International Journal of Engineering, Mathematical and Physical Sciences
ISSN: 2517-9934
Vol:8, No:5, 2014

Coupling Concept of two Parallel Research Codes for Two and Three Dimensional Fluid Structure Interaction Analysis

Luciano Garelli, Marco Schauer, Jorge D'Elia, Mario A. Storti, Sabine C. Langer

Abstract—This paper discuss a coupling strategy of two different software packages to provide fluid structure interaction (FSI) analysis. The basic idea is to combine the advantages of the two codes to create a powerful FSI solver for two and three dimensional analysis. The fluid part is computed by a program called *PETSc-FEM* a software developed at *Centro de Investigación de Métodos Computacionales –CIMEC*. The structural part of the coupled process is computed by the research code *elementary Parallel Solver –(*ELPASO*)* of the *Technische Universität Braunschweig, Institut für Konstruktionstechnik (IK)*.

Keywords—Computational Fluid Dynamics (CFD), Fluid Structure Interaction (FSI), Finite Element Method (FEM).

I. Introduction

THE design of many engineering systems has to consider fluid structure interactions, for instance aircraft, turbines and bridges, but also medical products like artificial heart implants. If effects of oscillatory interactions of fluid and structure is not considered while the design process, it can lead into catastrophe. One of the probably most famous examples of large-scale failure is the first Tacoma Narrows Bridge which collapsed in 1940. Aircraft wings and turbine blades can break due to FSI oscillations. Another example is the analysis of aneurysms in arteries and artificial heart valves here fluid structure interaction has also to be taken into account.

Due to the reason that the analysis of fluid dynamics as well as the analysis of complex structures in time domain requires high among of computational power, both codes are prepared for parallel computation on distributed memory systems, like computer clusters. Coupled problems like FSI will need even more computational performance. So that the FSI analysis benefits of the parallelism of the two codes. The communication of the processes is realised using the Message Passing Interface (MPI) [8]. The Portable, Extensible Toolkit for Scientific Computation (PETSc) [1]–[3] is used to provide all parallel matrices, vectors and solvers, needed for the computation.

Luciano Garelli, Jorge D'Elia and Mario A. Storti are with the Centro de Investigación de Métodos Computacionales, Consejo Nacional de Investigaciones Científicas y Técnicas (CONICET), Predio CONICET-Santa Fe, Colectora RN 168 Km 472, Paraje El Pozo, Argentina, (phone: +54-342-4511594, e-mail: lucianogarelli@gmail.com).

Marco Schauer and Sabine C. Langer are with Technische Universität Braunschweig, Institut für Konstruktionstechnik (IK), Langer kamp 8, 38106 Braunschweig, Germany, (phone +49 (0) 531 391 7108, fax +49 (0) 531 391 4572, m.schauer@tu-braunschweig.de, http://www.tu-braunschweig.de/ik).

II. Computational methods

The coupling process between these codes, *PETSc-FEM* [11] and ELPASO [4] will be carried out using a partitioned technique, which allows the use of preexisting specifics solvers. When a partitioned coupling technique is used, a three-field system is involved in the analysis: the structure, the fluid and the moving mesh solver. In general the governing equations of the fluid are written in an Eulerian framework so it must be rewritten to allow the motion of the mesh using an ALE (Arbitrary Lagrangian Eulerian) formulation. Regarding to the mesh movement, it is performed using a nodal relocation, maintaining the topology unchanged. To perform the nodal relocation of the mesh there exist several strategies, in *PETSc-FEM* to propagate the boundary motion into the volume mesh is solved a linear elastic or pseudo-elastic problem to deform the mesh. Also can be chosen a more sophisticated strategy to produce the mesh motion by solving an optimization problem, in order to obtain a better quality mesh.

The dynamic behaviour of a fluid flow is governed by the Navier-Stokes equations, which are set of coupled conservation laws. It can be enumerated as

- Conservation of mass,
- Conservation of momentum,
- Conservation of energy.

The Navier-Stokes equations can also be simplified in order to reproduce some particular kind of flows, as example if the viscosity is assumed to be zero, the fluid is treated as inviscid and is treated as incompressible if the density variations with respect to the reference density is negligible. These approximations are made based on the characteristics of the flow or based on the properties of the fluid. In this report a compressible, viscous flow is considered and is described as a general advective-diffusive system in order to simplify its interpretation.

The governing equation for the fluid can be written in a compact form as

$$\mathcal{R}(\mathbf{U}) \equiv \frac{\partial U_j}{\partial t} + \left(\mathcal{F}^c_{jk}(\mathbf{U}) - \mathcal{F}^d_{jk}(\mathbf{U}, \nabla\mathbf{U})\right)_{,k} = 0, \text{in}\Omega^t \quad (1)$$

where $1 \leq k \leq n_d$, n_d is the number of spatial dimensions, $1 \leq j \leq m$, m is the dimension of the state vector (e.g. $m = n_d+2$ for compressible flow), t is time, $(\,)_{,k}$ denotes derivative with respect to the k-th spatial dimension, $\mathbf{U} = (\rho, \rho\mathbf{u}, \rho e)^t \in \mathbb{R}^{n_d}$ is the unknown state vector expressed in conservative variable. Where ρ , $\mathbf{u}$ and e represents the density, the velocity

vector and the specific total energy respectively, and $\mathcal{F}_{jk}^{c,d} \in \mathbb{R}^{m\times n_d}$ are the convective and diffusive fluxes. Appropriate Dirichlet and Neumann conditions at the boundary and initial conditions must be imposed.

The differential equation (1) can be written in an integral form

$$\frac{\partial}{\partial t}\int_\Omega \begin{pmatrix}\rho\\ \rho\mathbf{u}\\ \rho E\end{pmatrix} d\Omega + \int_\Omega \begin{pmatrix}\rho\mathbf{u}\\ \rho\mathbf{u}\otimes\mathbf{u} + p\bar{\bar{I}} - \bar{\bar{\tau}}\\ \rho\mathbf{u}H - \bar{\bar{\tau}}\cdot\mathbf{u} - \kappa\nabla T\end{pmatrix}_{,k} d\Omega = 0, \quad (2)$$

where H is the total specific enthalpy

$$H = e + p/\rho + 1/2|\mathbf{u}|^2 = E + p/\rho \quad (3)$$

defined in terms of the specific internal energy ($\mathrm{h} = \mathrm{e} + \mathrm{p}/\rho$) and the specific kinetic energy, respectively.

This set of equations are closed by an equation of state, being for a polytropic gas

$$p = (\gamma - 1)[\rho e - \frac{1}{2}\rho||\mathbf{u}||^2], \quad (4a)$$

$$T = C_v[e - \frac{1}{2}\rho||\mathbf{u}||^2], \quad (4b)$$

where γ is ratio of specific heats and C_v is the specific heat at constant volume. In viscous fluxes, the stress tensor $\bar{\bar{\tau}}$ is defined for Newtonian flows as

$$\tau_{ij} = 2\mu\varepsilon_{ij}(\mathbf{u}) + \lambda(\nabla\cdot\mathbf{u})\delta_{ij}, \quad (5)$$

where the Stoke's hypothesis is

$$\lambda = -\frac{2}{3}\mu, \quad (6)$$

and the strain rate tensor is

$$\varepsilon_{ij}(\mathbf{u}) = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right). \quad (7)$$

Finally, the dynamic viscosity is assumed to be given by the Sutherland's law, which gives for an ideal gas the viscosity as function of the temperature,

$$\mu = \mu_0\left(\frac{T}{T_0}\right)^{3/2}\frac{T_0 + 110}{T + 110}, \quad (8)$$

where μ_0 is the viscosity at the reference temperature T_0.

In order to write the semi-discrete form of the compressible Navier-Stokes equations, it is convenient to write (1) in a quasi-linear form as

$$\frac{\partial\mathbf{U}}{\partial t} + \mathbf{A}_k\frac{\partial\mathbf{U}}{\partial x_k} - \frac{\partial}{\partial x_k}\left(\mathbf{K}_{ki}\frac{\partial\mathbf{U}}{\partial x_i}\right) = 0, \text{ in}\Omega^t\ \forall t\in(0,\mathcal{T}) \quad (9)$$

where

$$\frac{\partial\mathcal{F}^c}{\partial x_k} = \frac{\partial\mathcal{F}^c}{\partial\mathbf{U}}\frac{\partial\mathbf{U}}{\partial x_k} = \mathbf{A}_k\frac{\partial\mathbf{U}}{\partial x_k}, \quad (10a)$$

$$\frac{\partial\mathcal{F}^d}{\partial x_k} = \frac{\partial\mathcal{F}^d}{\partial\mathbf{U}}\frac{\partial\mathbf{U}}{\partial x_k} = \mathbf{K}_{ki}\frac{\partial\mathbf{U}}{\partial x_k}, \quad (10b)$$

and $\mathbf{A}_k$ are the Jacobians of advective fluxes and $\mathbf{K}_{ki}$ are the Jacobians of diffusive fluxes.

Previous to addressing the variational formulation of the governing equation, it is necessary to transform the equations to be solved in an ALE framework as describe in detail in [6], because the problem is posed in a time-dependent domain Ω^t, it can not be solved with standard fixed-domain methods. After multiplying equation (1) with a weighting function $w(\mathbf{x},t)$ and the ALE transformation we obtain

$$\frac{\mathrm{d}}{\mathrm{d}t}\left(\int_{\Omega^t} w\,U_j\,\mathrm{d}\Omega^t\right) + \int_{\Omega^t}\left(\mathcal{F}_{jk}^c - v_k^*U_j - \mathcal{F}_{jk}^d\right)_{,k} w\,\mathrm{d}\Omega^t = 0. \quad (11)$$

The variational formulation can be obtained by integrating by parts, so that

$$\frac{\mathrm{d}}{\mathrm{d}t}\left(H(w,\mathbf{U})\right) + F(w,\mathbf{U}) = 0, \quad (12)$$

where

$$\begin{aligned} H(w,\mathbf{U}) &= \int_{\Omega^t} w\,U_j\,\mathrm{d}\Omega^t,\\ F(w,\mathbf{U}) &= A(w,\mathbf{U}) + B(w,\mathbf{U}),\\ A(w,\mathbf{U}) &= -\int_{\Omega^t}\left(\mathcal{F}_{jk}^c - v_k^*U_j - \mathcal{F}_{jk}^d\right) w_{,k}\,\mathrm{d}\Omega^t,\\ B(w,\mathbf{U}) &= \int_{\Gamma^t}\left(\mathcal{F}_{jk}^c - v_k^*U_j - \mathcal{F}_{jk}^d\right) n_k w\,\mathrm{d}\Gamma^t. \end{aligned} \quad (13)$$

Γ^t is the boundary of Ω^t, n_k is its unit normal vector pointing to the exterior of Ω^t and v_k^* are the components of the mesh velocity.

Finally, the semi-discrete system is discretized in time with the trapezoidal rule, where

$$\begin{aligned} H(w,\mathbf{U}^{n+1}) - H(w,\mathbf{U}^n) &= -\int_{t^n}^{t^{n+1}} F(w,\mathbf{U}^{t'})\,\mathrm{d}t',\\ &\approx -\Delta t\,F(w,\mathbf{U}^{n+\theta}). \end{aligned} \quad (14)$$

with $0\le\theta\le 1$ and being $\mathbf{U}^{n+\theta}$ defined as

$$\mathbf{U}^{n+\theta} = (1-\theta)\mathbf{U}_n + \theta\mathbf{U}^{n+1}. \quad (15)$$

And during the time step it is assumed that the nodal points move with constant velocity, i.e.

$$\left.\begin{aligned} v_k^*(\boldsymbol{\xi}) &= \frac{x_k(\boldsymbol{\xi},t^{n+1}) - x_k(\boldsymbol{\xi},t^n)}{\Delta t},\\ x_k(\boldsymbol{\xi},t) &= x_k(\boldsymbol{\xi},t^n) + (t-t^n)v_k^*(\boldsymbol{\xi}), \end{aligned}\right\} t^n\le t\le t^{n+1}. \quad (16)$$

As was mentioned previously, in FSI problems the fluid interacts with a structure which deforms due to the forces exerted by the fluid, producing a change in the fluid domain, since the fluid-structure interface follows the structure displacement. In the discrete fluid flow problem, the change of the domain must be followed by a change in the discretization. The discretization of the new domain can be obtained through a re-meshing process or through a nodal relocation process. In general, the re-meshing process is undesirable because of the need of a projection of the flow field from the old to the new mesh, with the consequent loss of conservativity, possible addition of numerical diffusion and additional computational cost. In this coupling process, a relocation technique is used to update the nodal coordinates of the fluid mesh in response to the domain deformation, while keeping the topology unchanged.

A robust alternative to deform the mesh use a linear elasticity approach, where fluid mesh obey the linear elasticity

equation to obtain a smooth displacement field, setting as boundary condition the displacement of the interface. The equations describing the elastic medium under the hypothesis of small deformations without external forces are

$$\nabla \cdot \bar{\bar{\sigma}} = 0, \tag{17a}$$

$$\bar{\bar{\sigma}} = \lambda_s (\mathrm{tr}\bar{\bar{\mathrm{E}}})\bar{\bar{\mathrm{I}}} + 2\mu_s \bar{\bar{\mathrm{E}}}, \tag{17b}$$

$$\bar{\bar{E}} = \frac{1}{2}\left(\nabla \mathbf{x} + \nabla \mathbf{x}^T\right), \tag{17c}$$

where $\mathbf{x}$ is the displacement field and for constants λ_s and μ_s can be written as

$$\mu_s \nabla^2 \mathbf{x} + (\lambda_s + \mu_s)\nabla(\nabla \cdot \mathbf{x}) = 0. \tag{18}$$

In this approach the Lamé constants depend on the Young's modulus and Poisson's ratio, so a variable Young's modulus can be used in order to avoid severe mesh deformation in critical regions, like boundary layers, trailing-edge airfoil, relegating the mesh deformation to areas where the mesh is coarse. Exist other alternatives, like employing a distribution which is inversely proportional to the element volume, in order to deal with severe fluid mesh deformations. These strategies can be used to admit large mesh movement while maintaining good mesh qualities. Additionally, the moving mesh module of PETSc-FEM has implemented a robust method proposed by [7], where the mesh motion strategy is defined as an optimization problem. By its definition this strategy may be classified as a particular case of an elastostatic problem where the material constitutive law is defined in terms of the minimization of certain energy functional that takes into account the degree of element distortion. Some advantages of this strategy is its natural tendency to high quality meshes and its robustness.

The structural part of the coupled computation is provided by ELPASO a finite element method based software, which has been used for other couplings like FEM/SBFEM and FEM/BEM before [12]–[14]. Therefore finite elements are used to represent the structure for the time domain analysis. The displacement-based finite element method at an arbitrary time step can be written as

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{p} \tag{19}$$

where the vector $\mathbf{u}$ represents the nodal displacement $\dot{\mathbf{u}}$ the nodal velocities and $\ddot{\mathbf{u}}$ the nodal acceleration, and $\mathbf{p}$ denotes the applied nodal forces. Here, $\mathbf{M}$ is the mass matrix, $\mathbf{C}$ is the damping matrix and $\mathbf{K}$ denotes the stiffness matrix. Rayleigh damping is introduced by setting up the damping matrix as a linear combination of mass and stiffness matrix

$$\mathbf{C} = a\mathbf{K} + b\mathbf{M}. \tag{20}$$

Consider the time period T divided into n time steps with the duration $\Delta t = \frac{T}{n}$ and known initial values for $t = 0$

$$\mathbf{u}(t=0) = \mathbf{u}_0\,, \tag{21}$$

$$\dot{\mathbf{u}}(t=0) = \dot{\mathbf{u}}_0 \quad \text{and} \tag{22}$$

$$\ddot{\mathbf{u}}(t=0) = \ddot{\mathbf{u}}_0 \tag{23}$$

the equation of motion can be solved using Newmark scheme [9]. The values of the next time step can be evaluated using

$$\mathbf{u}_{n+1} = \mathbf{u}_n + \Delta t \dot{\mathbf{u}}_n + \left(\frac{1}{2} - \beta\right)\Delta t^2 \ddot{\mathbf{u}}_n + \beta \Delta t^2 \ddot{\mathbf{u}}_{n+1}\,, \tag{24}$$

$$\dot{\mathbf{u}}_{n+1} = \dot{\mathbf{u}}_n + (1-\gamma)\,\Delta t \ddot{\mathbf{u}}_n + \gamma \Delta t \ddot{\mathbf{u}}_{n+1} \tag{25}$$

and

$$\mathbf{M}\ddot{\mathbf{u}}_{n+1} + \mathbf{C}\dot{\mathbf{u}}_{n+1} + \mathbf{K}\mathbf{u}_{n+1} = \mathbf{p}_{n+1}\,. \tag{26}$$

The parameter β and γ are used to control the method. They should be defined as follows

$$\gamma \geq \frac{1}{2} \quad \text{and} \tag{27}$$

$$\beta = \frac{1}{4}\left(\gamma + \frac{1}{2}\right)^2. \tag{28}$$

Further details about ELPASO can be found in the users manual [4].

III. Coupling of fluid an structure

In this work a partitioned treatment will be used. The interaction process is carried out through the exchange of information at the fluid/structure interface in a staggered way. The structural solver establishes the position and velocity of the interface, while the fluid solver establishes the pressure and shear stresses on the interface. The principal advantage of the partitioned treatment, an the reason because it became so popular is that existing optimized solvers can be reused and coupled. The systems to be solved are smaller and better conditioned than in the monolithic case. However the disadvantage of this approach is that it requires a careful implementation in order to avoid serious degradation of the stability and accuracy. From this basic approach a weak (Explicit) scheme can be developed, or either a strong (Implicit) time coupling scheme. Each one of these schemes will be described below.

During the iterative process three codes CFD (Computational Fluid Dynamics), CSD (Computational Structure Dynamics) and CMD (Computational Mesh Dynamics) are running simultaneously.

The basic scheme proceeds as follows:

i) Transfer the motion of the wet boundary (interface) of the solid to the fluid problem.
ii) Update the position of the fluid boundary and the bulk fluid mesh accordingly.
iii) Advance the fluid system and compute new pressures.
iv) Convert the new fluid pressure (and stress field) into a structural load.
v) Advance the structural system under the flow loads.

From this basic description two different coupling schemes can be derived depending on how the prediction of the structural displacement for updating the position of the fluid boundary and compute new pressures is made. To proceed with the description of the scheme we define $\mathbf{w}^n$ to be the fluid state vector $(\rho, \mathbf{v}, p)$, $\mathbf{z}^n$ to be the displacement vector

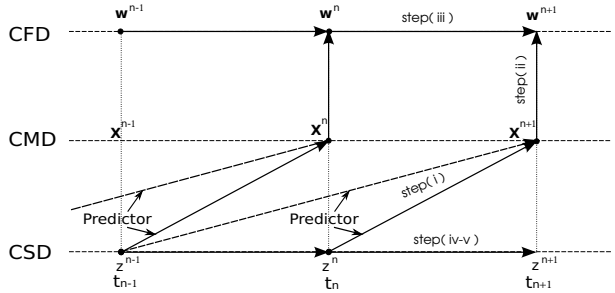

Fig. 1. Weak coupling scheme.

(structure state vector), $\dot{\mathbf{z}}^n$ the structure velocities and $\mathbf{x}^n$ the fluid mesh node positions at time t_n.

In the weak (explicit) coupling (See Fig. 1) the fluid is first advanced using the previously computed structure state $\mathbf{z}^n$ and a current estimated value $\mathbf{z}_p^{n+1}$. In this way, a new estimation for the fluid state $\mathbf{w}^{n+1}$ is computed. Next the structure is updated using the forces of the fluid from states $\mathbf{w}^n$ and $\mathbf{w}^{n+1}$. The estimated state $\mathbf{z}_p^{n+1}$ is predicted using a second or higher order approximation (29), were α_0 and α_1 are two real constants. The predictor (29) is trivial if $\alpha_0 = \alpha_1 = 0$, first-order time-accurate if $\alpha_0 = 1$ and second-order time-accurate if $\alpha_0 = 1$ and $\alpha_1 = 1/2$. This coupling scheme has been proposed in [5], [10], with good results in the resolution of aeroelastic problems.

$$\mathbf{z}_p^{(n+1)} = \mathbf{z}^n + \alpha_0 \Delta t \dot{\mathbf{z}}^n + \alpha_1 \Delta t (\dot{\mathbf{z}}^n - \dot{\mathbf{z}}^{n-1}). \qquad (29)$$

Once the coordinates of the structure are known, the coordinates of the fluid mesh nodes are computed by a CMD code, which is symbolized as:

$$\mathbf{x}^{n+1} = \mathrm{CMD}(\mathbf{z}^{n+1}). \qquad (30)$$

Finally, can be also adopted the strong (implicit) coupling, which have benefits is term of stability and is comparable with a monolithic coupling. In this coupling algorithm, the time step loop is equipped with an inner loop called "stage", so if the "stage loop" converges, then a "strongly coupled" algorithm is obtained. A schematic diagram is shown in Fig. 2. With this coupling strategy the computational cost increases

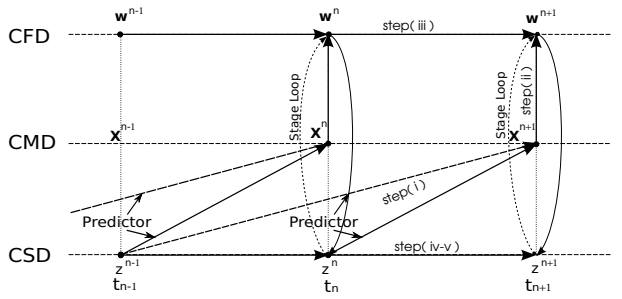

Fig. 2. Strong coupling scheme.

proportionally to the number of stages needed to achieve the desired error, but also it allows to use large time steps.

At the beginning of each fluid stage there is a computation of skin normals and velocities. This is necessary due to the time dependent slip boundary condition for the inviscid case, $((\mathbf{v}|_\Gamma - \dot{\mathbf{z}}|_\Gamma) \cdot \hat{\mathbf{n}} = 0)$ and also when using a non-slip boundary condition, where the fluid interface has the velocity of the moving solid wall, i.e., $\mathbf{v}|_\Gamma = \dot{\mathbf{z}}|_\Gamma$.

The load vector $\mathbf{p}$ applied to the structure is updated in each time step n

$$\mathbf{p}_n = \mathbf{p}_S + \mathbf{p}_F \,. \qquad (31)$$

It is composed as the sum of predefined loads applied on the structure $\mathbf{p}_S$ and forces acting on the structure due to the surrounding pressure field of the fluid $\mathbf{p}_F$.

IV. Interface

Both Programs *PETSc-FEM* and ELPASO are executed at the same time. A new implemented interface assures the transfer of informations between these two codes in both directions. The structural solver ELPASO applies nodal forces generated by the surrounding fluid to the structure and returns velocities and displacements to the fluid solver *PETSc-FEM*.

A. Named Pipe

The concept of named pipes (also known as a FIFO – first in first out – for its behaviour) is used to permit the communication and consequently the data transfer between the different codes. Named pipes are an extension to the traditional pipe concept on Unix and Unix-like systems, it is one of the methods of inter-process communication. The concept is also found in Microsoft Windows, although the semantics differ substantially. A traditional pipe is "unnamed" because it exists anonymously and persists only for as long as the process is running. A named pipe is system-persistent and exists beyond the life of the process and must be deleted once it is no longer being used. Processes generally attach to the named pipes (usually appearing as a file) to perform inter-process communication (IPC).

Instead of a conventional, unnamed, shell pipeline, a named pipeline makes use of the file system. It is explicitly created using *mkfifo()* or *mknod()*, and two separate processes can access the pipe by name one process can open it as a reader, and the other as a writer.

To use named pipes for the communication, the communicating parts of the two programs have to be executed on the same physical machine. Here the communication is done by the first process aka *rank0*. It has to be assured, that the instances of *rank0* are located in the same computer. Otherwise the communication via named pipe will fail.

Here four different pipes are created to permit coupling in both directions, which are named *adv2str.fifo*, *str2adv.fifo*, *adv2mmv.fifo* and *mmv2adv.fifo*. The first pipe *adv2str.fifo* controls the data transfers nodal forces from *PETSc-FEM* to ELPASO or CFD to CSD, the second pipe controls the transfer nodal velocities and displacements from ELPASO to *PETSc-FEM* or CSD to CFD. The pipes *adv2mmv.fifo* and *mmv2adv.fifo* are used to synchronise the CFD code with the CMD code.

V. Numerical examples

The two following numerical examples shown here are worked out using the weak coupling approach as described in Section III.

A. Two Dimensional Test Case

In the first test case is solved a compressible flow around a flexible beam which is fixed at the bottom, this simple problem allows to verify the exchange of information between the codes and simplify the debugging process. The structure is discretized as a frame using 18 *BeamBernoulli2* elements [4]. Those elements have two nodes with six degrees of freedom each, three for displacements in x, y and z direction and another tree for rotation around the x, y and z axis, which leads to a system size of 114 degrees of freedom. All six degrees of freedom are fixed at the bottom of the domain (Node 1 and 19). The structural material parameters which sused in this example are shown in Tables I and II.

TABLE I
Structures material parameters

E [Nm^{-2}]	ν [-]	ρ [kgm^{-3}]
2.1E+5	0.3	3900.0

TABLE II
Geometrical parameters

A [m^2]	I_x [m^4]	I_y [m^4]	I_z [m^4]]
7.64E-04	8.859E-07	8.01E-07	8.49E-08

The fluid domain is discretized with 40 elements in the x direction and 15 elements in the y direction with a total of 584 elements. At the inlet is imposed the density and the velocities in the x and y directions while at the outlet is imposed the pressure. At the bottom and at the top of the domain is imposed a slip condition ($\mathbf{v} \cdot \mathbf{n} = 0$), while at the interface with the structure is set a no slip condition ($\mathbf{v} = 0$). The properties of the fluid and the reference condition are describe in Table III and Table IV.

TABLE III
Fluid parameters.

ρ_{in} [Kg/m^3]	γ [-]	R [J·K/Kg]	ν [Pa· s]
1	1.4	1	1E-05

TABLE IV
Fluid parameters.

p_{out} [Pa]	u_{in} [m/s]	v_{in} [m/s]
1	2	0

The dimension of the fluid domain and the values of the boundary conditions is shown in Fig. 3.

The coupled problems behaviour is analysed for a period of 400 seconds, therefore 2000 time steps with a time step length $\Delta t = 0.2$ seconds are computed. Both codes use the same time step length. The structures initial conditions at $t = 0$ are set to zero. γ and β parameters of the time stepping scheme are set to 0.5 and 0.25, respectively. To achieve a stable simulation a Rayleigh damping is introduced according to (20), here a and b are set to 0.001.

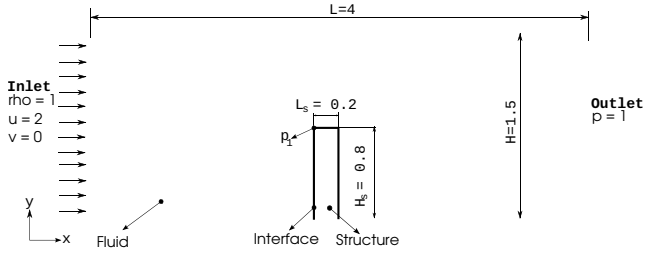

Fig. 3. Fluid domain and dimensions.

TABLE V
Coupling table of *PETSc-FEM*, elPaSo interface

PETSc-FEM node id	elPaSo node id	in/output line number	elPaSo $\mathbf{u}$, $\dot{\mathbf{u}}$, $\mathbf{p}$ vector index $_i$
17	1	1	0, 1
35	9	2	48, 49
36	11	3	60, 61
37	10	4	54, 55
38	19	5	108,109
39	12	6	66, 67
40	13	7	72, 73
41	14	8	78, 79
42	15	9	84, 85
43	16	10	90, 91
44	17	11	96, 97
45	18	12	102,103
120	2	13	6, 7
121	3	14	12, 13
122	4	15	18, 19
123	5	16	24, 25
124	6	17	30, 31
125	7	18	36, 37
126	8	19	42, 43

Table V shows the coupling information used by the programs. Due to the fact, that the ids of the nodes are not the same, the mapping has to be done. The first column of the table shows the fluid nodes, which are used for the coupling. In the second column the corresponding structural nodes are shown. To exchange the coupling information values are organized as shown in column three. The position of values in the system vectors $\mathbf{u}$, $\dot{\mathbf{u}}$, $\mathbf{p}$ in the structure solver are listed in the last column. The two index numbers are related to the displacement u_i, velocity $\dot{u}_i$ and force p_i components in x and y direction.

As results of this simulation is shown the magnitude of the displacement at point 1 in the structure (See Fig.3). In Fig. 4 can be observed the behaviour of the beam when it is subjected to the load produced by a fluid flow. After 300 time steps the beam reach an oscillating frequency which correspond to the natural frequency of the beam coupled with the fluid. The damping added to the beam plus the damping of the fluid makes that a stable position were achieved after 1700 time steps.

In Fig. 5 a sequence of images for time steps {0,50,140,240,500,1000} shows the deformation of the beam

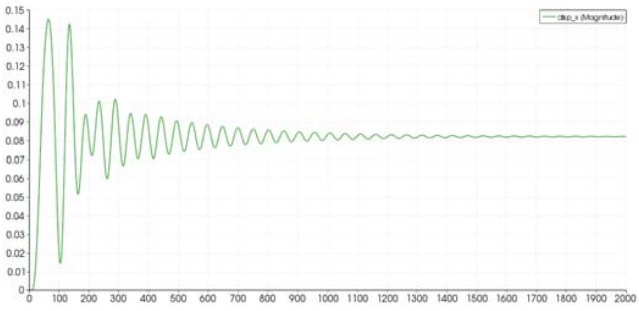

Fig. 4. Magnitude of the displacement at point 1.

and the streamlines of the fluid. In this sequence can be observed the oscillation of the beam and the effect that it has over the fluid field, changing the size and position of the vortex behind the beam.

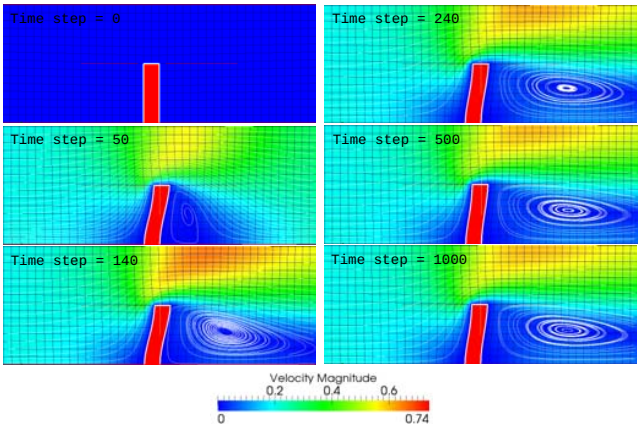

Fig. 5. Deformations of the beam and streamlines for time steps {0,50,140,240,500,1000}.

B. Three Dimensional Test Case

In this test case is solved the fluid field inside a 3D lid-driven cavity, which has a flexible wall at bottom (z=0). In Fig. 6 is shown the dimensions and the numbers of cell in each direction. In the upper face is imposed a velocity in the x direction with a value of 2 [m/s]. The fluid parameters are the same that used in the 2D test case (See Table III).

To discretize the structure at the bottom *PlShell4* elements are used [4], those have four nodes with six degrees of freedom each. A discretization with 40×40 elements leads to a system with 10086 unknowns. On all four edges the six degrees of freedom of each node are forced to be zero so that the plate is fixed on its edges. Material parameters are given in table VI. To achieve a stable simulation a Rayleigh damping is introduced according to(20), here a and b are set to 0.015.

In Fig.7) is plot the displacement magnitude of a point at the middle of the flexible plate. Due to the Rayleigh damping is introduced in the plate, the amplitude of the oscillation is decreasing and converges to a value around -0.039 [m].

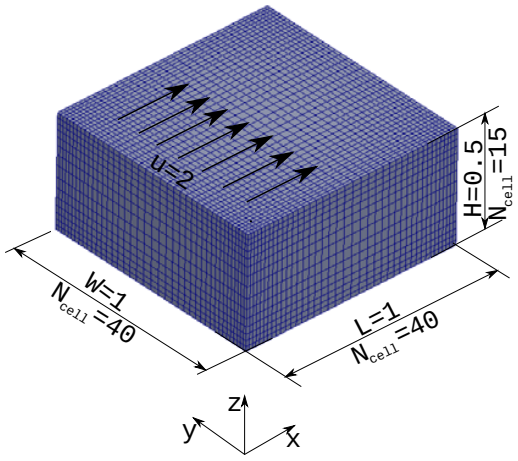

Fig. 6. 3D Lid-driven cavity.

TABLE VI
STRUCTURES MATERIAL PARAMETERS

E [Nm^{-2}]	ν [-]	t [m]	ρ [kgm^{-3}]
2.1E+4	0.3	0.01	2000.0

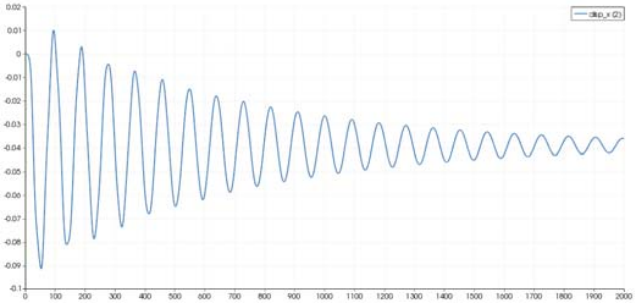

Fig. 7. Displacement Magnitude at the middle of the plate.

In Fig. 8 is shown a sequence of images for time steps {0,25,50,150,300,1000} with the deformation of the plate and the streamlines of the fluid. In this sequence can be observed how the deformation of the plate modify the fluid field, changing the size and position of the vortex.

VI. Conclusion

The weak (explicit) coupling of both programs works very well and leads to good results. Further improvements regarding the coupling like

- implicit coupling to improve stability of the computation
- analysis of real life models to gain the advantage of parallel computation
- meshes with different resolutions for fluid and structure domain

are the aim of future work. This includes staff exchange between the two partners; CIMEC, Santa Fe, Argentina and *Technische Universität Braunschweig, Institut für Konstruktionstechnik (IK)*, Germany.

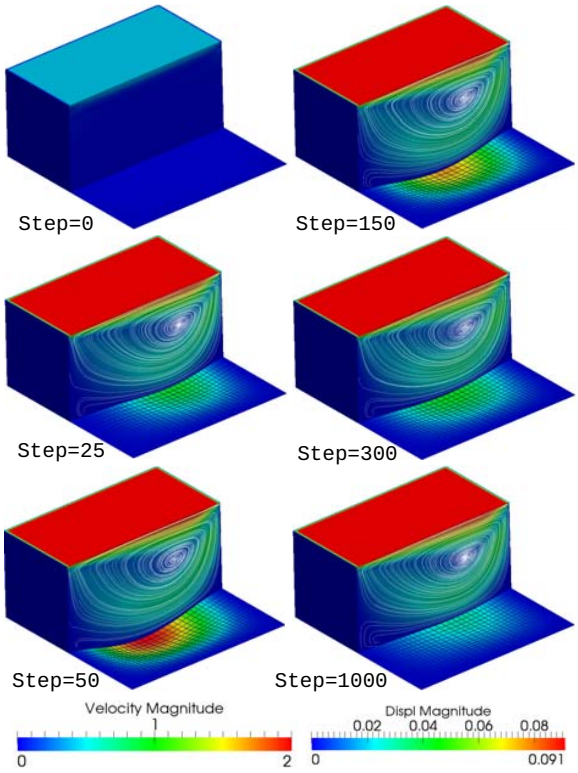

Fig. 8. Fluid Velocity, plate displacement and streamlines .

ACKNOWLEDGMENT

This common project work is supported by International Research Staff Exchange Scheme (IRSES), project NumSim PIRSES-GA2009-246977 Marie Curie Actions funded under the 7th Framework Programme of the European Commission. Also this work has received financial support from *Agencia Nacional de Promoción Científica y Tecnológica* (ANPCyT, Argentina, grant PICT 2492/2010), Extensive use of freely distributed software such as *GNU/Linux* OS, MPICH, PETSc, Metis, Octave, ParaView, and many others is done in this work.

REFERENCES

[1] Satish Balay, Jed Brown, Kris Buschelman, Victor Eijkhout, William D. Gropp, Dinesh Kaushik, Matthew G. Knepley, Lois Curfman McInnes, Barry F. Smith, and Hong Zhang. PETSc users manual. Technical Report ANL-95/11 - Revision 3.2, Argonne National Laboratory, 2011.

[2] Satish Balay, Jed Brown, Kris Buschelman, William D. Gropp, Dinesh Kaushik, Matthew G. Knepley, Lois Curfman McInnes, Barry F. Smith, and Hong Zhang. PETSc Web page. http://www.mcs.anl.gov/petsc, 2011.

[3] Satish Balay, William D. Gropp, Lois Curfman McInnes, and Barry F. Smith. Efficient management of parallelism in object oriented numerical software libraries. In E. Arge, A. M. Bruaset, and H. P. Langtangen, editors, *Modern Software Tools in Scientific Computing*, pages 163–202. Birkhäuser Press, 1997.

[4] Silja Beck, Dirk Clasen, Lutz Lehmann, Katherina Rurkowska, Marco Schauer, and Meike Wulkau. ELPASO – manual. Technical Report Revision: 369, TU Braunschweig, Institut für Angewandte Mechanik (2008–2013), Institut für Konstruktionstechnik (since 2013), 2008–2014. https://www.tu-braunschweig.de/ik

[5] C. Farhat, M. Lesoinne, and N. Maman. Mixed explicit/implicit time integration of coupled aeroelastic problems: Three-field formulation, geometric conservation and distributed solution. *International Journal for Numerical Methods in Fluids*, 21:(10):807–835, 1995.

[6] L. Garelli. *Fluid Structure Interaction using an Arbitrary Lagrangian Eulerian Formulation*. PhD thesis, Facultad de Ingeniería y Ciencias Hídricas, 2011.

[7] E. Lopez, NM. Nigro, MA. Storti, and J. Toth. A minimal element distortion strategy for computational mesh dynamics. *International Journal for Numerical Methods in Engineering.*, 69:9:1898–1929, 2007.

[8] MPI Forum. The Message Passing Interface (MPI) standard. http://www.mcs.anl.gov/mpi, 1994.

[9] N. Newmark. A method of computation for structural dynamics. *Journal of Engineering Mechanics Division*, 85:67, 1959.

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

[11] 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, 1999-2010. http://www.cimec.org.ar/petscfem.

[12] K. Rurkowska, M. Schauer, S. Langer. *On Neumann - Dirichlet coupling strategy of Finite Element and Boundary Element method in elastodynamics* Proceedings in Applied Mathematics and Mechanics, 11, 256–266, 2011

[13] Marco Schauer, Jose E. Roman, Enrique S. Quintana-Ortí, Sabine Langer. *Parallel Computation of 3-D Soil-Structure Interaction in Time Domain with a Coupled FEM/SBFEM Approach* Journal of Scientific Computing, 52, 446–467, 2012

[14] M. Schauer, S. Langer, J.E. Roman, E.S. Quintana-Ort. *Large Scale Simulation of Wave Propagation in Soils Interacting with Structures with the SBFEM Method* Journal of Computational Acoustics, 19, (1), 75–93, 2011