Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 8. 2D Flow over a cylinder using the ICFD solver

Contents

1 Introduction

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 show you how to setup a 2D CFD simulation using LS-PrePost 4.3. The geometry is not complicated but the flow is very complex since a laminar von Karman vortex street, see Figure 1, is developed behind the cylinder for Reynolds numbers between 40 and 150. After Reynolds number 150 does the transition to a turbulent vortex street starts and occurs to Reynolds number 300 where the vortex street is fully turbulent.

The Reynolds number is a dimensionless number that is used to classify flows to get similar flow conditions. The definition of the Reynolds number is the ratio between inertial forces and viscous forces and can be written as:

$$Re = \frac{U\rho L}{\mu}$$

U free stream velocity, ρ fluid density, L characteristic length, and μ the dynamic viscosity. The characteristic length in this case is the diameter of the cylinder.

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 keyword manual vol. 3.

1.1 Prerequisites

- Basic knowledge about LS-DYNA and LS-PrePost.
- Basic understanding of FEM and CFD.
- LS-PrePost 4.3 or later and LS-DYNA R8.1 or later.

2 Problem description

It is a 2D example with flow over a circle in a rectangular domain. This example uses a Reynolds number of 100 and therefore you can say that the units in the example is dimensionless. The mesh that will be used in the example will be quite coarse since it is a demonstration example. You will see that you get a Von-Karman vortex street after the cylinder when the flow has stabilized.
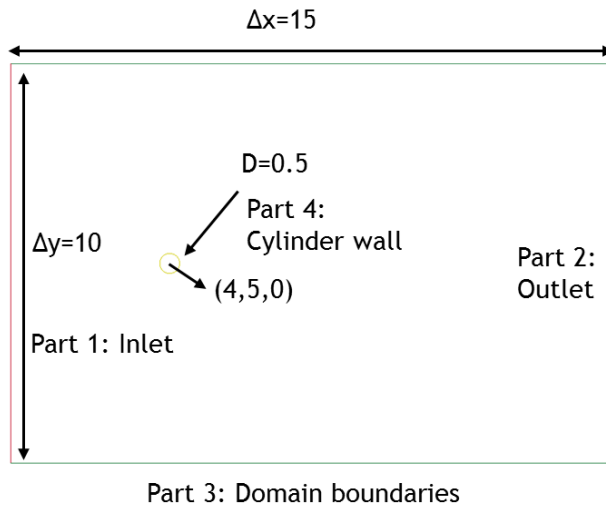
Part 3: Domain boundaries

Figure 2 Geometry of the example.

2.1 Files included in the tutorial

- The solution to the exercise can is found in **2Dcylinder_main.k**.
- The mesh created in Section 3.3 is found in **mesh_2Dcylinder.k**.

3 Setting up the CFD simulation

Before we can start defining the model we must define the geometry and mesh it. You can skip this step by importing the finished mesh in the file mesh_2Dcylinder.k.

3.1 Creating the geometry and meshing it

Start by opening LS-PrePost 4.3 or a later version, first create the geometry in the 2D Mesh Generation.

1. Click on Mesh.
2. In the left menu click on 2DMesh.
3. Now you will get a popup window where you will create the geometry of the model. The toolbar is in the top right corner with all the available tools and to the left is the drawing area.
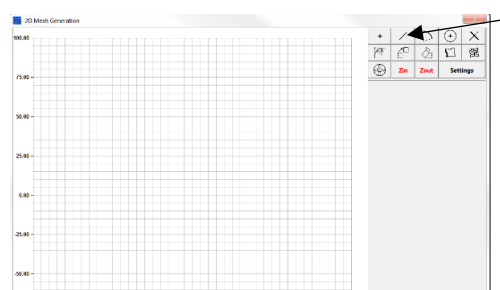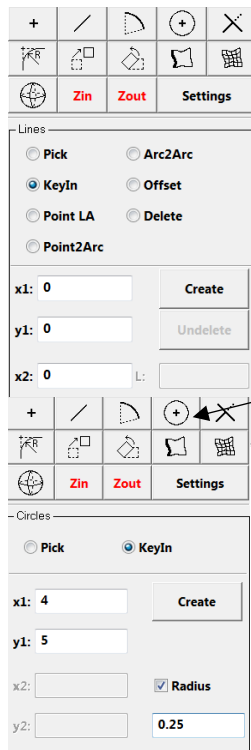

4. Click on the Line tool to create the rectangular boundary of the fluid domain.

Figure 3 2D Mesh Generation popup window.

5. Now you will see the control options for the line tool below the toolbar, click the **KeyIn** option to enter the coordinates for the first line.
6. Enter zero in the x1 and y1 field.
7. Enter zero in the x2 field and 10 in the y2 field.
8. Now press **Create** and you will see the line in the drawing area.
9. Create the next line by changing the x1 and x2 fields to 15, keep the y-values.
10. You should now have two vertical lines in the drawing area and to connect them change to the **Pick** option.
11. Connect the lines by clicking on the top points in the drawing area.
12. Do the same for the bottom points.
13. To create the circle, press the circle tool in the toolbar.
14. Use the **KeyIn** option.
15. Enter a x1 value of 4 and a y1 value of 5.
16. Click the **Radius** option and enter a value of 0.25.
17. Then press Create to create the circle and view it in the drawing area.
18. If the geometry looks like Figure 2 press **Exit** in the 2D Mesh Generation window.

3.2 Meshing the geometry

The ICFD solver uses a new type of mesh called *MESH, the format is the same but the heading has changed. Start by creating a regular LS-DYNA mesh and then convert it, the same applies if you are using a different pre-processor than LS-PrePost output it in a LS-DYNA format and then convert the mesh with LS-PrePost or a text editor. The ICFD solver has an automatic mesh generator and for it to work, the boundaries must be meshed with beam elements in 2D and shell elements in 3D. When creating the mesh use one part for each boundary condition that you will use in the simulation.
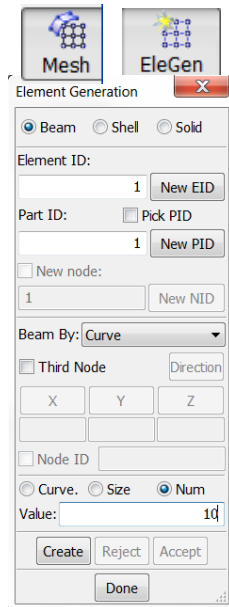
1. To generate the Mesh, use the Element generation tool. Click on Mesh and then EleGen in the left menu.
2. Click on the Beam option in the Element Generation popup window.
3. Change 'Beam By:' to Curve.
4. Select the Num and enter 10, this means that you will get 10 elements across the curve.
5. Select the curve that corresponds to Part 1 the Inlet and press **Create** and **Accept**.
6. Now change the 'Part ID:' to 2 and select the line that corresponds to Part 2, then press **Create** and **Accept**.
7. Now change the 'Part ID:' to 3 and use a Num value of 15, repeat step 5 but select the lines for Part 3.
8. For the Circle change to the option Size and enter 0.01.
9. Change the 'Part ID:' to 4 and select the curves for the circle, press **Create** and **Accept**.

Part 3: Domain boundaries
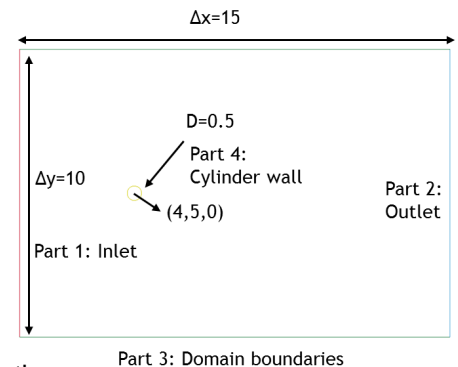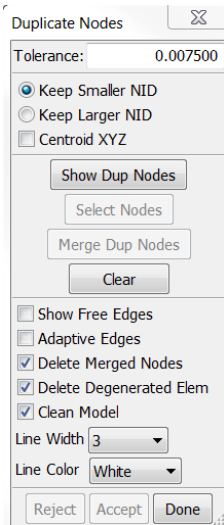
10. Now click on EleTol in the menu and then DupNod.
11. In the popup window press 'Show Dup Nodes' and then press 'Merge Dup Nodes'.
12. Now press Accept to approve the modification, the reason why this step is needed is that the ICFD mesh cannot have two nodes in the same location and the domain needs to be water tight.
13. Save the file as fluid_mesh.k for example.

3.3 Converting the mesh to the ICFD mesh format.

The first option is to use a text editor and change the heading of the keywords *NODE and *ELEMENT_BEAM to *MESH_NODE and *MESH_SURFACE_ELEMENT respectively. Only keep the nodes and elements remove all other keywords like *PART.

Second option is to use LS-PrePost.

Click on Mesh in the menu and then MSMesh in the left menu, the file that contains the mesh should be open in LS-PrePost.
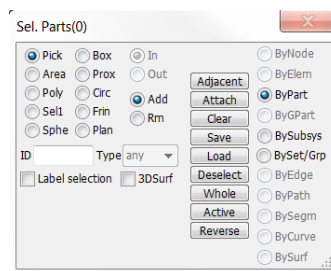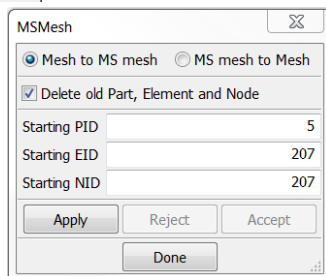
1. In the MSMesh window change Starting PID, Starting EID, and Starting NID to one.
2. In the Sel. Parts window select ByPart.
3. Press **Whole** to select all active parts in the current model.
4. Press **Apply** in the MSMesh window and if you are satisfied with the result press Accept.
5. Save the file as fluid_mesh.k for example, by clicking on File/Save/Save Keyword.

## 3.4 Create the CFD model

Open a new LS-PrePost window since it is recommended to have your mesh in one file and all other keywords in another file.

1. Click on Model in the menu and then Keywrd in the left menu to open the keyword manager.
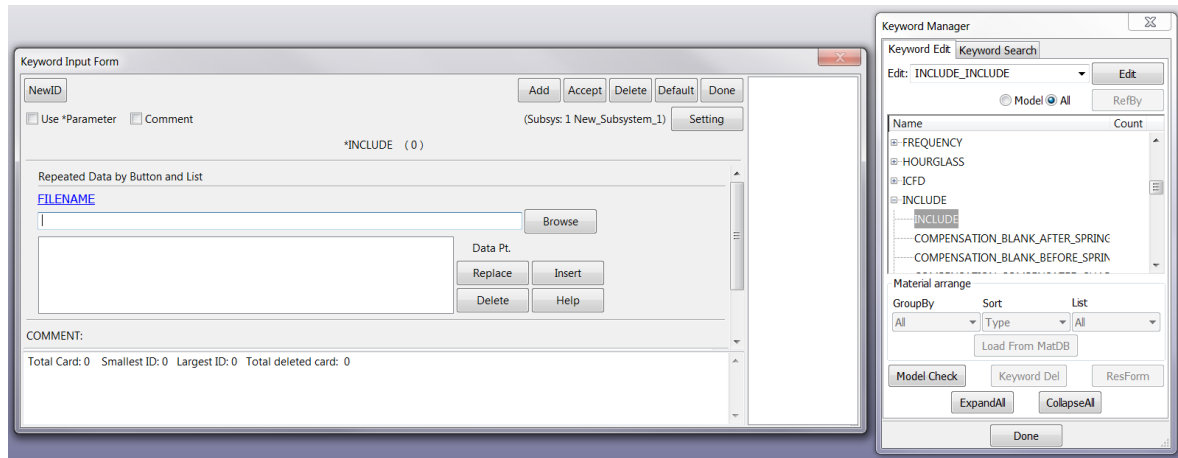
2. Locate the keyword *INCLUDE in the Keyword Manager and double click on it.
3. Locate the mesh file, fluid_mesh.k, if you have all files in the same folder you only need the filename and not the path to the file.
4. Then press **Insert** and then **Accept** before pressing **Done**. If you do not press Accept the changes made on the keyword is not saved.
5. To view the mesh, you must save the model and open it again. Save it as main_fluid.k and open it in a new LS-PrePost window.
6. Locate the keyword *ICFD_SECTION and double click on it.
7. The keyword *ICFD_SECTION needs to be defined but currently it has no purpose.
8. Press **Accept** and then **Done**.

LS-DYNA / LS-PrePost
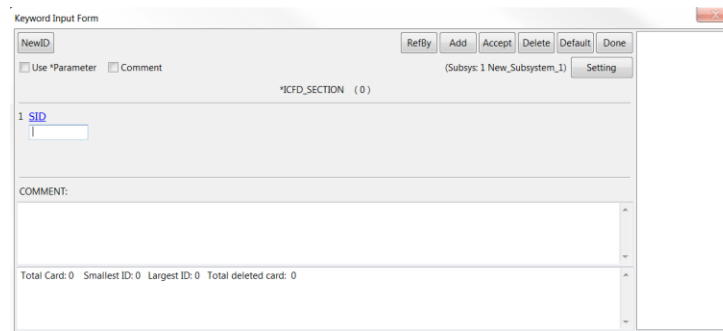
9. The next keyword defines the fluid properties, locate and double click on the keyword *ICFD_MAT in the Keyword Manager.
10. Enter a density of one on the parameter RO and a dynamic viscosity of 0.005 the parameter VIS.
11. Press **Accept** and **Done**.
12. Now locate the keyword *ICFD_PART if you did not have it defined in the mesh file it is automatically created. Double click it and make sure that the MID and SECID are the same as we defined previously, they should have the ID one if you did not change the ID value when defining the section and material keywords.

13. Either you enter the ID value on the parameters SECID and MID or click on the black circle next to the parameter.
14. Make this change for all four parts and remember to press Accept before changing part otherwise will the change not be saved.
15. Open the keyword *ICFD_PART_VOL.

16. This keyword is used to tell the solver which material the volume nodes will have. Enter the correct SECID and MID and enter all four parts before pressing Insert and then Accept. If you have more than eight parts, define the first eight and then enter a new row with the rest of the parts.

LS-DYNA / LS-PrePost

3.4.1 Boundary Conditions

This problem will use four different types of boundary conditions, one for each part see Figure 4. The inlet will have a prescribed velocity of one, the outlet will have a prescribed pressure of zero. The circle will have a no-slip condition which means that the velocity will be zero at the wall. The domain boundaries part 3 will have a free-slip condition which means that the normal velocity is zero but not the tangential velocity.
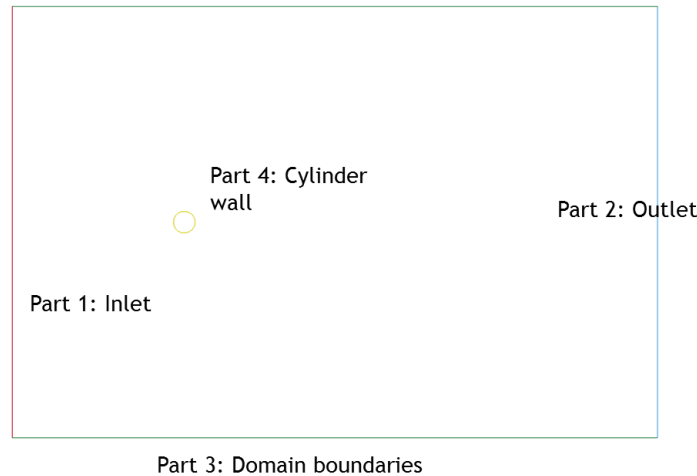

Figure 4 Boundary conditions.

1. The keywords that we will use to prescribe the velocity and the pressure uses a load curve so start by clicking on the keyword *DEFINE_CURVE to create two load curves one for the pressure and one for the velocity.

2. In the field O1 enter the value 1 and press **Insert**, now change the A1 value to 10000 and press **Insert**. This creates a curve that has a constant value of one during the whole simulation. Press **Accept** to save the curve.
3. Press NewID in the top left corner to create a new curve, change value of the O1 parameter for both points to zero by clicking on them and changing the value and then pressing **Replace**. Then press accept, if you want you can add a title to the curves.

LS-DYNA / LS-PrePost

4. Now press **Done** and locate the keyword *ICFD_BOUNDARY_PRESCRIBED_VEL in the Keyword Manager.

5. Enter part one in the parameter PID, this defines where the velocity will be applied.
6. Set the DOF to one which means that the x-velocity will be prescribed, the VAD parameter describes the velocity profile use the default option one a linear profile.
7. Select load curve 1, that had a constant value of one in the parameter LCID, press **Insert** and **Accept**.
8. Press **NewID** change the DOF parameter to 2, meaning y-velocity and change the load curve to number 2 that had a constant value of zero.
9. Press **Replace**, **Accept** and **Done**.
10. Open the keyword *ICFD_BOUNDARY_PRESCRIBED_PRE.

11. Enter part 2 which is the outlet in the PID parameter and use load curve 2 in the LCID parameter.
12. Press **Insert**, **Accept** and **Done**.

LS-DYNA / LS-PrePost

13. Locate the keyword *ICFD_BOUNDARY_NONSLIP that will be applied on the circle, part 4.

14. Enter Part 4 and press **Insert**, **Accept** and **Done**.
15. Open the keyword *ICFD_BOUNDARY_FREESLIP.

16. Enter Part 3 and press **Insert**, **Accept** and **Done**.

3.4.2 Control cards

That was all boundary conditions. What is left to define is the time step and end time. This is done with the control cards.

1. Find the keyword *ICFD_CONTROL_TIME and open it.

2. Set the end time to 100 on the parameter TTM and use the default values on the rest of the parameters. Click **Accept** and **Done**. A zero time-step size means that the solver uses an automatic time step based on the CFL criterion.

3. An optional keyword is the *ICFD_CONTROL_OUTPUT that controls the message level and output format.

4. The MSGL parameter controls how much information that is printed from the solver, the OUTL parameter controls if the results should be outputted in another format than the d3plot format. The DTOUT parameter controls how often these results should be printed and the last parameter controls if the fluid volume mesh should be printed to a separate file.

3.4.3 Controlling the boundary layer

To control and generate the boundary layer of Part 4, the circle, use the keyword *MESH_BL. This keyword tells the solver to generate a boundary layer mesh. You as a user have several ways to control how the boundary layer is created, for more information see the Keyword Manual Vol. III.

1. Open the keyword *MESH_BL.

2. The first parameter PID tells the solver from which part the boundary layer will grow, enter part 4 which is the circle.

3. The NELTH parameter controls the number of nodes between the wall and the last node in the boundary layer. Enter two which means that a boundary layer with three elements will be generated, keep the default values for the other parameters.

3.4.4 Outputting the results

To output the results, use the keyword *DATABASE_BINARY_D3PLOT.

1. Find the keyword in the Keyword Manager and open it.
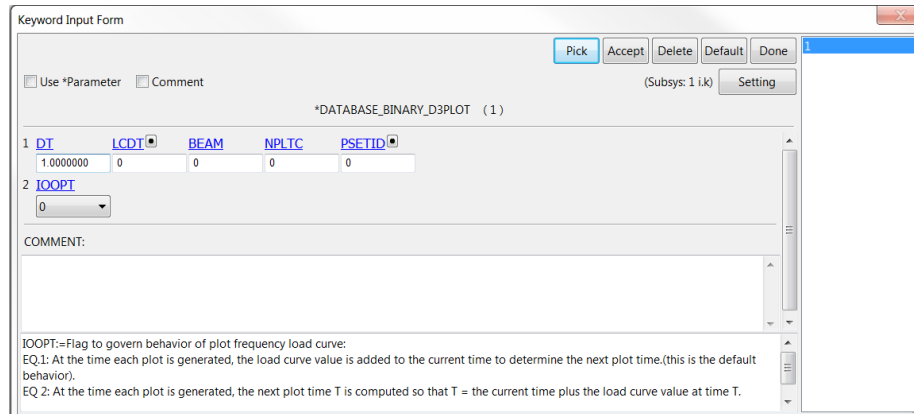
2. Enter one in the parameter DT which means that you will get one hundred states since the end time was 100.

4 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.

5 Post-processing

Use LS-PrePost 4.3 or a later version. For the most recently features, use LS-PrePost 4.5 which is the current beta version. To view the results, open the d3plot file with LS-PrePost. From version 4.2 a new GUI is available for multi-physics located under MS/MS-Post.

5.1 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. In the current beta version 4.5 of LS-PrePost, the post window is attached to the left side of the graphical window and it is displayed when a d3plot file contains data from a CFD simulation or a multi-physics simulation.

5.2 Basic post processing

The window is divided into three parts, Objects window, Objects Properties, and Display Options. The Objects window controls the visibility of objects and it is here that you create new objects by right clicking on an object.
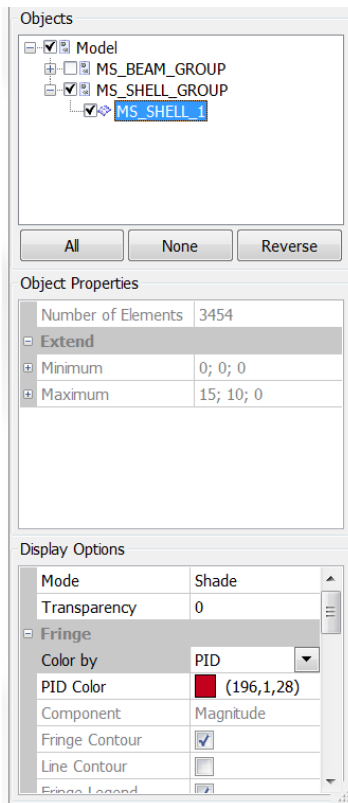
1. Since this is a 2D case you can display the results directly on the fluid domain by clicking on MS_SHELL_1.
2. In the Display Options change the Color by to Fluid velocity for example.
3. Then use the Animate bar to change the state or press play to see how the flow develops over time.
4. If you scroll down in the Display option you will find different ways to control the legend. Click on the Keep Min/Max box to keep the current max and min values of the legend for all states.
5. Notice also that if you click on the legend and drag it you can change the position of the legend.
6. To change the name of the legend, click on the corresponding object and then click one more time on the name. Now you can type a name of that object and press enter, the heading of the legend will also be updated.
7. To create streamlines right click on MS_SHELL_1 and select streamlines, make sure that the streamline object is selected in the Objects window.
8. In the Object Properties set P0 to (3,6,0) and P1 to (3,4,0).
9. Set the parameter NumXpt to 20 for example, this controls the number of stremlines that is generated.
10. Notice that the streamline object has a * in front of the name, this means that it needs to be updated. Right click on the streamline object in the Object window and pres update.

6 Optional exercises

Play around with the different options and see how it affects the results, here is some suggestions that you can test.

1. Refine the mesh especially the boundary mesh.
2. Change the time step size.
3. Change the boundary layer and test the other methods that are available.
4. Add a refinement zone with the keyword *MESH_SIZE_SHAPE.

7 Summary and where to learn more

Now you can set up a basic 2D CFD case and a 3D case is set up in a similar way. If you want more tutorials go to the YouTube channel LS-DYNA Corporate Tutorial & Content. It contains several tutorials and at www.dynaexamples.com you can download several examples of different types of simulations. If you have any questions please contact your local distributor of LS-DYNA for support or suggestions of available courses.