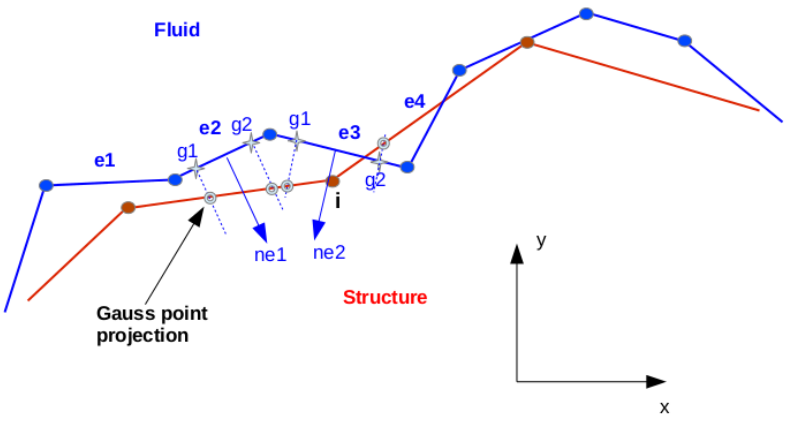

Figure 3.2: Conservative projection scheme in 2-D

$$\int_{\Gamma_{fs}} N_s^i N_s^j \hat{p}_{sj} d\Gamma = \int_{\Gamma_{fs}} N_s^i N_f^j \hat{p}_{fj} d\Gamma \tag{3.3}$$

where N_s^j is the shape function at vertex-j of the structure boundary mesh, N_f^j is the shape function at vertex-j on the fluid boundary, $\hat{p}_{fj}$ is the pressure at vertex-j of the fluid mesh computed with FEM, $\hat{p}_{sj}$ is the pressure at vertex-j of the structure boundary mesh computed by projection (the unknown of the projection problem) and $n_s$is the number of vertices on the structure boundary mesh. On the left-hand side of eq.(3.3) there appears the mass matrix of the structure boundary elements and on the right-hand side there is an integral with the product of shape functions from both the structure and the fluid discretizations. Such integral is approximated in this work by using a Gauss numerical quadrature, introducing the Gauss points on the fluid elements and projecting them by their corresponding normals onto the structure elements [3]. The linear system of equations that is finally solved can be written as,

$$M_{cs} p_s = r \tag{3.4}$$

where M_{cs} is the global mass matrix for the elements on the structure boundary, $p_s$is the solution vector with the projected pressures and r is the right-hand side vector in which the entry-i associated with vertex-i of the structure is evaluated as follows,

$$r_i = \sum_{e=1}^{Nefl} \sum_{g=1}^{Nptg} N_s^i(x_g)\, Af^{(e)}\, W(g)\, \hat{p}_f(x_g) \tag{3.5}$$

being N_{efl} the number of elements at the fluid mesh on the interface of both domains for which the summation over the Gauss points is not identically null, N_{ptg} is the number of Gauss points for the current fluid element, $A_f^{(e)}$ is the area (or length) of the current fluid element, $W(g)$ is the weight at the Gauss point g with coordinates x_g, $\hat{p}_f(x_g)$ is the fluid pressure evaluated at the Gauss point g and $N_s^i(x_g)$ is the value of the shape function associated with vertex-i on the structure boundary mesh, evaluated at Gauss point g. Solving the linear system of equations (3.4) allows to find the pressures on the structure side. Figure (3.2) depicts the elements previously mentioned.

4 Numerical Tests

The following section present the results of two basic numerical tests which were solved with both projection strategies in order to check their correct implementation and to compare the results obtained in each case. To this end, the projected solution is compared to the fluid solution in a vertex to vertex fashion as well as in its integral value. In all the tests, the solution to project is imposed on the fluid boundary side and it is chosen before hand to show the differences of the solution projected on the structure side. In both tests, only one Gauss point per fluid element is used for the conservative strategy.

4.1 Test 1

The solution on the fluid boundary side is defined by the function

$$f(x) = 1/\exp(4x) \tag{4.1}$$

while the integration interval (the common boundary) is given by $0 \leq x \leq 1$. The value of the exact integral of $f(x)$ in this interval is equal to $I1 = 0.2454211$.

Next, the interval on the fluid side is equally partitioned into 20 sub-intervals. On the structure side, the interval is subdivided into three elements as follows: the first element is $\overline{\text{n1n2}} = [0, 0.3]$, the second $\overline{\text{n2n3}} = [0.3, 0.7]$ and the third one $\overline{\text{n3n4}} = [0.7, 1.0]$. The nodal values computed by using the conservative strategy are:$u_1 = 0.914290$, $u_2 = 0.256954$, $u_3 = 0.046125$ and $u_4 = 0.020117$. Applying the composite trapezoidal rule to compute the integral on the structure boundary, the result is equal to $I2 = 0.24624$ and the corresponding relative error is $e_C = 0.00333$.

On the other hand, if the direct projection strategy is applied, the corresponding nodal values are: $u_1 = 1.0$, $u_2 = 0.301194$,$u_3 = 0.060810$ and $u_4 = 0.018316$, so that the integral computed with the composite trapezoidal rule gives the result $I3 = 0.27945$ with a relative error value of $e_D = .13865$. Figure (4.1) shows the function to be projected on the fluid side (blue line) and on the structure side obtained with the conservative scheme (red line). The direct projection, as expected, is exact if it is compared on a vertex to vertex basis.

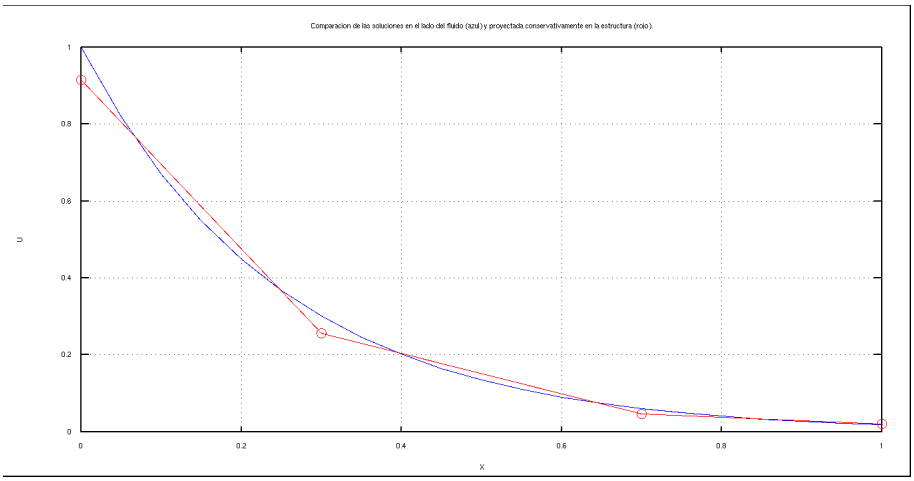

Figure 4.1: Test 1: Solutions on the fluid side (blue) and structure side (red) computed with the conservative scheme.

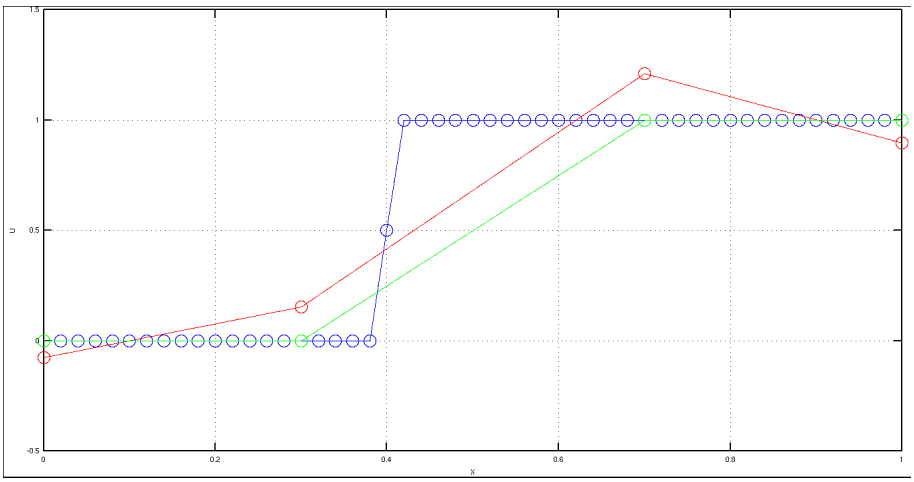

Figure 4.2: Test 2: Solutions on the fluid side (blue) and structure side computed with the conservative (red) and direct (green) scheme.

4.2 Test 2

In this test the solution to be projected is a step function defined in the interval $0 \leq x \leq 1$, which has the jump at coordinate $x = 0.4$. On the fluid boundary, the interval is uniformly divided into 50 sub-intervals and on the structure side it is used the same subdivision as in Test 1. The exact value of the integral on the fluid side is equal to $I1 = 0.60$. Next, the conservative projection strategy is applied and the nodal values for the solution on the structure side are: $u_1 = -0.0770856$, $u_2 = 0.154171$, $u_3 = 1.20947$ and $u_4 = 0.895267$. With these values and applying the composite trapezoidal rule, the integral on the structure side is equal to $I2 = 0.60000156$ and the corresponding relative error is $e_C = 2.6e - 6$.

On the other hand, if the direct projection scheme is applied, the corresponding nodal values on the structure side are: $u_1 = 0$, $u_2 = 0$, $u_3 = 1$ and $u_4 = 1$. Applying the composite trapezoidal rule, the value of the integral is equal to $I3 = 0.5$ and the corresponding relative error is $e_D = 0.16667$. Figure (4.2) shows the solution on the fluid side (blue line) as well as the solutions projected on the structure side with both the conservative (red line) and the direct (green line) strategies.

4.3 Test 3

In this test the fluid domain is a square of side length $L = 2$ with a hole of radius $R = 0.3$ in its center, which will be considered the fluid-structure interface. The fluid domain is discretized with an unstructured mesh of 4176 triangles which was generated with Gmsh [5]. The fluid-structure interface is divided into 40 line segments of equal length. The function $sin(2\theta)$ is imposed on the interval $\theta = [0, \pi]$ of the interface and the function $sin(\theta)$ is imposed on the interval $\theta = [\pi, 2\pi]$, where θ is considered positive in the counterclockwise direction, being zero on the positive x-axis. The structure domain is the circle of radius $R = 0.3$ which is discretized with an unstructured mesh of 76 triangles, also generated with Gmsh. Its boundary is divided into 20 line segments of equal length. Both meshes have non-matching vertices at the fluid-structure interface as it can be seen in figure (4.3). Figure(4.4) depicts the solution on the fluid (imposed) and on the structure, this latter computed with the conservative projection strategy. On the other hand, figure (4.5) shows the solutions along the interface for the fluid (blue) and for the structure computed both with the direct strategy (red) and conservative (green) strategies. The exact integral value of the solution

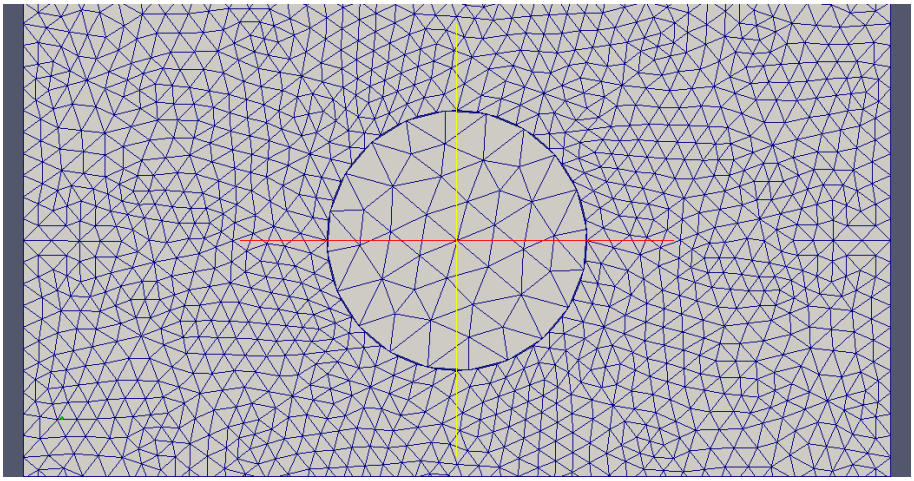

Figure 4.3: Test 3: Detail of non-matching meshes at the fluid - structure interface.

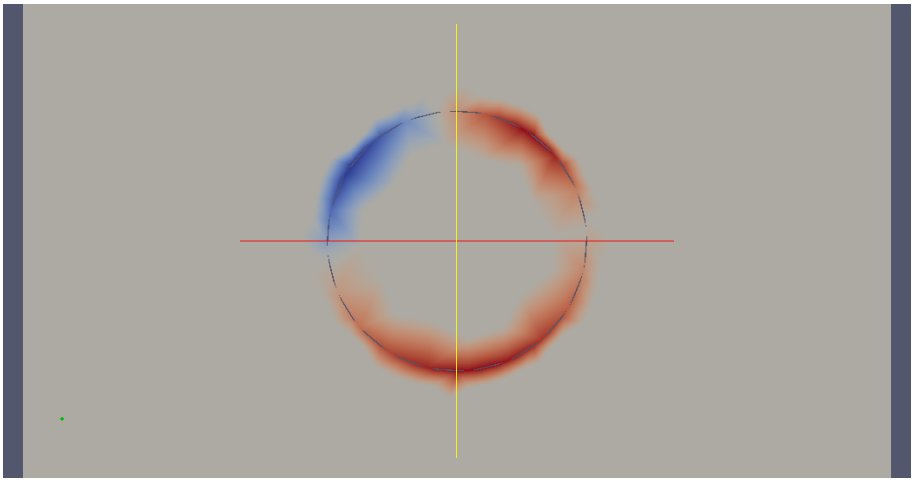

Figure 4.4: Test 3: Solution at the fluid - structure interface computed with the conservative scheme.

along the boundary is $I_1 = -2$. On the other hand, the integral values on the structure side computed with the solution obtained with the direct projection scheme is equal to $I_2 = -2.0777$ and with the conservative scheme is $I_3 = -2.0851$, so that the relative errors are $e_D = 0.038827$ and $e_C = 0.042529$, correspondingly.

4.4 Conclusions

Based on the results obtained in the numerical tests it can be concluded that the conservative scheme is much more accurate than the direct projection scheme if what matters is to preserve the total load transfer from the fluid boundary mesh to the structure boundary mesh. This behaviour is even more notorious when the solution on the fluid boundary exhibits jumps or sudden changes. The meshes used in the tests are very simple but all of them are non-matching at the interface so that the software developed provides a great advantage in computing the solution of FSI problems if compared to the solution schemes that require matching meshes.

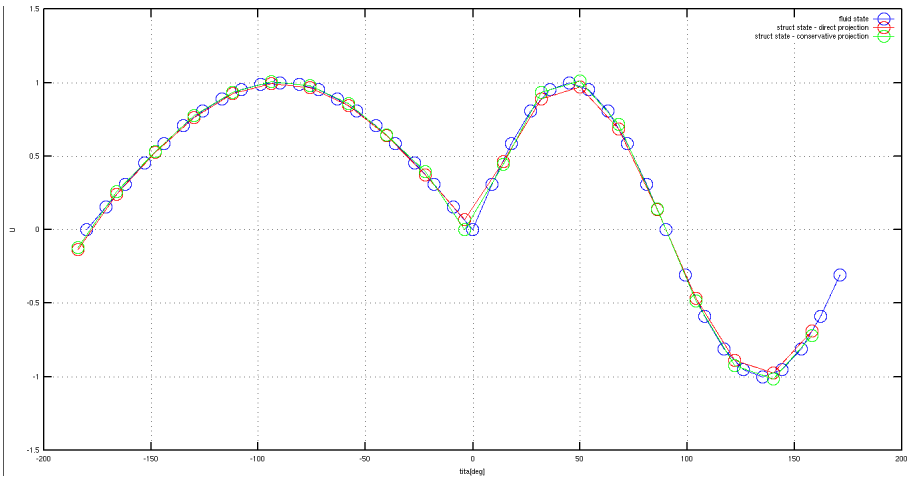

Figure 4.5: Test 3: Comparison of the computed solutions along the boundary.

5 Future works

The researchers of the IK and CIMEC agreed that future collaborative work should continue because of common interests between both institutes. A list of future works were discussed and here are reported by following a chronological order,

- Continue with the development and testing of the projection schemes to integrate the software into the PETSc-FEM package in order to automate the procedure of load transfer from the fluid to the structure mesh.
- Extend the projection software to solve the solution projection in 3-D problems.
- Include some kind of monotone conservative projection strategy.
- Development and implementation of a surface tracking scheme that allows the fluid mesh to follow the movement of the structure mesh. In most of the methods, the mesh deformation of the structure at some points is interpolated to the fluid mesh points by fitting some function through the control points on the structure boundary[3].

References

[1] S. Arya and D.M. Mount. *ANN: Approximate Nearest Neighbors Library.* Department of Computer Science, University of Maryland, 2010.

[2] Silja Beck, Dirk Clasen, Lutz Lehmann, Katherina Rurkowska, Marco Schauer, and Meike Wulkau. elpaso - manual, technical report revision:369. Technical report, Institut fur Angewandte Mechanik, Institut fur Konstruktionstechnik, TU Braunschweig, 2008-2014.

[3] J.R. Cebral. *Loose Coupling Algorithms for Fluid-Structure Interaction.* PhD Thesis, Institute for Computational Sciences and Informatics, George Mason University, 1996.

[4] L. Garelli and M. Schauer. Two and three dimensional coupling concept of PETSc-FEM and elPaSo for fluid structure interaction analysis. Technical report, Institut fur

Konstruktionstechnik, TU Braunschweig and Centro de Investigación en Métodos Computacionales, CIMEC-CONICET, 2013. http://venus.ceride.gov.ar/twiki/pub/Cimec/WebHome/report-mschauer-130515.pdf.

[5] C. Geauzaine and J.F. Remacle. Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. *International Journal for Numerical Methods in Engineering*, 79(11):1309–1331, 2009.

[6] R. Lohner. *Applied Computational Fluid Dynamics Techniques - An Introduction Based on Finite Element Methods*. John Wiley and Sons, Chichester, second ed. edition, 2008.

[7] M. Storti, N. Nigro, R. Paz, L. Dalcín, L. Battaglia, E. López, and G.A. Ríos Rodriguez. *PETSc-FEM, A General Purpose, Parallel, Multi-Physics FEM Program*. CIMEC-CONICET-UNL, 1999-2014. http://www.cimec.org.ar/petscfem.