

# **LS-DYNA® R7: Strong Fluid Structure Interaction (FSI) capabilities and associated meshing tools for the incompressible CFD solver (ICFD), applications and examples.**

Facundo Del Pin  
İñaki Çaldichoury

*Livermore Software Technology Corporation  
7374 Las Positas Road  
Livermore, CA 94551*

## **Abstract**

*LS-DYNA version R7 includes CFD solvers for both compressible and incompressible flows. The solvers may run as standalone CFD solvers or they could be coupled to the LS-DYNA solid mechanics and thermal solvers for fluid structure interaction (FSI) and conjugate heat transfer problems.*

*This paper will focus on the Incompressible CFD solver in LS-DYNA (ICFD) and its Fluid-solid interaction capabilities (FSI). Fluid structure interaction problems occur in physics whenever the flow over a structure causes deformation or displacement which in turn may influence the way how the fluid behaves. One of the solver's main features is the implementation of a robust strong FSI coupling which opens a wide new range of applications in the range of aerodynamics, hydrodynamics, hemodynamics and so forth. Several examples will be provided for illustration and discussion.*

*The ICFD solver is the first in LS-DYNA to make use of a new volume mesher that takes surface meshes bounding the fluid domain as input. For FSI problems that involve big displacements, the volume mesher algorithms need to be robust and flexible. Some of the latest developments and mesh control tools that are made available for the user will therefore also be introduced.*

## 1- Introduction

Fluid structure interaction problems occur in physics whenever the flow over a structure causes deformation which in turn, may influence the way the fluid behaves. In order to accurately representing these interactions, the fluid part and the structural part evolve in a coupled system where fluid forces applied on the solid and solid displacements through the fluid interact at each time step.

This opens a very wide range of analysis and applications such as the fatigue of airplane wings, airfoils or turbine blades for the aerospace and turbo machinery industry, the study of galloping, vibrations and fluttering of flaps, bridges or similar structures, the analysis of artificial heart valve openings or other prosthesis for the bio-medical industry and so on.

LS-DYNA version R7 double precision includes the incompressible flow solver (ICFD) which can run as a stand-alone CFD solver or be coupled with the structural and thermal solvers of LS-DYNA. One of the solver's main features is the implementation of a robust strong FSI coupling that be triggered through the use of LS-DYNA's implicit solver for the solid mechanics part. The main focus of this paper will be a description of its mechanism and potential applications.

## 2- Types of FSI coupling

Fluid-structure interaction problems involving an incompressible viscous flow and elastic non-linear-structure have been solved in the past using different methods.

The monolithic approach considers the fluid and the solid as a single domain with the fluid and solid equations solved together in a coupled way. However, solving the pressure together with the rest of the unknowns (typically velocities or displacements) is too expensive from the computational point of view: the non-linear system to be solved is large and ill-conditioned with always non-defined positive matrices.

A second approach would be to segregate the pressure from the velocity in the monolithic scheme. This would still imply that the fluid and solid equations are solved in a coupled way in the same system and would therefore require an implementation of the solid equations directly in the ICFD solver. This would not be a practical solution and would contradict the objective of the present solver which is to make fully use of LS-DYNA's mechanical solver capabilities in order to solve complex fluid structure interaction problems.

A third approach would be by using a partitioned (or staggered) method ( [1], [2]) where the fluid and solid equations are uncoupled and that therefore allows using specifically designed codes on the different domains and offer significant benefits in terms of efficiency: smaller and better conditioned subsystems are solved instead of a single problem. It is the method adopted by the present solver in order to solve FSI problems.

In the partitioned approach, two schemes are distinguished: loosely (or weakly) ([17]) coupled scheme or strongly coupled scheme ( [3], [4]). Both are available when using the ICFD solver in LS-DYNA. Loosely coupled schemes require only one solution of either field per time step in a sequentially staggered manner and are thus particularly appealing in terms of efficiency. However, they tend to become unstable when the "added-mass effect" is significant ( [5]).

In fluid mechanics, added mass or virtual mass is the inertia added to a system because an accelerating or decelerating body must move some volume of surrounding fluid as it moves through it, since the object and fluid cannot occupy the same physical space simultaneously. However the name "added-mass effect"

has been used in the literature to indicate the numerical instabilities that typically occur in the internal flow of an incompressible fluid whose density is close to the structure density. The added mass effect therefore does usually not occur in aero-elasticity problems as the solid's density is a lot higher than the fluid's density, but it becomes very important in several other applications such as bio-mechanics where the materials are normally muscles and arteries and the fluid is blood.

Strongly coupled schemes require the convergence of the fluid and solid variables at the interface and give, after an iterative process, the same results as non-partitioned schemes. However, they are also subject to the "added mass effect" resulting in a non-convergence of the solution. Special stabilization techniques must therefore be developed in order to diminish its influence and aim for a wider range of applications for fluid structure interactions. Figure 1 gives a summary of the different FSI couplings possible, those implemented in the ICFD solver and the influence of the "added mass effect". Both loose and strong coupling are available to the user but in this paper the focus will be brought on the strong coupling only.

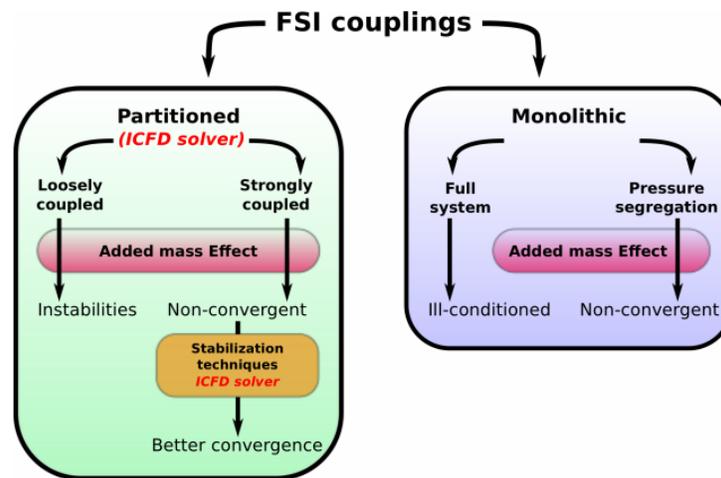


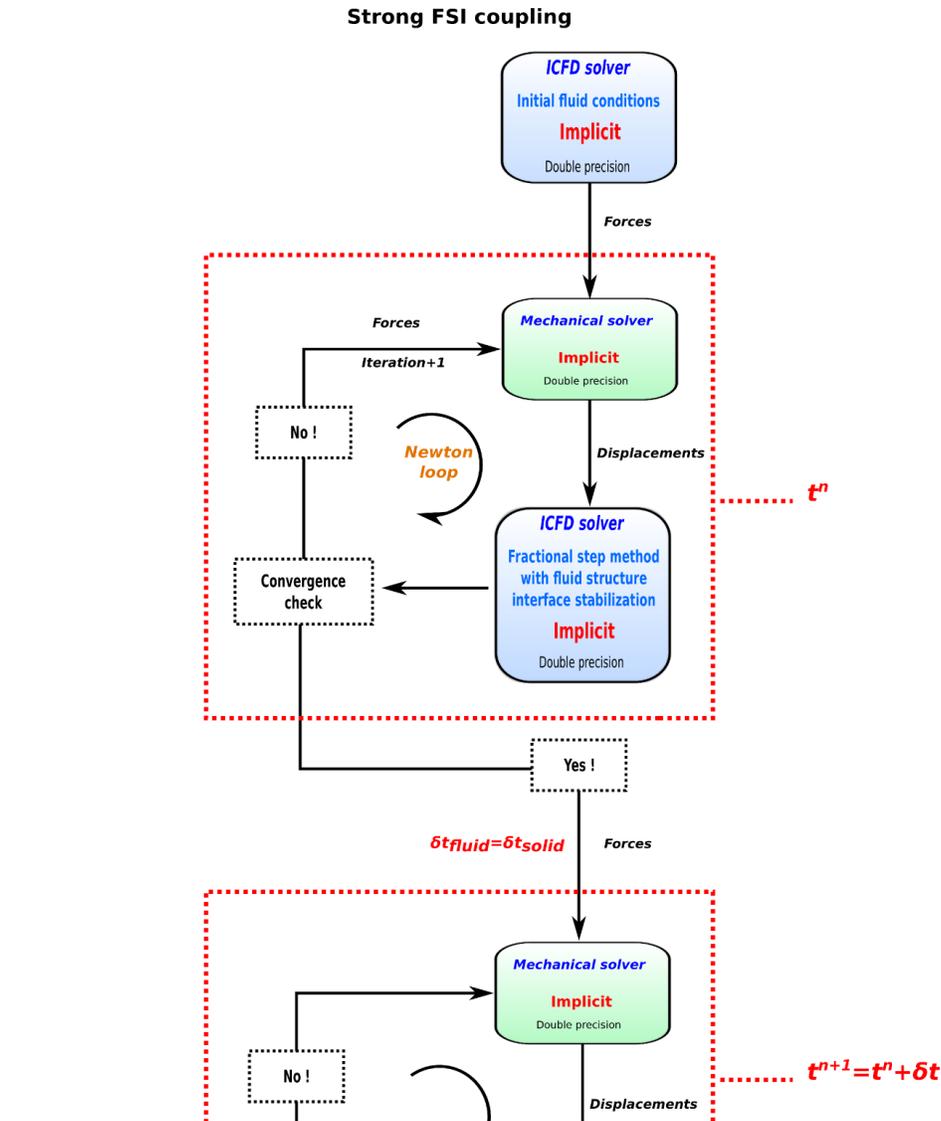
Figure 1 Summary of the different FSI couplings and the problems induced by the added mass effect.

### 3- Strong FSI coupling in LS-DYNA R7

#### 3.1- Resolution scheme

Both the ICFD solver and the implicit solver for solid mechanics are two independent solvers that both usually use a distinct time step. However, when the FSI coupling is activated, the strong coupling mechanism will be automatically triggered. In that case, the smallest time step of the two solvers will be used in order to solve the coupled problem. During the initial resolution step, the ICFD solver will solve the fluid domain using the initial velocity and pressure conditions. The forces applied on the structure will then be applied on the solid and recognized as pressure load boundary conditions. The solid will use those results to solve its own solid domain. It will then give back the displacements of its boundary nodes to the fluid. The solid and the fluid then go into a convergence procedure (Newton Loop) where a certain number of iterations are conducted until a convergence criterion is reached. The two solvers are then in agreement regarding the solid fluid interface boundary conditions (pressure and displacements). After

convergence is reached, the solver then proceeds to the next time step where this procedure is repeated. Figure 2 sums up this strong coupling mechanism.



### 3.2- Mesh handling

In a CFD only analysis, the moving reference frame is fixed in space and a full Eulerian formulation is achieved. However, in cases of Fluid Structure Interaction (FSI) problems, the fluid boundary nodes between the solid and the fluid are Lagrangian and the whole fluid mesh domain deforms with the structure. This approach allows a strong and exact imposition of the solid boundary conditions on the fluid.

The solid and fluid geometry must match at the interface but not necessarily the meshes. Figure 3 illustrates the mesh behavior on a FSI case.

The ICFD solver uses an automatic volume mesh generator for the fluid domain. However, robust tools are necessary in order to cope with the severe mesh deformations and distortions that can occur in FSI cases due to the structure's displacements. By default, the solver will apply a remeshing of the entire fluid domain when an element gets inverted. The user has also the possibility to force a remeshing if elements that are too distorted are detected. Further advanced meshing tools are made available to the user. For instance the user can trigger an automatic remeshing where the solver will use an a-posteriori error estimator to compute a new mesh size bounded by the user to satisfy a maximum perceptual error based on the work by [6]. For every nodal value, it is therefore possible to associate an error value. As a result, if too many nodes reach the critical value set by the user, the solver will apply a remeshing and refine in the zones where the error was high. Figure 4 gives an example where this option is being used.

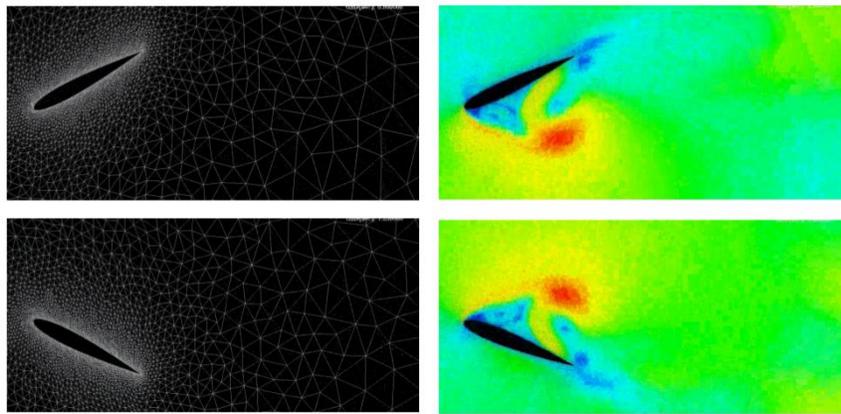


Figure 3 NACA airfoil profile oscillating in fluid. Incoming flow velocity from left boundary

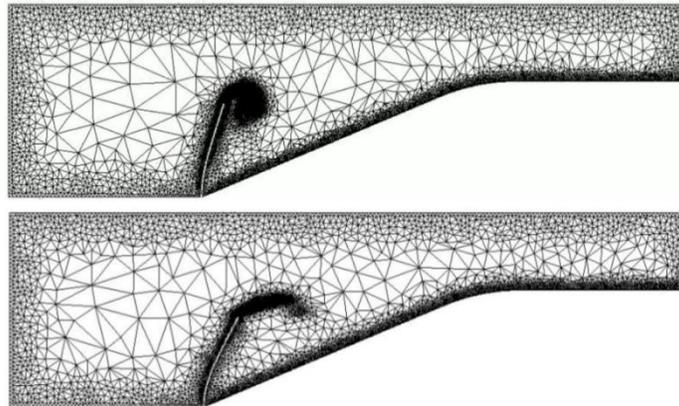


Figure 4 Incoming flow from the left boundary causing the bending of elastic flag and adaptive remeshing in zones of interest

#### 4- Validation test case: the Turek problem

In this example we consider the flow in a 2D channel past a cylinder with an attached elastic "flag". This is the FSI benchmark problem proposed by Turek and Hron [7]. This problem combines the two single-physics problem of the flow past a cylinder with a "flag" and the deformation of a finite-thickness cantilever

beam due to its interaction with the fluid. It is a challenging problem as it tackles some of the main issues when dealing with fully coupled FSI problems namely large mesh deformations and the fluid's density being close to the density of the structure which usually results in critical instabilities. Figure 5 offers a sketch of the problem with  $L_{channel} = 2.5$ ,  $HL_{channel} = 0.41$ ,  $d = 0.1$ ,  $L_{flag} = 0.35$ ,  $H_{flag} = 0.02$  and with the following cylinder center coordinates:  $(x=0.2, y=0.2)$ . A mesh size of 0.0025 will be used resulting in a 125 000 element mesh. Table 1 gives a summary of the parameters of three FSI cases that can be run. A strong FSI coupling is mandatory in order to solve the cases where the fluid and the solid's densities are equal.

Table 2 shows the result for the FSI1 case regarding the lift and drag values. In that case the flag's displacements are very small and hard to reproduce which explains that the lift is slightly different from the reference result. For the FSI2 and FSI3 cases, periodic solutions can be observed with the largest deformation occurring in the FSI2 case as can be observed on Figure 6. Frequent re-meshing is needed. The oscillation period is of 0.5 and 0.19 for the FSI2 and FSI3 cases respectfully which is in good agreement with the results of [7]. Figure 7 , Figure 8, and Table 2 show the good agreement between the present analysis and the reference results. It is interesting to observe that a better agreement is found in the FSI3 case regarding the lift values. One possible explanation could be that the wall effects become more important in the FSI2 case.

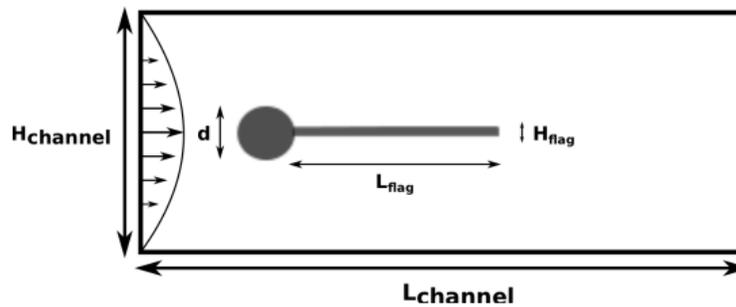


Figure 5 Test case sketch

Table 1 Parameter settings

	FSI 1	FSI 2	FSI 3
<b>Solid Density</b>	1000	10 000	1000
<b>Poisson Coefficient</b>	0.4	0.4	0.4
<b>Young Modulus</b>	1.4e6	1.4e6	5.6e6
<b>Fluid Density</b>	1000	1000	1000
<b>Fluid Viscosity</b>	1	1	1
<b>Average inflow velocity</b>	0.2	1	2
<b>Reynolds number</b>	20	100	200

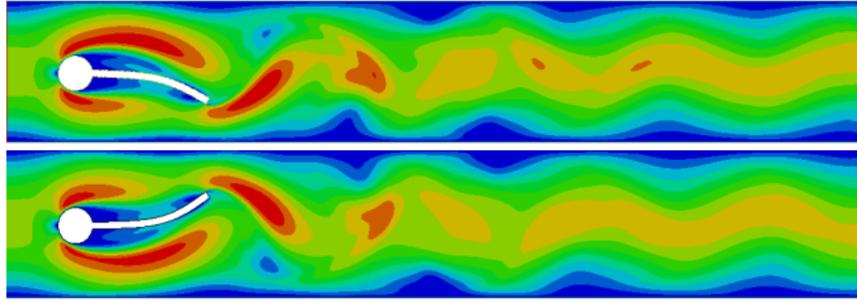


Figure 6 Flag deformation in the FSI2 case

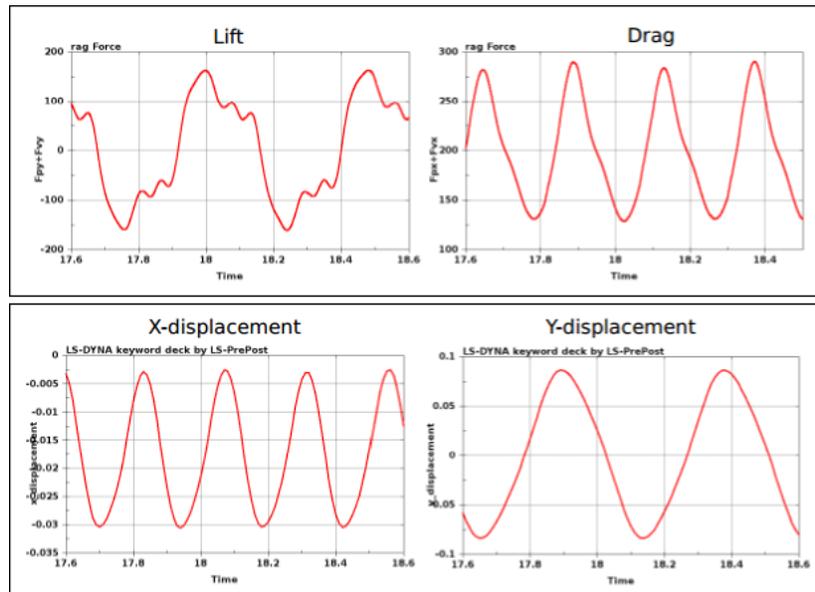


Figure 7 FSI 2 Lift Drag and tip displacements

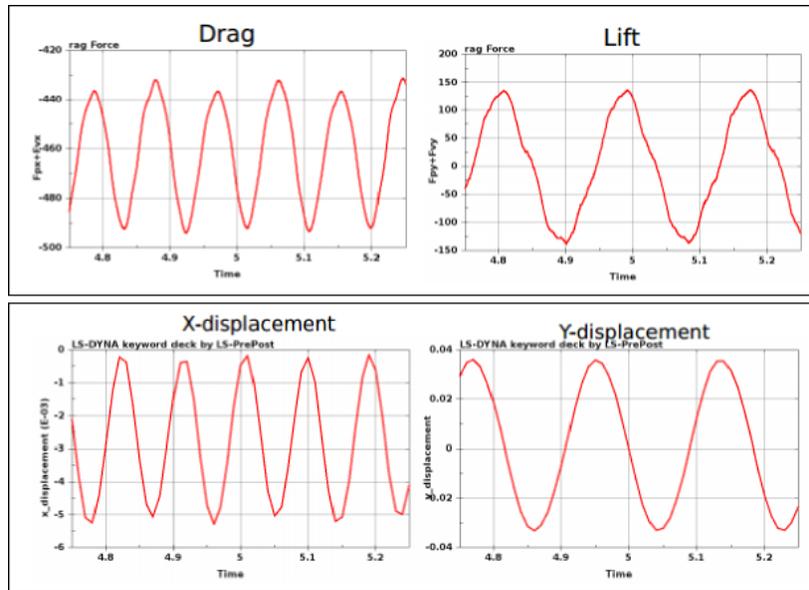


Figure 8 FSI 3 Lift, Drag and tip displacements

Table 2 Comparison between current analysis and reference results by [7]

	X-displacements ( $\times 10e-3$ )	Y-displacements ( $\times 10e-3$ )	Lift	Drag
<b>FSI 1</b>				
Present analysis	-	-	1.117	14.22
Reference results	0.0227	0.8209	0.7638	14.295
<b>FSI 2</b>				
Present analysis	-16.54 $\pm$ 14.00	1.45 $\pm$ 85.11	1.33 $\pm$ 161.9	209.35 $\pm$ 80.91
Reference results	-14.58 $\pm$ 12.44	1.23 $\pm$ 80.60	0.88 $\pm$ 234.2	208.83 $\pm$ 73.75
<b>FSI 3</b>				
Present analysis	-2.76 $\pm$ 2.64	1.44 $\pm$ 34.9	3.45 $\pm$ 143.5	464 $\pm$ 33
Reference results	-2.69 $\pm$ 2.53	1.48 $\pm$ 34.38	2.22 $\pm$ 149.78	457.3 $\pm$ 22.66

## 5- Further application examples

### 5.1- Non-inertial reference frames for wind turbine applications

One of the features of the solver is to define a rotating non-inertial reference frame. The reference frame considered here is rotating at a constant speed and thus it is undergoing acceleration with respect to the stationary frame. This rotating frame is then called a non-inertial reference frame. Potential applications include wind turbines, turbomachinery and every other rotating problem that admits a steady state solution.

This feature was used in combination with the strong coupling FSI feature in order to study the deformation and torque of the wind turbine blades. The geometry used was similar to the one by [8] and the results were consistent and of the same order of magnitude (See Figure 9).

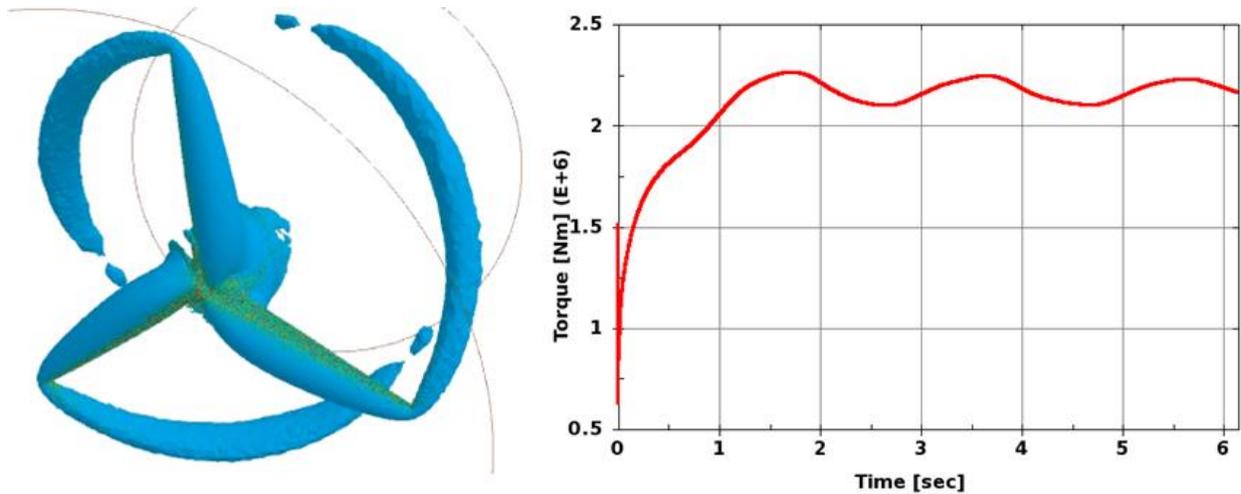


Figure 9 Pressure isosurface in the non-inertial reference frame and torque applied to the blades

## 5.2- Heart valve opening and closing

Another important application for the strong FSI coupling feature are heart valves and hemomechanics. The example shown on Figure 10 is currently under investigation in collaboration with Hossein Mohammadi of McGill University. It features an artificial heart valve with three leaflets. The three leaflets are initially shut but then open under the blood pressure. They are then forced shut again but a very strong counter pressure (See Figure 11). The complexity of this lies in the very low Young's modulus of the leaflets, their low density compared to the blood's density and their strong and rapid displacements which implies strong re-meshing capabilities. The blood could potentially be represented by a non-Newtonian material by using the Power law model for Non-Newtonian fluids included in the solver.

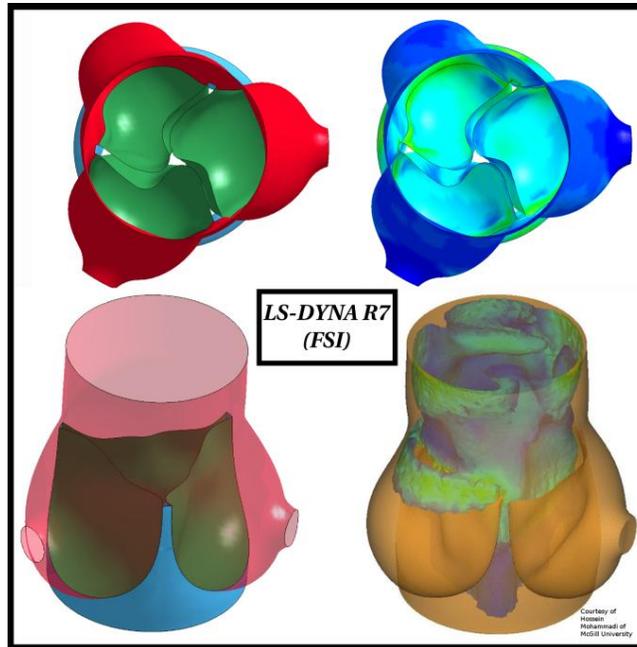


Figure 10 Heart valve: Von Mises stresses and isosurfaces of pressure

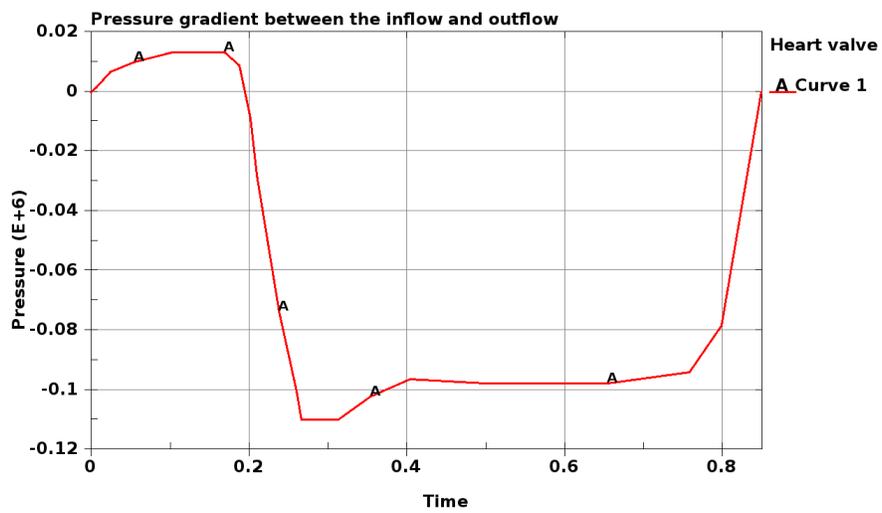


Figure 11 Blood pressure gradient applied between the inflow and the outflow of the arterial heart valve

## References

- [1] N. M, T. S, W. W et R. E, «Robustness and efficiency aspects for computational fluid structure interaction,» *Computational Science and High Performance Computing II*, pp. 99-114, 2006.
- [2] P. S et F. C, «Partitioned procedures for the transient solution of coupled aeroelastic problems,» *Comp. Meth Appl. Meth. Eng.*, vol. 190, pp. 3147-3170, 2001.
- [3] F. C, C. K et F. C, «Partitionaed analysis of coupled mechanical systems,» *Comp. Meth. Appl. Mech. Eng.*, vol. 190, pp. 3247-3270, 2001.
- [4] D. W et P. D, «A computational framework for fluid-structure interaction element formulation and applications,» *Comp. Meth Appl. Mech. Eng.*, vol. 195, pp. 5754-5779, 2006.
- [5] C. P, F. G. J et N. F, «Added-mass effect in the design of partitioned algorithms for fluid structure problems,» *Comp. Meth. Appl. Mech. Eng.*, vol. 194, pp. 4506-4527, 2005.
- [6] Z. O et Z. J, «A simple error estimator and adaptive procedure for practical engineering analysis,» *International Journal for Numerical Methods in Engineering*, vol. 24, pp. 337-357, 1987.
- [7] T. S et H. J, «Proposal for Numerical Benchmarking of Fluid Structure Interaction between an elastic object and a laminar incompressible flow,» *bungartz & m. schaefer vol 53*, 2006.