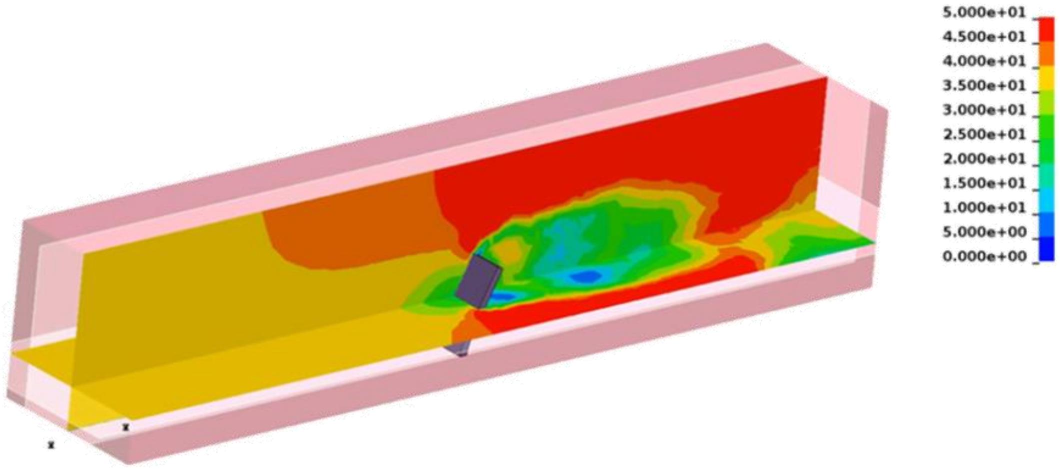
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 9. FSI with the ICFD solver



Contents

1 Introduction

With Fluid Structure Interaction, we mean the interaction of a movable or deformable structure with a fluid (internal and/or external fluid). In this tutorial, you will first setup a fluid simulation for the Incompressible Computational Fluid Dynamics (ICFD) solver in LS-DYNA and then add a deformable structure and let it interact with fluid by activating the FSI-coupling in the ICFD-solver. The simulation will be run as a transient dynamic Variational Multiscale Model (VMM) that is similar to a Large Eddy Simulation (LES) but the filtering of scales occurs in the discretization.

The ICFD solver is available from LS-DYNA R7, but we recommend that you use the latest version (or at least version R9) as the solver performance in the later versions is much improved. The ICFD solver must always be run using the double precision version of LS-DYNA. The ICFD solver uses a fully implicit time-stepping and solution method. The ICFD solver is an incompressible CFD solver based on the Finite Element Method (FEM). Modern FEM based ICFD-solvers of the type used in LS-DYNA are a fairly recent addition to the CFD-solver market and most other CFD solvers on the market today are based on the Finite Difference Method (FDM) or Finite Volume Method (FVM).

This tutorial will only show how the fluid and the FSI are set up. The setup of the structure/solid part will not be demonstrated, but the input files for the solid are provided for the user to take a look at. In this tutorial, the FSI is set to use a, so called strong, two-way connection, which means that both the fluid and solid solver use the same time step and a Newton iteration is performed to ascertain that force equilibrium is obtained between fluid and structure in each time step. This requires that the solid is solved with the implicit solver since the ICFD solver is an implicit solver.

If you want to learn more about a specific keyword or if you feel that some information is missing in this tutorial regarding a keyword look in the LS-DYNA keyword manual vol. 3.

1.1 Prerequisites

- Basic knowledge about LS-DYNA and LS-PrePost.
- Basic understanding of FEM and CFD.
- A computer with at least 4 CPUs (running on 1 CPU will lead to rather long run times).
- LS-PrePost 4.3 or later and LS-DYNA R9.1 or later.

2 Problem description

The task is to first simulate the flow over a fixed rigid plate in a box-like domain, and then add FSI so that the plate can deform due to the interaction with the flow. The fluid is water and the material in the plate is soft enough so that it will deform. SI-units are used throughout this tutorial.

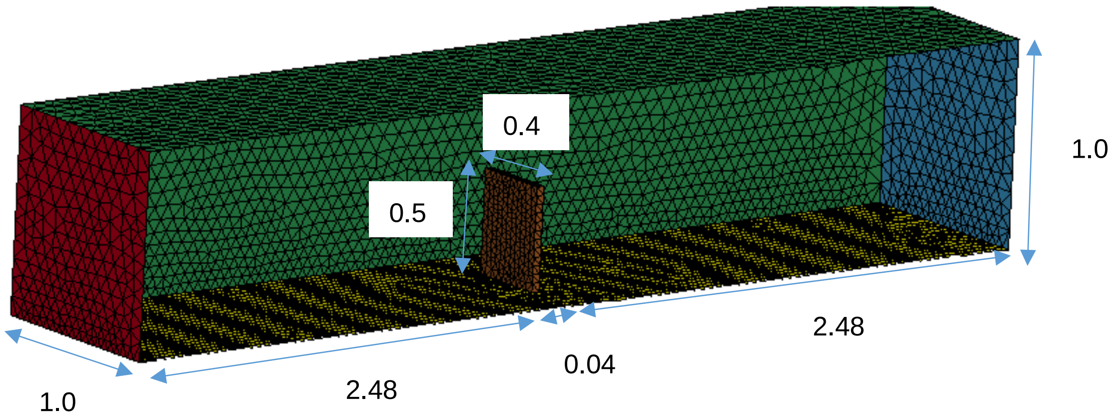


Figure 1 The geometry of the case that will be set up; all dimensions are in meters.

2.1 Files included with the tutorial

- fluid_main.k – contains the control cards for the fluid (ICFD) solver.
- mesh_fluid.k – contains the fluid mesh.
- solid_main.k – contains the structural keywords.
- control_cards_nonlin.key – contains control cards for the structural solver.
- database_cards_dynamic.key – contains output keywords for the structural solver.
- mesh_solid.k – contains the structural mesh.
- main_fsi.k – contains the keywords that controls the FSI connection.
- solid_geometry.step – contains the geometry for the structural part.
- fluid_geometry.step – contains the geometry for the fluid part.

2.2 Setting up a CFD simulation in LS-DYNA

First, we need a mesh for the fluid and in this tutorial you do not have to make it yourself since it is already prepared in the file mesh_fluid.k. If you want to mesh the fluid domain see Section 2.3 and if not go to Section 2.4. It is always good to create a separate folder for all the simulation files so you have them in one place.

2.3 Meshing the fluid domain

To mesh the fluid domain load the file fluid_geometry.step into your preferred meshing tool that supports LS-DYNA. Note: This tutorial uses the automatic volume mesher in the ICFD solver, which creates the volume mesh from an input surface mesh, but if you want it is possible to use a user defined tetrahedral volume mesh, see the keyword manual volume 3 for more information. The surface mesh needs to be watertight and not include any duplicate nodes for the automatic volume mesher to work.

Part 1 will be the inlet, part 2 the outlet, part 4 the ground, part 5 the plate surface, and part 3 the boundary to the surroundings.

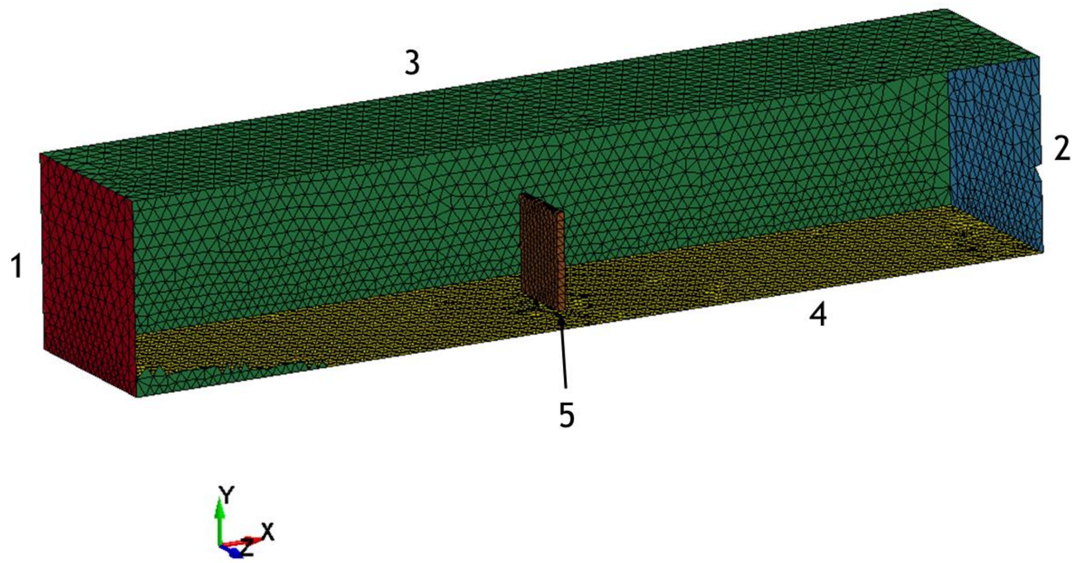


Figure 2 Parts for the boundary conditions, the blanked surface also belongs to part 3.

Create a triangular mesh using Auto Mesher in LS-PrePost. Select surface 1 and use 0.09 as element size. When the mesh for surface 1 has been created, select surface 2, untoggle “Mesh by GPart” and type in Part ID 2 and create the mesh for part 2.

Select the three surfaces for part 3, toggle the “Connect Boundary Nodes” option, type in Part ID 3 and create the mesh for part 3.

Create the mesh on surface 4 in the same way but use a mesh size of 0.05. Use an element size of 0.04 for the five surfaces on part 5.

Make sure the boundary is water tight by doing a duplicate node check and merge any duplicate nodes found.

Running the analysis with this mesh resolution will take about 1 h on 4 cores so if you have more computational resources or can wait a bit more you can use a finer mesh.

When the mesh is defined save it as an LS-DYNA keyword file and open it in LS-PrePost to convert the structural mesh to a ICFD surface mesh. The ICFD solver uses the *MESH_ family of keywords to define the fluid mesh, so the *NODE and *ELEMENT_SHELL keywords need to be changed and also the parts. This can be done with a text editor or with LS-PrePost, in this tutorial LS-PrePost will be used.

1. Open the mesh in LS-PrePost and select Mesh/MSMesh, see Figure 3.
2. In the popup window MSMesh change Starting PID, Starting EID, and Starting NID to 1 in order to keep the part numbering.
3. In the other window press Whole to select all parts.
4. Then press Apply in the MSMesh window and then Accept and Done.

5. The next step is to make sure that you only have keywords in the following groups: ICFD, MESH, Title, and Keyword. If you have some other keywords remove them using the tool Model/Keywrd, it is also here you check if you have other keywords.

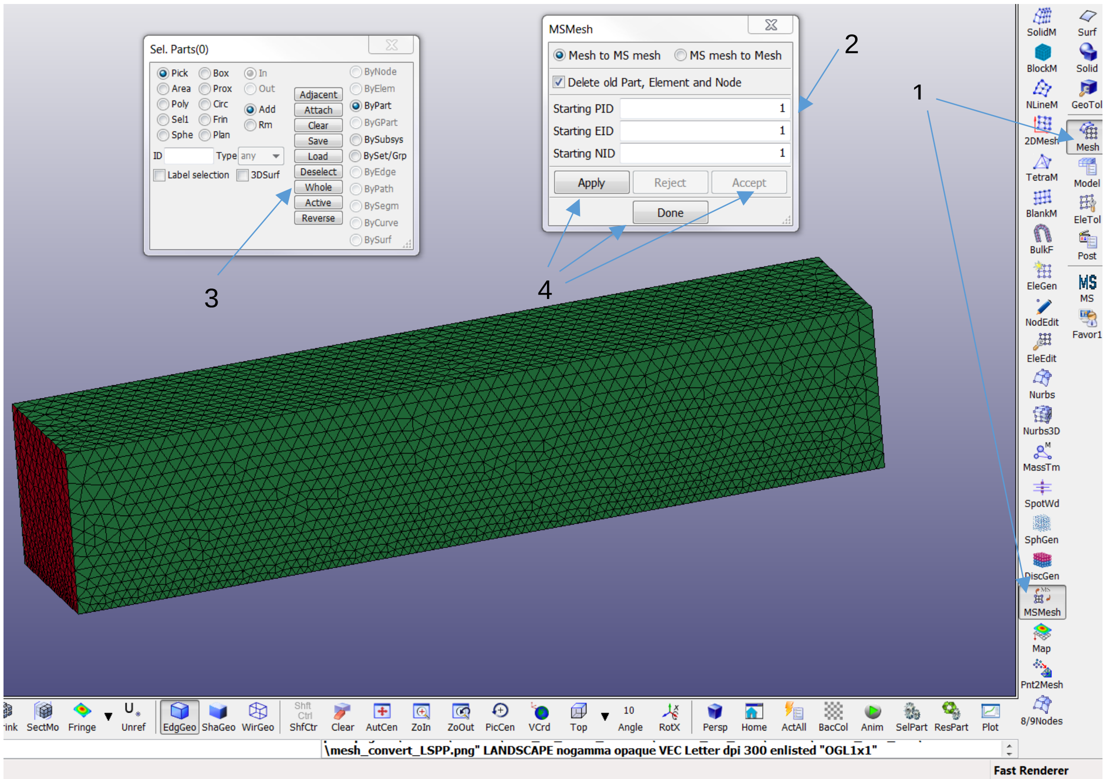


Figure 3 How to convert the mesh to a ICFD mesh.

6. Next change the *ICFD_PART keywords so that each part has the section id and material id set to 1. Do this by going to Model/Keywrd and select the *ICFD keywords and double click on PART see Figure 4.

7. In the new window, see Figure 4, you will see five parts to the right and when selecting each part you will see which section id and material id it has. To set the section id and material id to 1 select part 1 and then change SECID and MID to one and then press Accept. You will get a warning that the section id and material id does not exist, but just press “ignore” since they will be added in the fluid_main.k file. **Note that if you do not press Accept before going to the next part the change will not be saved**. Do this for all parts.

8. Then save the mesh as mesh_fluid.k using File/Save/Save keyword.

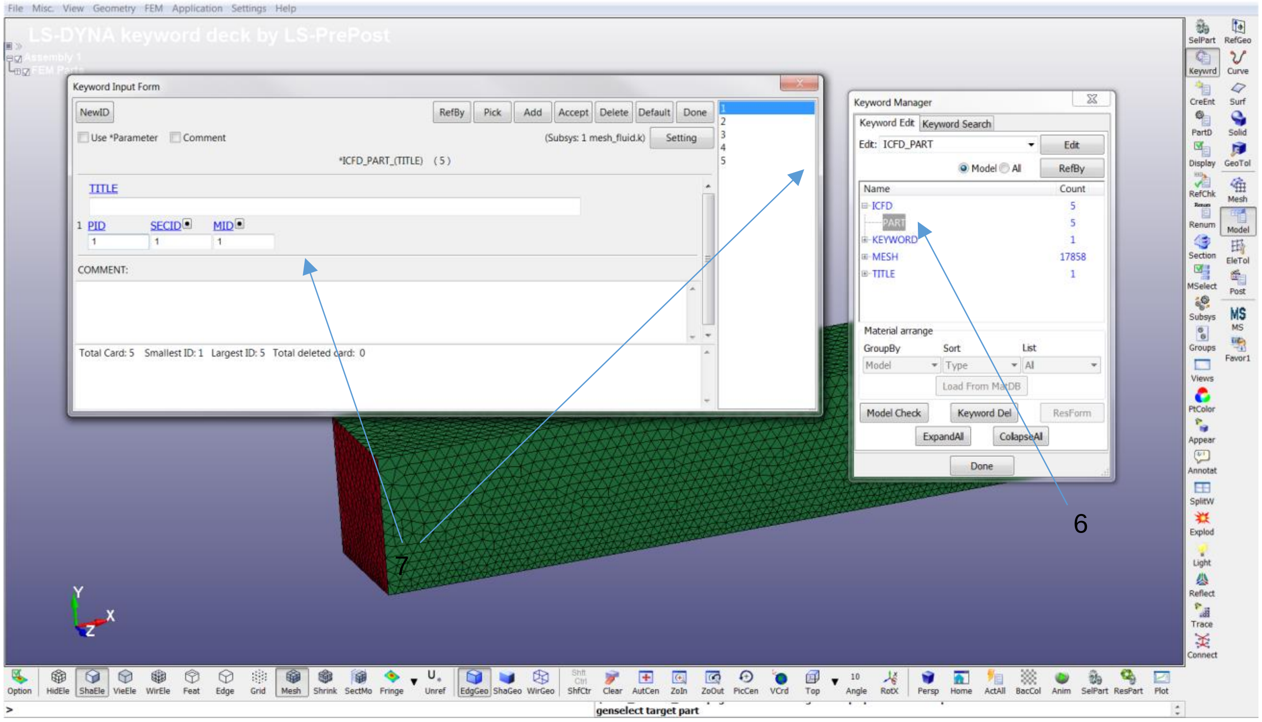


Figure 4 The *ICFD_PART keyword.

2.4 Setting up the CFD simulation

When the mesh is finished (and saved in the file mesh_fluid.k), open a new session of LS-PrePost and then select Model/Keywrd and go to the keyword-group *INCLUDE.

1. Double click on the first keyword *INCLUDE.

2. Press Browse and select the file mesh_fluid.k. Note: you will get the complete path of that file but you can remove everything except the file name if you have all keyword files in the same folder.

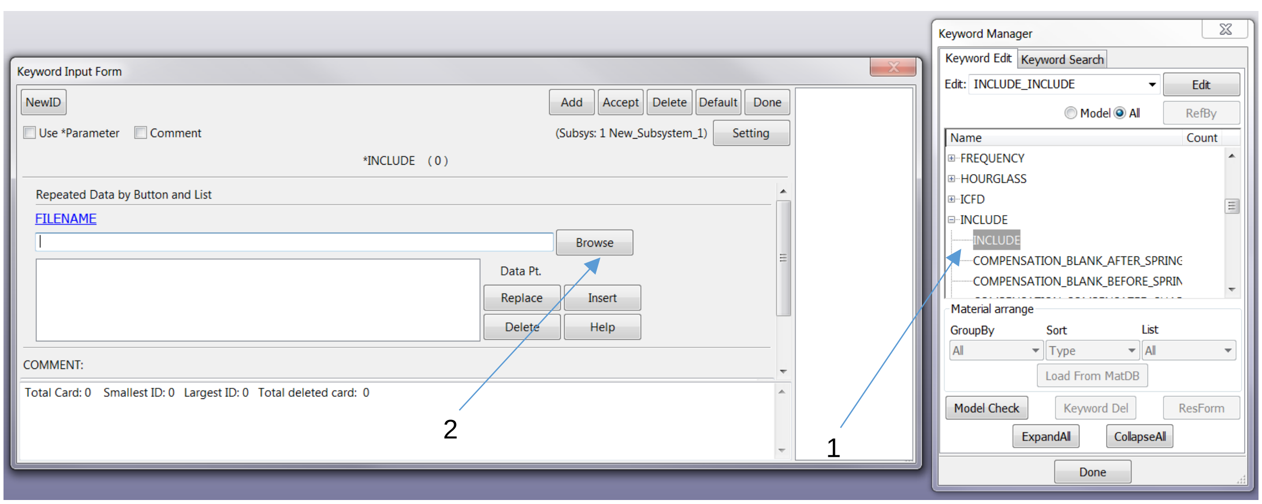


Figure 5 Include keyword.

3. Now save the file as fluid_main.k and open it again, this is only so that we will see the model when we setup the simulation. Close LS-PrePost.
4. Now open the file fluid_main.k again and you will see the model. The first two keywords that we will define are *ICFD_SECTION and *ICFD_MAT. Select Model/Keywrd like in Figure 4, but now you have to select “All” in the Keyword manager otherwise you will only see the keywords in your model. Then go to the *ICFD group and double click on the *ICFD_SECTION keyword and press accept. This keyword must be defined, but it does not do anything in the current version.
5. Now double click on the *ICFD_MAT keyword, this keyword defines the material properties of the fluid. In this example water is used to get a higher force on the plate. RO stands for density and set it to 998.2 kg/m³, VIS is the dynamic viscosity and set it to 0.001005 Pa·s, see Figure 6 below. Then press Accept and Done.

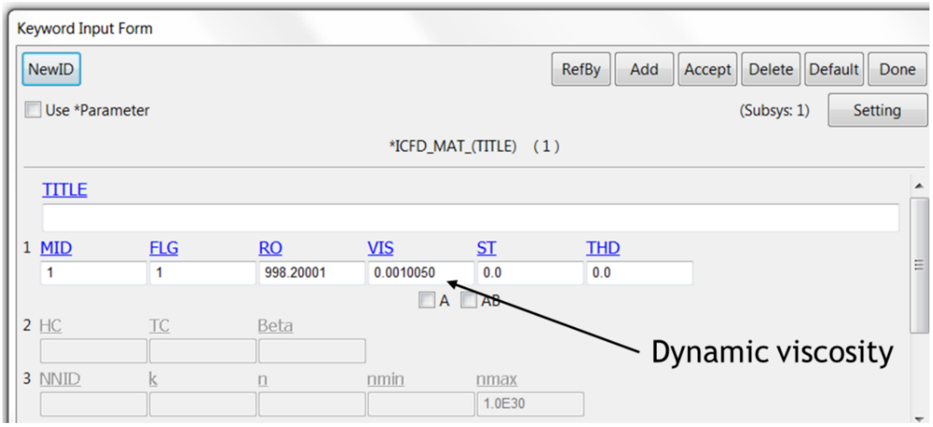


Figure 6 *ICFD_MAT keyword.

6. To associate the fluid properties with the fluid volume, double click on *ICFD_PART_VOL, see Figure 7. Here select the section id and material that you created in the previous step. The PID is the part id of the volume. The SPID parameters should include all surface parts that enclose the volume. In this case all the parts, input all parts and then press Insert. If more than eight parts is used insert another row.

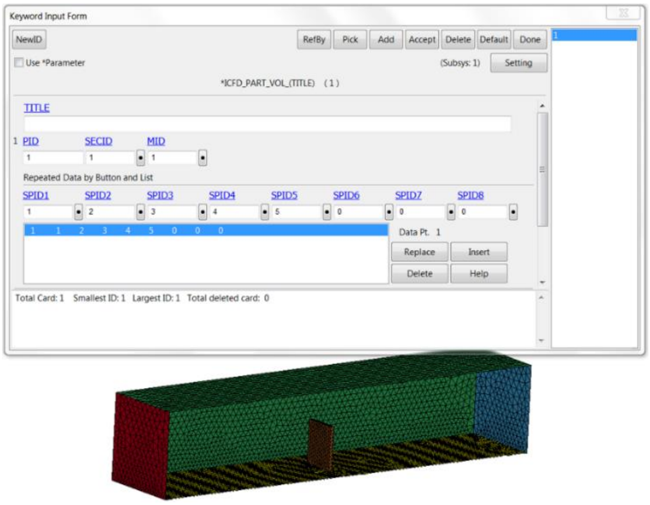


Figure 7 *ICFD_PART_VOL keyword.

2.5 Boundary conditions

In this simulation four different boundary conditions will be used. For the Inlet, a prescribed velocity will be used and for the outlet a zero pressure condition will be used. For the ground and the plate a non-slip condition will be used. For the surroundings, a free-slip condition will be used.

1. Start by defining the load curves that will be used for the inlet and outlet. Go to the keyword group *DEFINE and double click on curve. Press NewID to get a new load curve id, in O1 enter 0 and press Insert, then enter 1 in A1 and change O1 to one and press insert. Now create a third point with A1 100 and keep O1 at one. Then press Accept and if you want you can plot the load curve by pressing Plot. Now press NewID to create another curve, then click on the first row of values and change O1 to zero and then press Replace. Do the same for the other rows of data so that you have a curve that is constant zero then press Accept and Done.

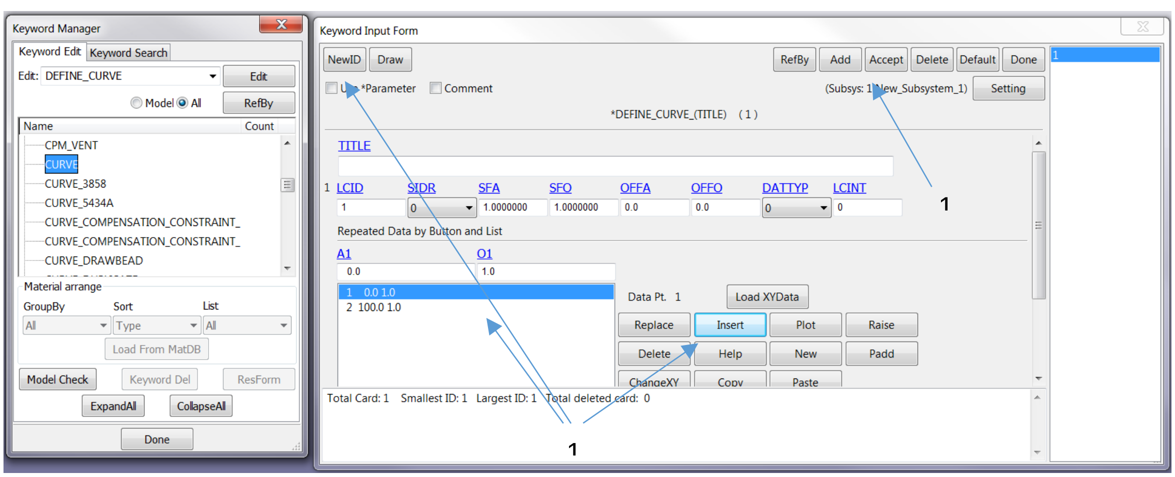


Figure 8 How to define a load curve.

2. To define the boundary condition for the Inlet, double click on *ICFD_BOUNDARY_PRESCRIBED_VEL, see figure 9. Select part 1 which is the inlet, then set DOF to one since the

normal is in the x-direction. Keep VAD at one and then click on the black dot next to LCID to choose load curve (LCID) 1 that has a constant value of one. Then set the scale factor SF to 40 which means that the load curve that is used will be scaled with this value so after the ramp the velocity will be 40 m/s.

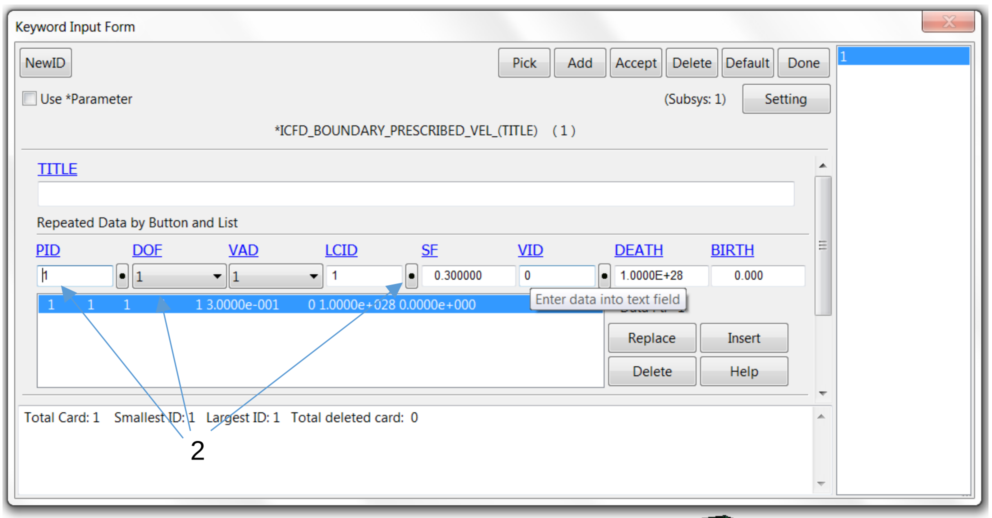


Figure 9 *ICFD_BOUNDARY_PRESCRIBED_VEL keyword.

3. The next boundary condition that should be defined is the outlet, go to the keyword *BOUNDARY_PRESCRIBED_PRE under the *ICFD group in the keyword manager. Set the PID to 2 and set the load curve id to 2, which is the curve with a constant value of zero. Now press Insert and then Accept and Done.

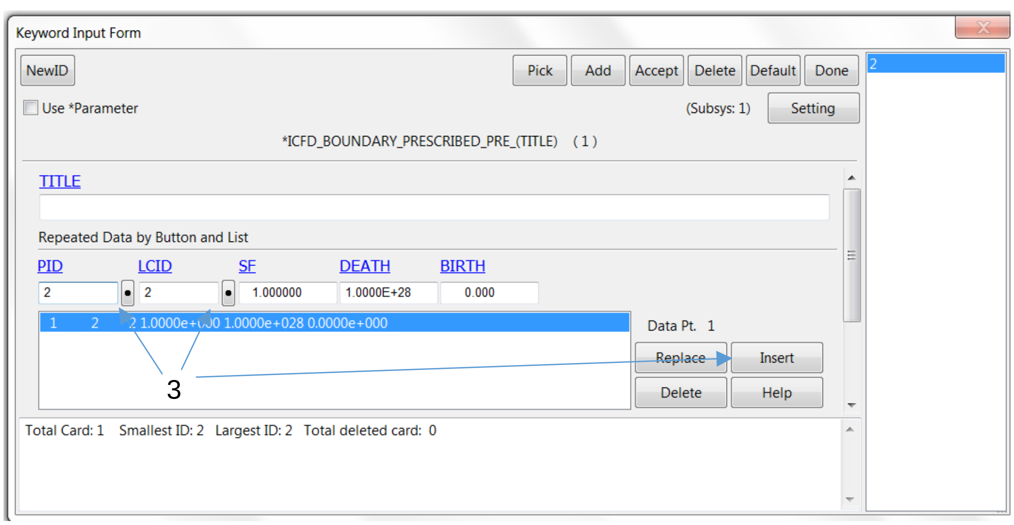


Figure 10 *ICFD_BOUNDARY_PRESCRIBED keyword

4. Now that we have defined the inlet and outlet conditions, we have to define the walls. The first condition is the non-slip condition, which means that the velocity at the wall is zero. It will be applied on parts 4 and 5, which are the ground and plate surfaces. Select the keyword *BOUNDARY_NONSLIP in the keyword manager under the ICFD group, see Figure 11. Select part 4 and

press Insert, do the same for part 5. When both parts have been selected, press Accept and then Done, see Figure 11.

5. The remaining boundary condition is for the surroundings where we will use the free-slip condition, which means that the tangential velocity will be non-zero. Select the keyword *BOUNDARY_FREESLIP in the keyword manager and then select part 3 and press insert, finish with pressing Accept and Done, see Figure 12.

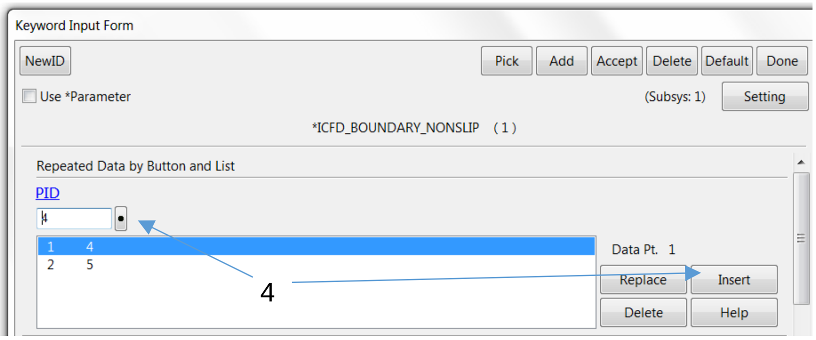


Figure 11 *ICFD_BOUNDARY_NONSLIP keyword.

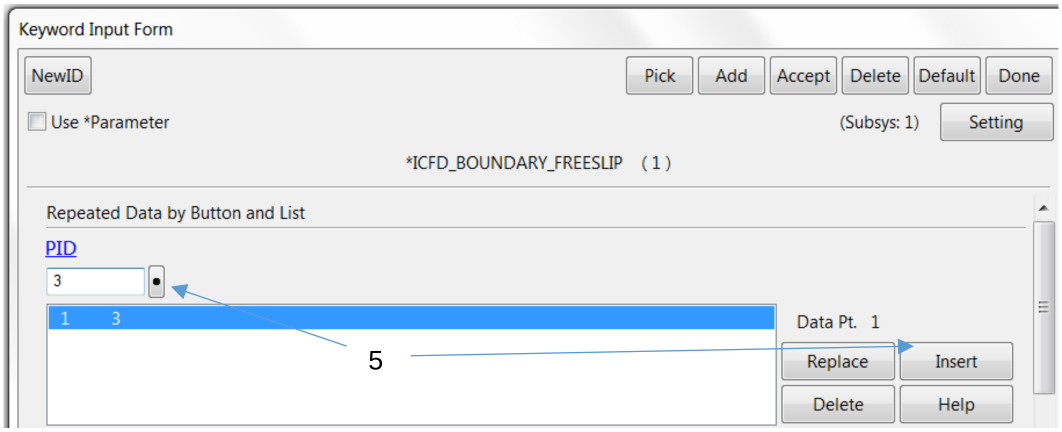


Figure 12 *ICFD_BOUNDARY_FREESLIP keyword

2.6 Control cards and mesh cards.

After all the boundary conditions are defined, the remaining thing to do is to setup the control cards for the solver and the cards that control the volume mesh generation. In this simulation, we will use two control cards, one that controls the time and time step and one that controls the output.

1. The first keyword that we will add is the *ICFD_CONTROL_TIME keyword, double click on it in the keyword manager. Set the parameter TTM, the end time of the simulation, to 1.5 seconds and make sure that DT is set to zero. Setting DT to zero means that an automatic time step

based on the CFL-criterion will be used. With the other parameters, it is possible to scale the time step or limit it to an interval. When done press Accept and Done.

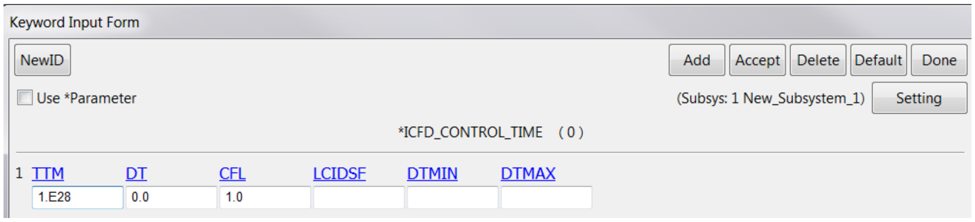


Figure 13 *ICFD_CONTROL_TIME keyword.

2. The next control card that will be added is *CONTROL_OUTPUT that controls the output from the ICFD solver. The MSGL parameter controls how much information that will be printed about the iterations of the solver. The OUTL controls in which file format the solution results will be output and DTOUT controls how often if another format than d3plot is used. LSPPOUT controls if the fluid volume mesh should be output to a separate file. Set MSGL to 4 and leave DTOUT to the default value. If you want another format than d3plot you could set it to 0.01 or a higher value

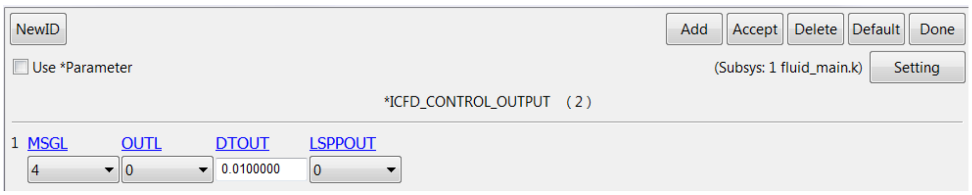


Figure 14 Keyword ICFD_CONTROL_TIME

to save some space. To store the keyword press, Accept and Done.

3. Next add the keyword *DATABASE_BINARY_D3PLOT and set DT to 0.05 in order to get 30 output d3plot states with results. If this keyword is not added you will get the results for each time step.

4. The final keyword is the one that controls the volume mesh generation, called *MESH_VOLUME. This keyword defines the volume that will be meshed based on the defined surface meshes. In this case, we will choose part 1-5 in each PID similar to what we did for ICFD_PART_VOL. To find the keyword go to the group *MESH in the keyword manager. When all five parts have been

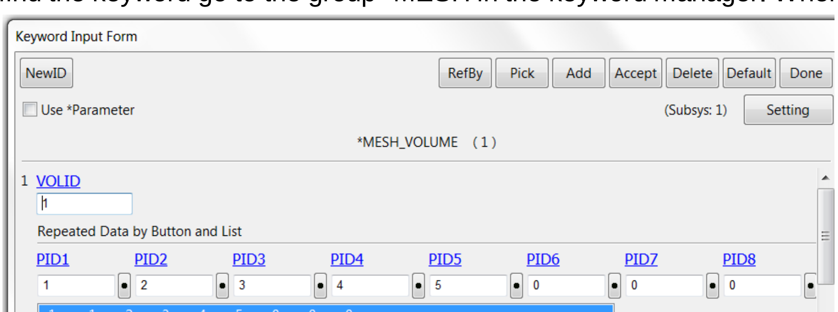


Figure 15 Keyword MESH_VOLUME

selected in a separate PID click on insert and then accept and done.

5. Now save the model by using File/Save/Save Keyword.

3 Run the simulation

ICFD analyses must be run using the double precision version of LS-DYNA. Use the latest version of LS-DYNA available, at least R9.1. You do not need to allocate that much memory using the **memory** command line option of LS-DYNA since the ICFD solver allocates most of the memory required for the solution automatically. The MPP version of LS-DYNA runs much faster than the SMP version if more than about 4-8 cores are used for the simulation.

On Linux, see the **LS-DYNA Keywords User's Manual Vol. I** for information about how to start LS-DYNA.

On Windows, run the simulation using **LS-Run**, see exercise **1. Getting Started** for more information on how to do this. To use the double precision version, make sure to use the "SMP double" or "MPP double" **Preset** in LS-Run.

4 Post-processing the results

When the simulation is finished or after a while when the simulation has progressed so far that the solver has output some results from the simulation you can open the d3plot file in LS-PrePost. In the following section, we will look at the basic post-processing tools in LS-PrePost 4.3. Starting in LS-PrePost version 4.2 a specific post-processing tool is available for CFD results called MS-Post.

4.1 MS-Post

When opening a d3plot file containing ICFD results the ICFD menu will automatically be opened. If you have closed it and need to open it again, click on the MS button and then on the MS-Post button.

4.2 The ICFD window

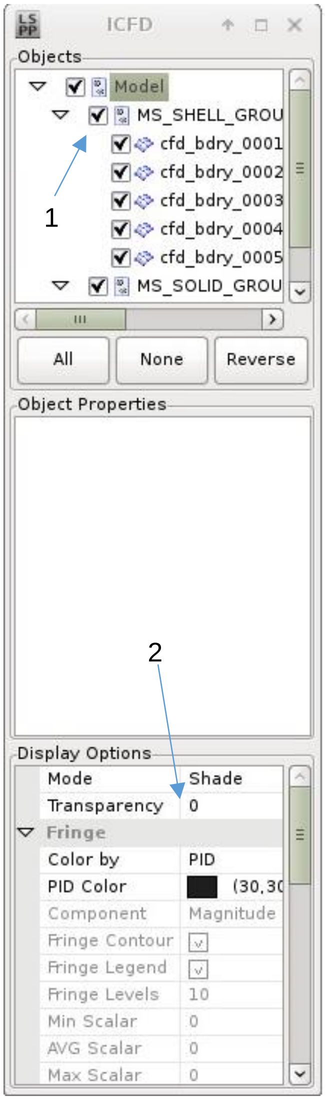


Figure 16 ICFD menu

When opening the d3plot-file you will see this window to the left, the ICFD menu. From ICFD menu everything is controlled when it comes to post-processing objects.

The first part of the window is called the Objects manager, it contains the Objects, it is here you add and remove different objects. To add an object, right click on the object that it should apply to.

1. One can also control the visibility of the objects, start by unchecking the MS_SHELL_GROUP that contains all the shell parts. Now you will see the fluid volume, you could press the arrow before the MS_SHELL_GROUP to minimize it.

   The Object Properties window contains the options for the selected operation. No options are available for MS_SHELL and MS_SOLID parts – only for the operations created on them.

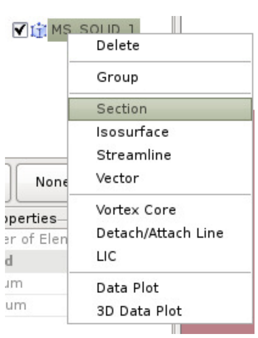
   For example, a section plane has two options in this window, the position of the plane and the normal to the plane.

   The next window is the Display options, here you have all the visualizations options for the selected Object in the Objects manager.

2. In Figure 16 the model is selected. Try to change a Display option, such as Transparency. After you have seen the effect reset it to zero.

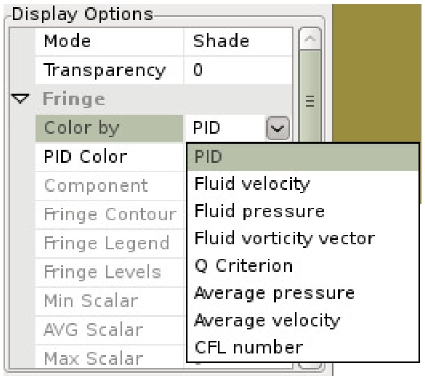
   Note: You can set a value between 1 and 10 for the transparency, often it is more practical to change display options for an entire group, i.e. MS_SHELL_GROUP and MS_SOLID_GROUP.

3. Now click on MS_SOLID_1 under MS_SOLID_GROUP in the Object window and then set the transparency to five in the Display Options.



4. Then right click on MS_SOLID_1 to see the menu with all the possible objects for the fluid volume. Start by selecting a Section plane, then change the normal in the Objects Properties to 0,0,1 and press Enter so that we have a plane right through the fluid domain.

5. Use the Animate bar to get to the last d3plot state, so that the time shows almost one second.



6. Then go to the Display Options, it will still have the section plane marked in the Object window. Click on Color by and select the fluid velocity, then scroll down until you see the Keep Min/Max option and check that box. Now you will have a fixed scale for the velocity when you go from state to state using the Animate bar.

7. Change the option “Color by” to “pressure”, but remember to uncheck the option “Keep min/max” to get an updated scale-range for the pressure. It is possible to create as many section planes as you want so play around with it to see if you can find any interesting flow phenomena. Note that the legends can be moved by putting the mouse on the color bar and pressing the left mouse button and moving the mouse to a better place and then releasing the mouse button.

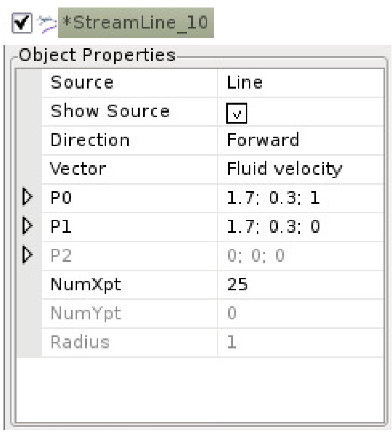


Figure 17 Streamline options in the Object Properties Window

8. To delete or add an option to an object, right click on that object and select what you want to do. It is also possible to group objects.

9. Now right click on MS_SOLID_1 and select streamline, then go to the Objects Properties window. Set point P0 and P1 to 1.7, 0.3, 1 and 1.7, 0.3, 0 respectively.

10. Keep the Source as Line and the Direction to forward, then set NumXpt to 25. This means that 25 lines will be drawn from a line source and only in the forward direction.

11. To view the streamlines, you have to right click on streamlines and select Update. When you see this symbol * in front of an object it means that it needs to be updated.

12. For coloring or changing the line appearance of the streamline go to the Display Option and have a look at Fringe and Line options.

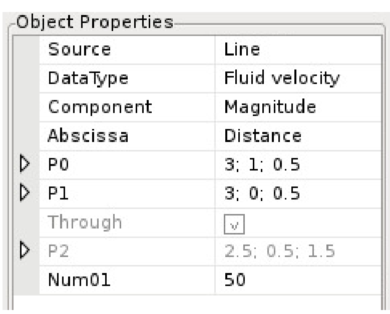


Figure 18 Options for the Data Plot in the Object Properties.

13. To plot the velocity over a line, right click on MS_SOLID_1 and select Data Plot.

14. Go to the Object Properties, having DataPlot selected in the Objects manager and change the Source to Line.

15. Set point P0 to 3, 1, 0.5 and P1 to 3, 0, 0.5, then set Num01 to 50 which means that 50 data points will be used to plot the line.

16. To see the plot right click DataPlot in the Objects window and select update. To hide the window, uncheck the box in front of the Data Plot in the Objects window.

5 Add FSI to the problem

To add FSI to the simulation make a directory called FSI and copy the following files to that folder:

- Solid_main.k – contains the structural keywords.
- Control_cards_nonlin.key – contains control cards for the structural solver.
- Database_cards_dynamic.key – contains output keywords for the structural solver.
- Mesh_solid.k – contains the structural mesh.

Also copy the files fluid_main.k and mesh_fluid.k that were used in the fluid analysis. Start by looking through the structural input deck, as you will see, the end time is set to 1.5 s for the structural input deck. When performing an FSI simulation you often want the same end time for both fluid and the structure. The following steps needs to be changed to run the FSI problem.

1. Open a new LS-PrePost session and include (*INCLUDE) the files fluid_main.k and solid_main.k, see the fluid part if you do not remember how to do.
2. Remove the *DATABASE_BINARY_D3PLOT keyword in fluid_main.k since the file database_cards_dynamic.key already have that keyword.
3. Go to the keyword manager by clicking Model/Keywrd and then select the keyword *ICFD_BOUNDARY_FSI and select part number five now press accept and done.
4. The next keyword that is needed is the *ICFD_CONTROL_FSI keyword, double click it and press Accept and done. We will use the default values in this simulation but later you can play around with the different ways the solvers can be coupled.
5. Now save the model as main_fsi.k and run this file using LS-DYNA.
6. Look at the results with LS-PrePost when the simulation is finished, notice that you now have a structural part in the post-processing. A few variables are available in the MS-post window, if you want to use additional post options for the solid use the Post option.

6 Additional exercises

1. Add boundary layers by adding the keyword *MESH_BL and the keyword *MESH_BL_SYM, start with 2 boundary layers and see how it affects the results.
2. Change the Young's modulus in the structural part and see how it affects the behavior of the fluid and structural part.
3. Change the inlet velocity or the density of the fluid and see what happens.
4. Add a local mesh refinement by using either *MESH_SIZE or *MESH_SIZE_SHAPE.
5. Try another turbulence model by using *ICFD_CONTROL_TURBULENCE but remember that the mesh size and the boundary layer requirement are different for different turbulence models.

7 Summary

Now you can set up a basic CFD simulation in LS-DYNA and also know how to add FSI to the simulation. This tutorial only included the basic futures of the ICFD solver, for more advanced options see the keyword manual or contact your local distributor of LS-DYNA for information about suitable courses or to get support.