Introducing a System Coupling Solution including LS-DYNA and Fluent

David Aspenberg[1], Isheng Yeh[1], Oleg Chernukhin[1]

[1] Ansys, part of Synopsys

1 Introduction

System Coupling is a software that has been around for many years, connecting multiple Ansys software to create multi-physics solutions. Although LS-DYNA incorporates several multi-physics solutions within a single code, it would sometimes be beneficial to be able to access more advanced solver features from other specialized software.

The coupling between LS-DYNA and Fluent was first presented to the public in November 2023 in a beta state [1]. Since then, graphical user interface support has been improved and some initial software bugs have been fixed, which has led to this coupling soon being officially released.

As a first step in this development, the coupling exchanges force and displacement data. The workflow to accomplish this coupling within System Coupling's graphical user interface is demonstrated on a benchmark model, highlighting the steps that need to be performed on each solver side. The solution is also validated using a benchmark model.

2 About System Coupling

System Coupling is a software that aims at connecting different simulation software within, but also outside of, the Ansys package. It manages data exchange between the different solvers and coordinates independent solver executions. It also includes features like convergence checking tools, solution stabilization methods, HPC deployment, etc.

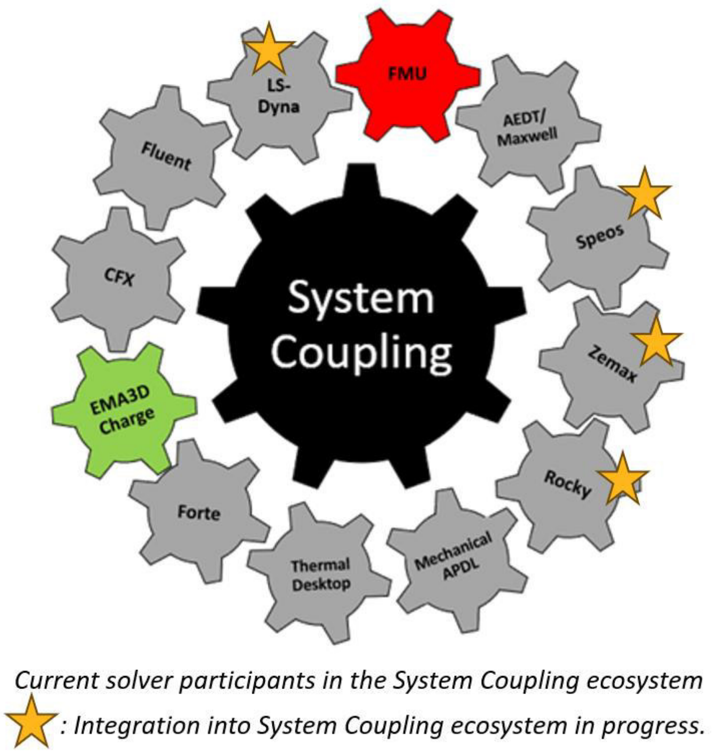

Fig.1: Software that are currently integrated into the System Coupling ecosystem.

System Coupling is included in the Ansys software installation. It does not require a license of its own, but it does check that you have at least one type of Ansys license installed.

There are several ways of executing a System Coupling run:
- Interactively by working in the System Coupling stand-alone GUI.
- Batch execution using a command line.
- Through the integration in Workbench.
- Using PySystemCoupling.

During this development, the focus has been on getting things to work properly in the System Coupling stand-alone GUI, whereas the other running modes have not been explored so far.

Prior to release, coupling to LS-DYNA is available as a hidden alpha feature in System Coupling. It is advised to use as recent version as possible due to the recent developments that have been made, preferably at least 25R2 for coupling with implicit LS-DYNA and 26R1 for coupling with explicit LS-DYNA.

3 Fluent/LS-DYNA FSI coupling

In a fluid-structure interaction (FSI), the fluid flow may exert pressure and/or thermal loads on a structure. In this first implementation of Fluent/LS-DYNA coupling, only structural loads are addressed. When the pressure load is large enough to cause significant deformations on the structure, such that the fluid flow itself is altered, it is essential that the pressure loads and deformations are exchanged between the solvers to retrieve a correct solution. This is where we would use a coupled approach. Conversely, a sequential approach where the flow problem is solved first separately, and the pressure loads applied in a subsequent structural simulation would be sufficient for small structural deformations.

For the newly developed Fluent/LS-DYNA coupling, Fluent handles the fluid flow and LS-DYNA the structural deformations. Both 1-way and 2-way coupling types are available. When System Coupling reads the LS-DYNA input file during set-up using the System Coupling GUI, it automatically determines whether the structural solution in LS-DYNA is using the explicit or implicit solver, and this will determine the iterative approach used by System Coupling to resolve the coupling between the two solvers. In the following, it is assumed that Fluent is run with an implicit solution scheme.

If LS-DYNA is using the implicit solver, a 2-way iterative coupling approach as illustrated in Fig. 2 is automatically suggested. For each coupling time step, several coupling iterations are performed in which forces coming from the pressure load is sent from Fluent to LS-DYNA and a displacement is sent from LS-DYNA to Fluent. Within each coupling loop, the different solvers independently seek convergence in the solver loop, in which absolute convergence is not an absolute requirement until the last coupling iteration. If different time step sizes are required on each solver side, it is advised to pick the smallest of these as the common time step.

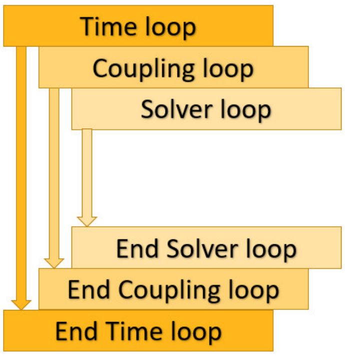

Fig.2: Illustration of the 2-way iterative solving scheme.

Using the 2-way iterative coupling approach enables the use of convergence stabilization and acceleration methods, such as relaxation factors in the data transfer and/or quasi-Newton stabilization schemes, algorithms that are already available within the System Coupling framework.

If LS-DYNA instead is using the explicit solver, it traverses through its (smaller) explicit time steps once at the start of each coupling time step and does not participate in coupling iterations. This could be considered a 1-way workflow in the sense that it does not apply coupling iterations. LS-DYNA uses the forces from the last Fluent step. Structural displacements are still sent to the fluid solver; it is merely the coupling loop that is omitted. This approach is therefore inherently less numerically stable than the fully implicit approach. Furthermore, there are currently no stabilization schemes available to improve stability for this type of coupling. The only possible remedy for instabilities would be to decrease the implicit time step on the fluid side.

4 Coupled simulations set-up and postprocessing

This section covers the extra steps that are required to go from a stand-alone simulation on each solver side to a coupled simulation in System Coupling.

4.1 LS-DYNA input

Three additional keywords are required to the LS-DYNA input deck, namely **`*SET_SEGMENT_N8`**, **`*COSIM_SYC_CONTROL`** and **`*COSIM_SYC_INTERFACE`**. These keywords are all supported in LS-PrePost 4.13, and the input format is shown in Fig 3.

```
*SET_SEGMENT_N8
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$#       sid
         100
$#        n1        n2        n3        n4        n5        n6        n7        n8
         395       397       394       393       838       836       834       835
         361       394       397       427       771       836       840       772
...
*COSIM_SYC_CONTROL
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$#             sycid       opt
              lsdyna         C
*COSIM_SYC_INTERFACE
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$#             sycid      nreg
              lsdyna         1
$ --- For each coupling region
$#           regname    regtyp     regid      nexp      nimp
              FSIN_1    SETSEG       100         1         1
$ --- For each variable to be exported to coupling participant
$#           varname   dynatyp    syctyp     scext     scloc      scvs
                INCD      INCD      INCD
$ --- For each variable to be imported to ls-dyna
$#           varname   dynatyp    syctyp     scext     scloc      scvs
                FORC     FORCE     FORCE
```

Fig.3: The additional LS-DYNA keywords required for a coupled analysis using System Coupling. This example contains only one coupled interface.

The first keyword **`*SET_SEGMENT_N8`** defines which regions in the model are to be coupled to the Fluent side. For linear elements, the segments are described by three or four nodes, whereas second order elements require six or eight nodes. This differs from how segment sets are normally created in LS-DYNA, where only corner nodes are given. However, both the generation and visualization of **`*SET_SEGMENT_N8`** sets can be made in LS-PrePost 4.13, see Fig. 4.

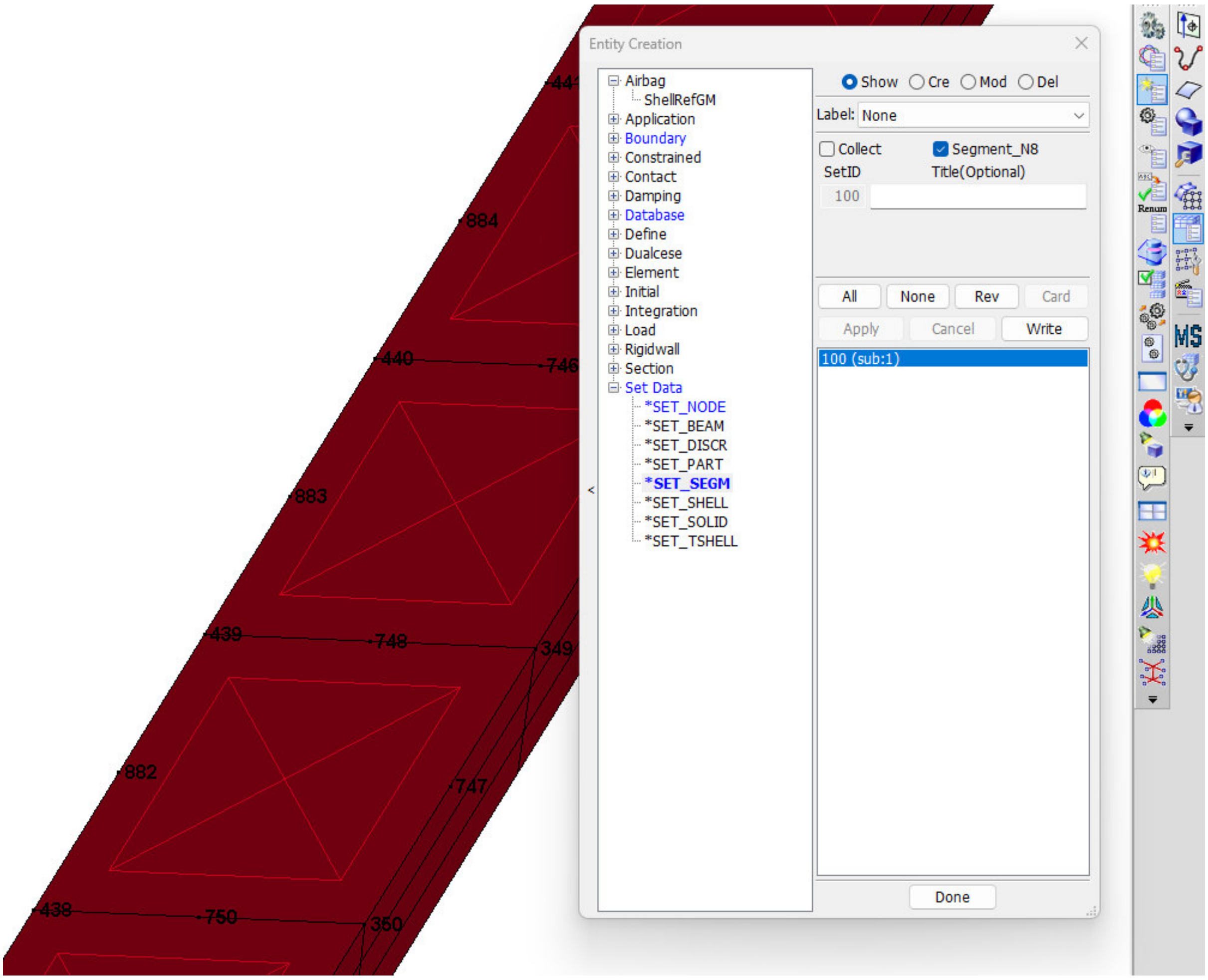

*Fig.4: Segment set handling for *SET_SEGMENT_N8 in LS-PrePost 4.13.*

The second keyword **`*COSIM_SYC_CONTROL`** defines the coupling and the general co-simulation settings.

Finally, **`*COSIM_SYC_INTERFACE`** specifies the variables to be exchanged for the different coupling regions, which in this initial implementation are always forces and displacements.

It is recommended to use an LS-DYNA R16.1 (or later) solver version.

4.2 Fluent input

On the Fluent side, one must add Dynamic Meshing to allow the fluid mesh to deform as the structure deforms. The interfaces that are coupled to LS-DYNA and modified by System Coupling are selected in Fluent and assigned a System Coupling Dynamic Mesh Zone type, see Fig. 5. This essentially means that System Coupling will control the deformation of these interfaces, information coming from the deformations calculated in LS-DYNA.

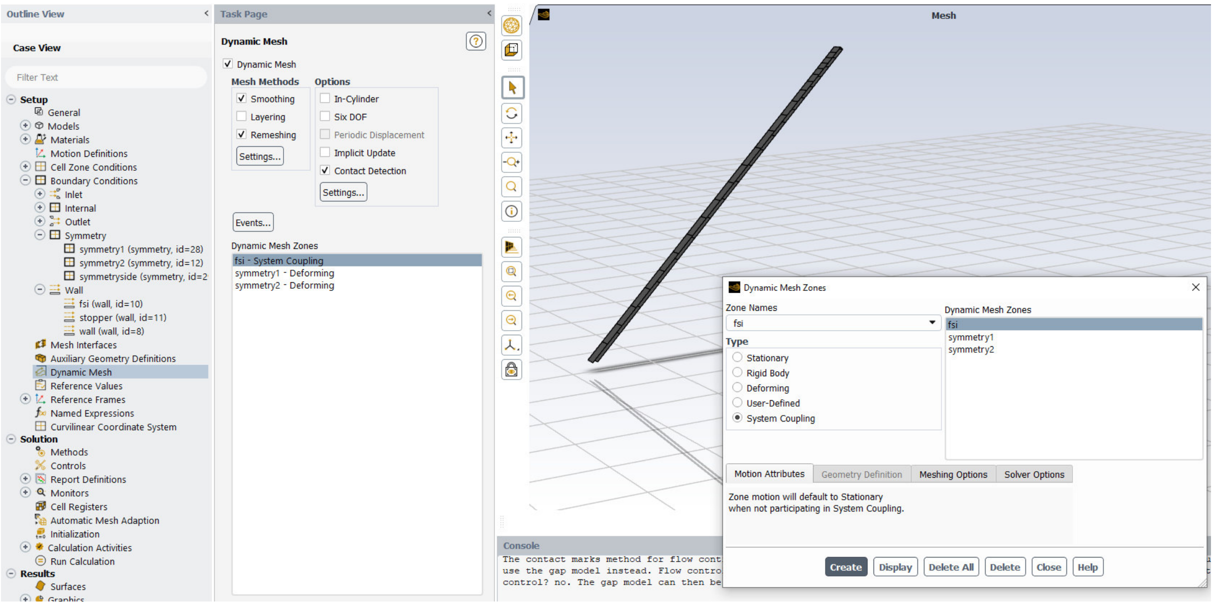

Fig.5: Definition of the coupled interfaces in Fluent.

In the image above (cf. the valve example presented below) the deformation of the coupling interfaces will also affect the mesh on the adjacent symmetry planes. Therefore, these symmetry planes must also have a Dynamic Mesh definition, but of a Deforming type.

In Fluent, you also need to output a System Coupling Participant setup file (.scp file). These files are XML-formatted and provide System Coupling with the participant-specific information needed to connect the participants to a coupling run. Note that this step is not required on the LS-DYNA setup side.

All recent versions of Fluent should work with this coupling.

4.3 Setting up the solution in System Coupling GUI

To set up the coupled simulation run in the System Coupling GUI, the current workflow follows these steps:

1. Add the two participants by pointing at the keyword file for LS-DYNA and the .scp file for Fluent.
2. Add a coupling interface and select e.g. Fluent as side one and LS-DYNA as side two along with their respective defined coupling regions.
3. Add FSI data transfer.
4. Select the LS-DYNA executable to be used.
5. Fill in End Time, Time Step Size, and Minimum and Maximum Iterations to fit your needs.

The rest is automatically set, but with many options for modifications if needed. In current releases, it is also necessary to activate the hidden alpha features in the System Coupling GUI.

4.4 Postprocessing interface results

The postprocessing of the coupled simulation can be performed in a regular fashion on the respective sides of the coupled run. That is, structural post-processing can be done in e.g. LS-PrePost and Fluent results in the Fluent GUI.

For the transferred forces and displacements, the System Coupling GUI provides a direct link to Ansys EnSight, which is a general-purpose postprocessing software. As an example, a vector plot of the transferred nodal forces applied on the LS-DYNA side is shown in Fig. 6.

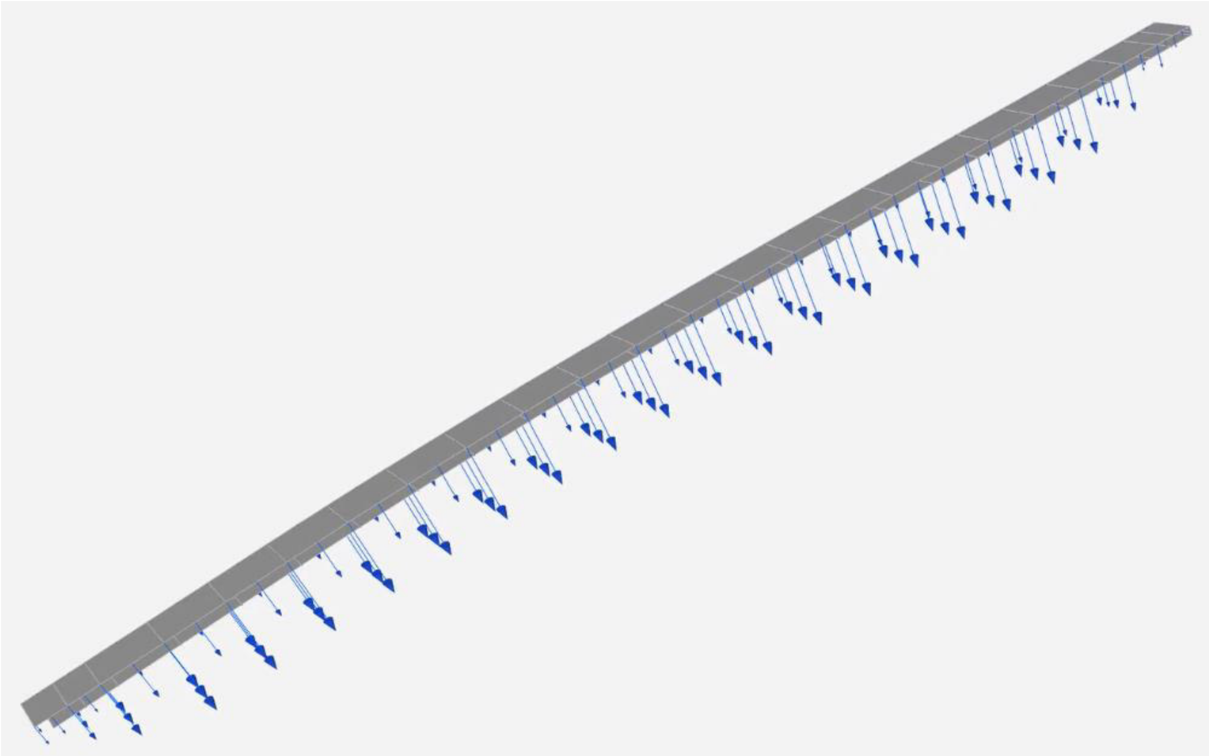

Fig.6: A vector plot of the transferred forces from Fluent to LS-DYNA.

5 Reed valve example

To demonstrate the coupling, a simple reed valve example has been used. The fluid domain modelled in Fluent has a pressure inlet boundary condition is applied in one end and a zero gauge pressure outlet in the other end. The deformable reed is modelled in LS-DYNA with an elastic material and using higher order solid elements, cf. Fig. 7.

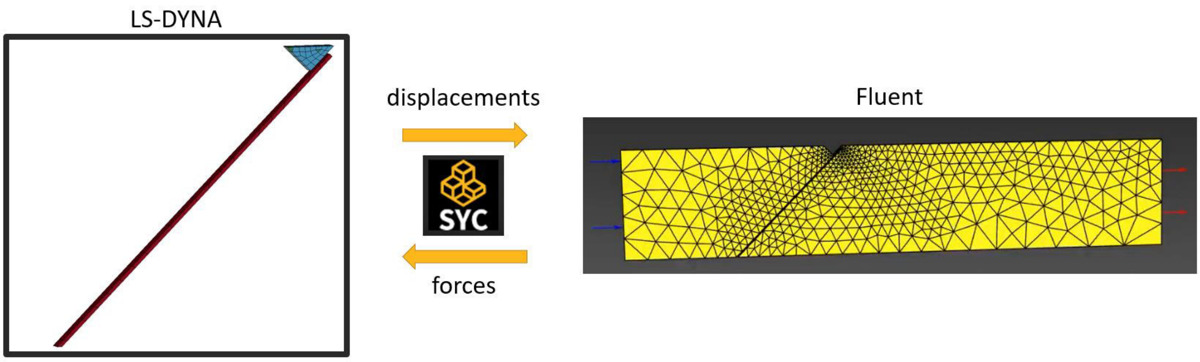

Fig.7: Decomposition of the reed valve example.

This example was run using both the explicit and the implicit solver for LS-DYNA, i.e. testing both the 1-way and 2-way coupling approaches. Termination time was set to 10 ms and the implicit time step to 0.1 ms. In the explicit version, a time step of 2.8 ns was used, but mass scaling could have been applied to increase the time step and decrease the simulation time. A comparison of the horizontal tip displacements is shown in Fig. 8.

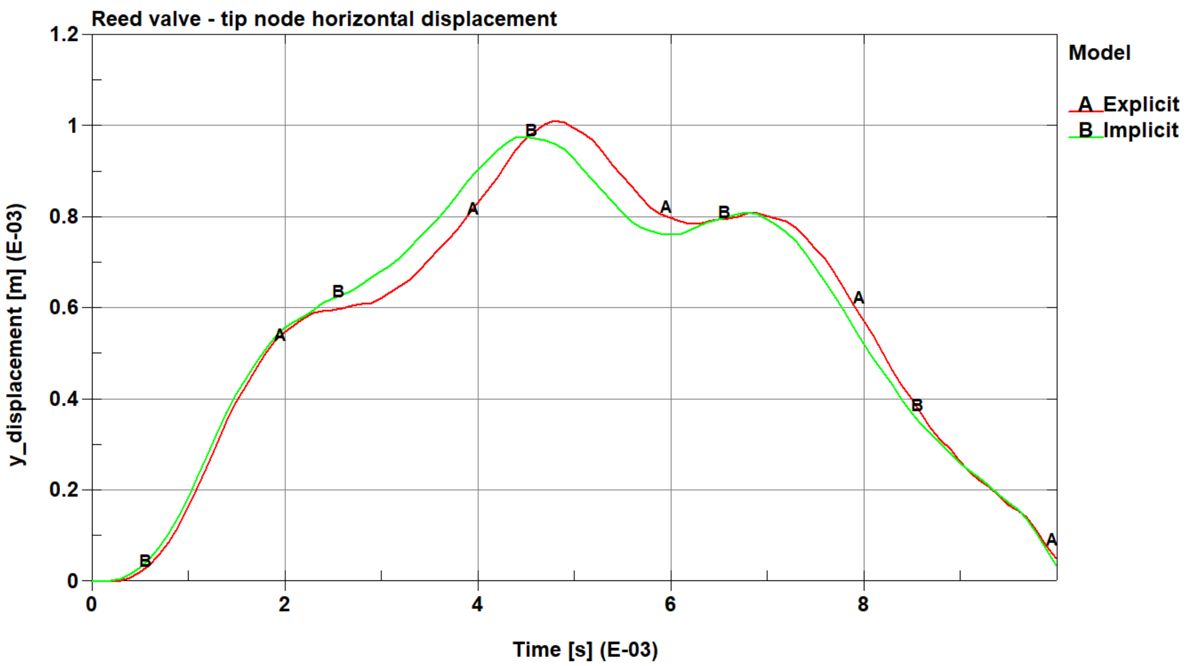

Fig.8: Horizontal tip displacement comparison between the explicit and implicit solutions.

6 Turek-Hron FSI benchmark example

To validate the coupling, a popular benchmark problem for FSI problems was also run, the so called Turek-Hron benchmark problem [2]. This problem is unstable by nature; the flexible structure deforms significantly when subjected to the pressure loads from the fluid, resulting in self-induced oscillations. This problem can pose some challenges from a numerical stability perspective, particularly when using a more elastic material or when the fluid and solid densities are assigned with similar magnitudes. The parameters from the setup named FSI2 in the paper have been straightforward adopted, see Fig. 9 and Table 1.

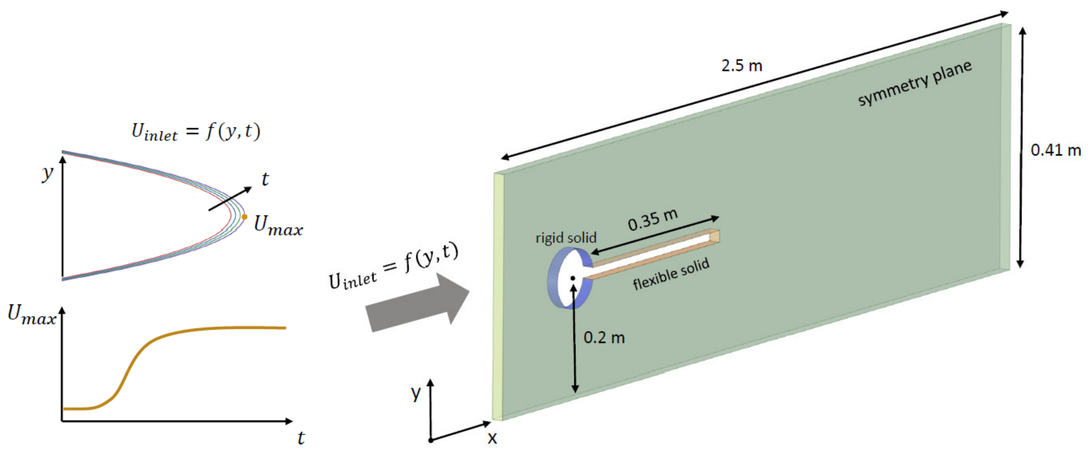

Fig.9: Geometry and flow profile for the Turek-Hron FSI benchmark example.

Parameter		**Value according to FSI2 setup**
Solid	Density, ρ^s $\left[10^3\ \frac{kg}{m^3}\right]$	10
	Poisson's ratio, ν^s	0.4
	Lamés second constant (shear modulus), μ^s $\left[10^6\ \frac{kg}{ms^2}\right]$	0.5
Fluid	Density, ρ^f $\left[10^3\ \frac{kg}{m^3}\right]$	1
	Kinematic viscosity, ν^f $\left[10^{-3}\ \frac{m^2}{s}\right]$	1
	Mean inflow velocity, $\bar{U}$ $\left[\frac{m}{s}\right]$	1

Table 1: Model parameters applied in the Turek-Hron benchmark example.

The 2-way iterative coupling approach was applied in this example. An implicit time step of 0.01 s was used. To aid convergence, a relaxation factor of 0.5 on the force transfer was applied, without which the solution became unstable. No other stabilizations were required. The simulation was run for 14 s and results were in good agreement with the results from the original paper, cf. Fig 10-12 and Table 2. It was also attempted to solve this example by using the explicit solver of LS-DYNA with an unchanged time step on the Fluent side, but this solution approach was found unstable.

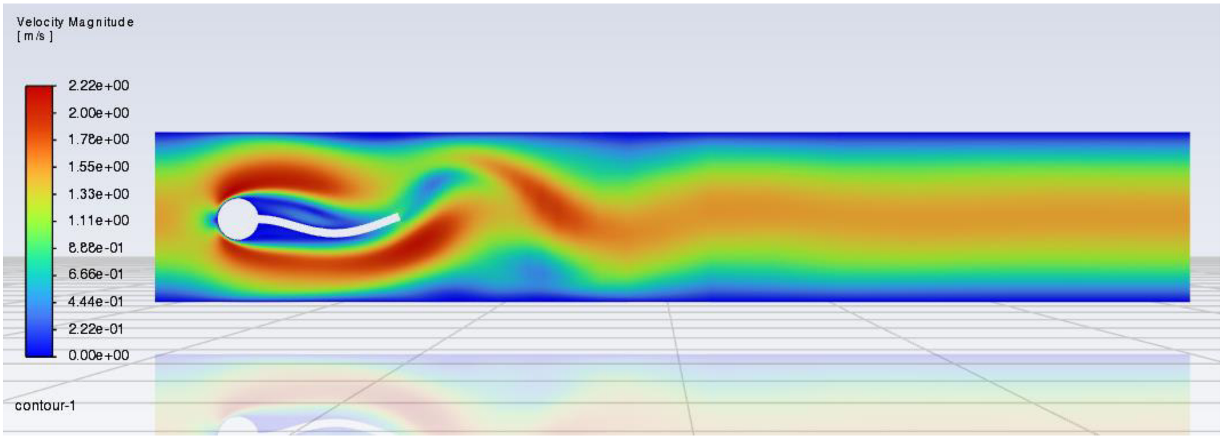

Fig.10: Velocity field of fluid flow in Turek-Hron benchmark example.

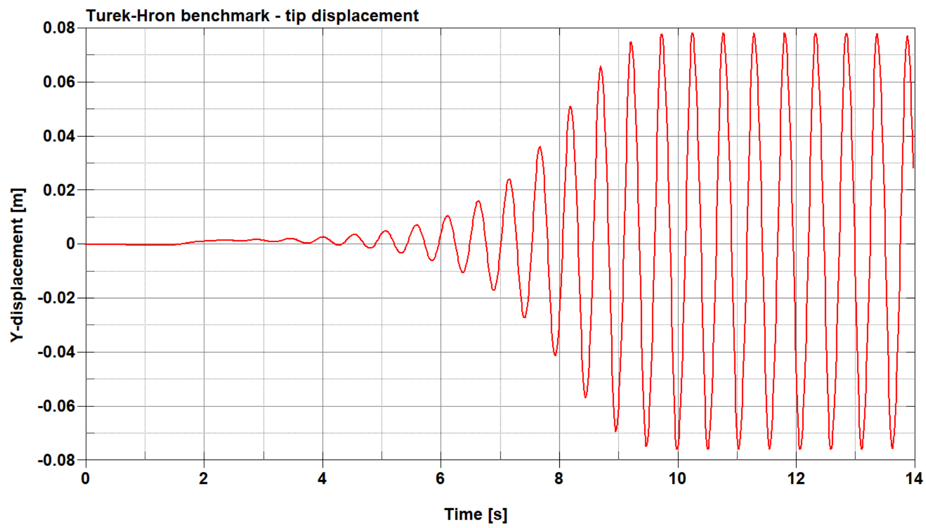

Fig.11: Tip vertical displacement of the flexible profile.

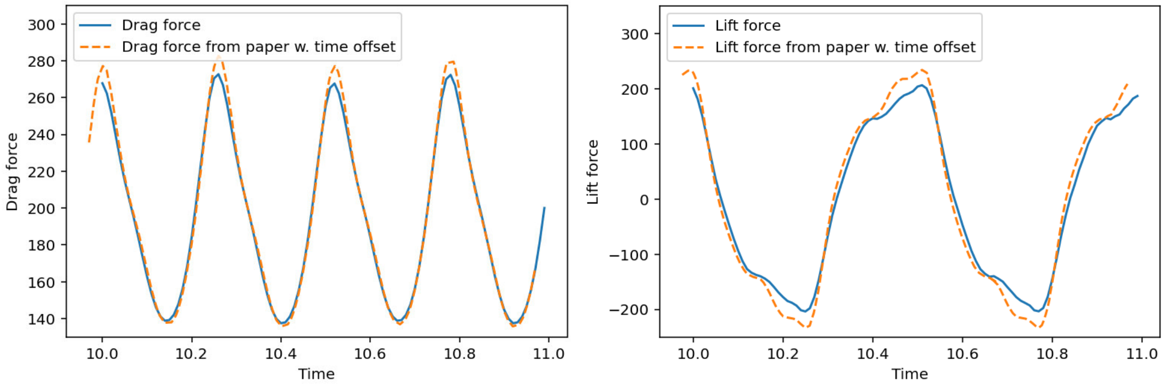

Fig.12: Drag and lift forces comparisons between the simulation results and the results in the original paper (curves from paper offset in time).

Model	Amplitude y-disp	Amplitude x-disp
Turek-Hron paper (#refine 2)	79.2 mm	12.0 mm
Fluent + LS-DYNA	77.1 mm	11.9 mm

Table 2: Amplitude comparisons for end tip displacements.

7 Future work

A current limitation is that System Coupling is not allowed to control the time step in LS-DYNA. In the 2-way coupling, one must manually make sure that the time steps are equal on both sides. A fix for this has already been implemented in the development version of LS-DYNA but is yet to be tested. When tested and validated, this limitation will be lifted in future LS-DYNA releases.

Requests for adding thermal coupling have already been received, and the development of this coupling type has already started.

8 Summary

A new workflow that connects Fluent and LS-DYNA for FSI simulations has been presented. Since the original presentation of this coupling, several things have been improved. In addition to regular bug fixes, support for LS-DYNA preprocessing in LS-PrePost has been added, as well as improved support for LS-DYNA participants in the System Coupling GUI. The solution has now also been validated with a well-known benchmark example.

9 Literature

[1] I. Yeh: "Co-simulation in LS-DYNA FMU and SyC", 2023 North American LS-DYNA User Forum.

[2] S. Turek and J. Hron: "Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow", Fluid-Structure Interaction, Springer Lecture Notes in Computational Science and Engineering, Vol. 53, Ed. H.-J. Bungartz & M. Schaefer, Springer Verlag 2006.