

# How to set up a 2D CFD case in LS-DYNA

Marcus Timgren  
DYNAmore Nordic

# Upcoming webex

- The following Webex about the ICFD solver are scheduled.
  - How to set up a 2D FSI case in LS-DYNA
    - Jan 23
  - How to set up a 3D CFD case in LS-DYNA
    - Feb 7
  - How to set up a 3D FSI case in LS-DYNA
    - Feb 28
  - Difference between weak and strong coupling in FSI simulations in LS-DYNA
    - Mars 7
  - CFD Mesh controls in LS-DYNA
    - Mars 21
  - How to model conjugate heat transfer in LS-DYNA
    - April 4
  - How to model flow through porous media in LS-DYNA
    - April 25
  - Coupling between DEM particles and ICFD solver
    - Maj 16
  - How to model sloshing using the ICFD solver
    - Maj 23

# Agenda CFD in LS-DYNA

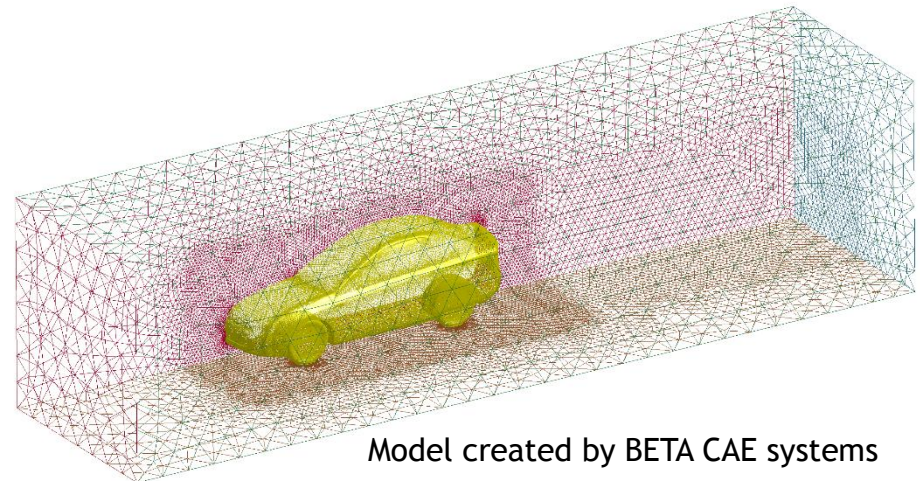
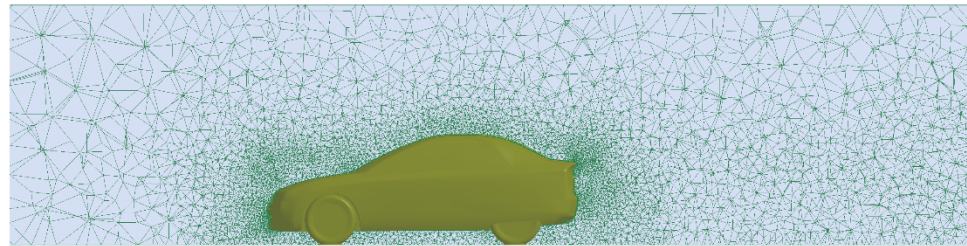
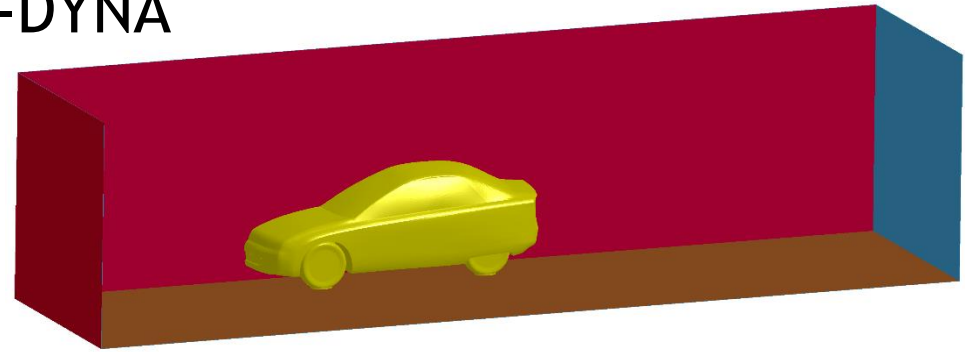
- Intro to the ICFD solver in LS-DYNA
- Model intro
- Mesh generation
- Part and fluid properties
- Boundary conditions
- Control cards
- Mesh cards
- Output results
- Run the model
- Results

# Intro to the ICFD solver in LS-DYNA

- An implicit solver that uses double precision and is available from version R7.0.
- Supports both 2D and 3D calculations.
- Has dynamic memory handling.
- Works both for SMP, (no scalability in SMP), and MPP.
- Can be coupled to the structural solver, the thermal solver, discrete element method, and the Electromagnetic solver (EM).
- Has two new sets of keywords \*ICFD and \*MESH.
- Only a transient solver at the moment, steady state is under development.
- Uses the Finite element method (FEM).

# Intro to the ICFD solver in LS-DYNA

- The solver has an automatic volume mesher with an option to create a boundary layer mesh.
- The only input needed is the surface mesh.
- It is also possible to use a predefined volume mesh, (only tetra/triangles elements supported in 3D/2D).



Model created by BETA CAE systems

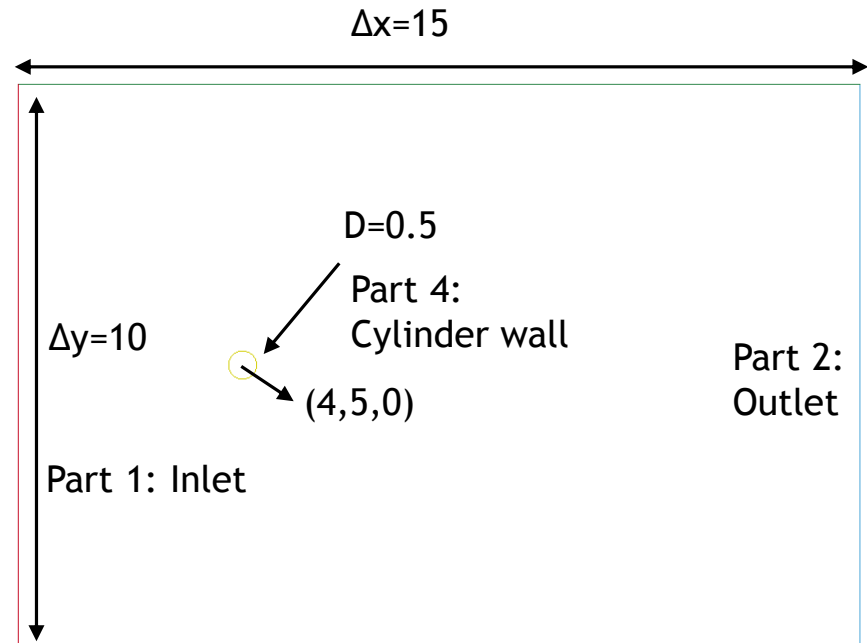
# Model intro

## ■ Flow over a cylinder

- The domain is a rectangle with a circle that the fluid flows over, see picture.
- The units are dimensionless since a Reynolds number of 100 is used.
- The model will have four boundary conditions.
  - Inlet
  - Outlet
  - Free-slip
  - No-slip
- Fluid properties
  - Density is set to 1.
  - Dynamic viscosity is set to 0.005.
- Remember the Reynolds number:
  - $Re_D = \frac{DU\rho}{\mu} = \frac{0.5*1*1}{0.005} = 100$

- The complete model can be downloaded at:

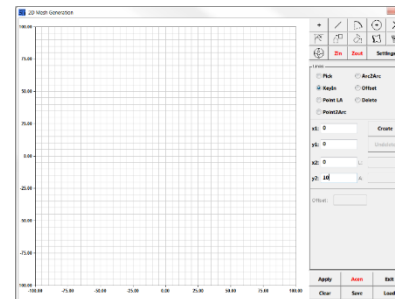
[http://www.dynaexamples.com/icfd/basic-examples/cylinder\\_flow](http://www.dynaexamples.com/icfd/basic-examples/cylinder_flow).



Part 3: Domain boundaries

# Mesh generation

- In this presentation LS-PrePost will be used.
  - Recommended version for CFD 4.3 or a higher version.
- Using the automatic mesher.
  - Create your domain boundaries and mesh them by beam elements, remember to create the geometry in the x-y plane.
  - Create one separate PID for each boundary condition that you will use.
  - Check that no duplicated nodes exists and that the boundaries are connected.
  - If using a different pre-processor than LS-PrePost output the mesh in LS-DYNA format.
  - At the moment Primer and LS-PrePost are the only pre-processors that support the ICFD solver keywords.
- If you want to mesh and create the geometry in LS-PrePost follow these steps.
  - Use the 2D Mesh Generation in LS-PrePost.
  - Click on Mesh and then 2DMesh, now you will get a popup window where you can create the geometry.

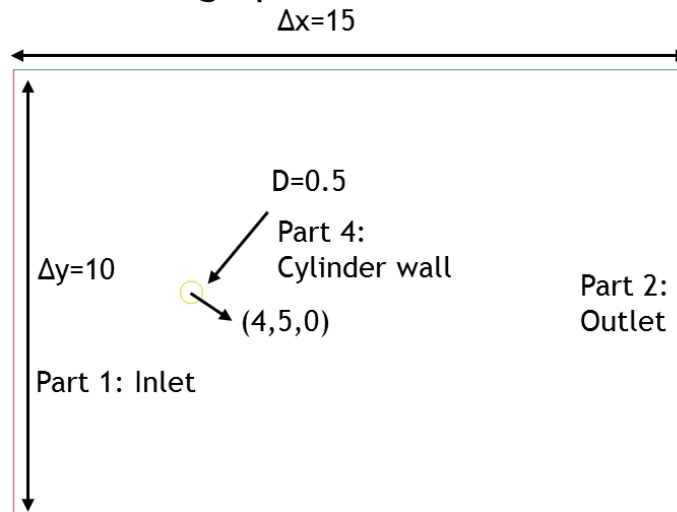


# Mesh generation

## ■ The 2D Mesh Generator

### ■ Creating the boundaries.

1. In the toolbar to the right press the line tool.
2. Select KeyIn to enter coordinates for the inlet line, the lower left corner has the coordinates (0,0) in the picture below.
3. Enter the coordinates for the first line and press Create.
4. Then change the x-coordinate to 15 and press Create.
5. Now change to the Pick mode and connect the two boundaries in the graphical window.



Part 3: Domain boundaries

The screenshot shows the software interface for the 2D Mesh Generator. The toolbar at the top contains various tools, including a line tool (indicated by an arrow). The 'Lines' panel is active, showing the 'KeyIn' mode selected. The 'x1' field is set to 0, and the 'y1' field is set to 0. The 'x2' field is set to 0, and the 'y2' field is set to 10. The 'Create' button is highlighted. The 'Offset' field is empty. The 'Apply', 'Acen', and 'Exit' buttons are visible at the bottom.

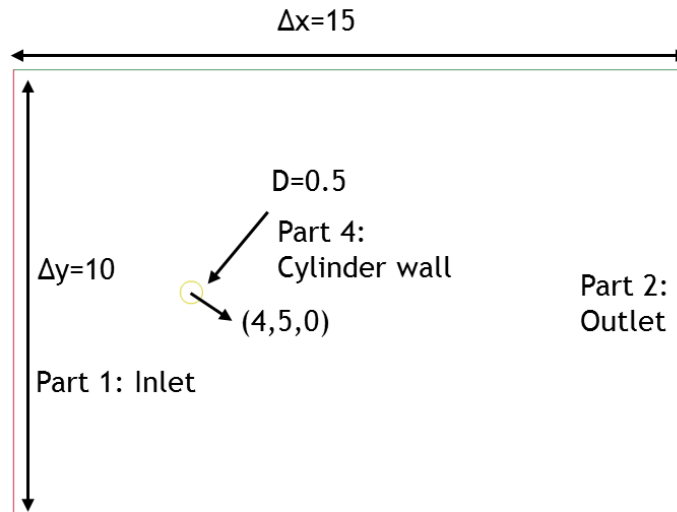


# Mesh generation

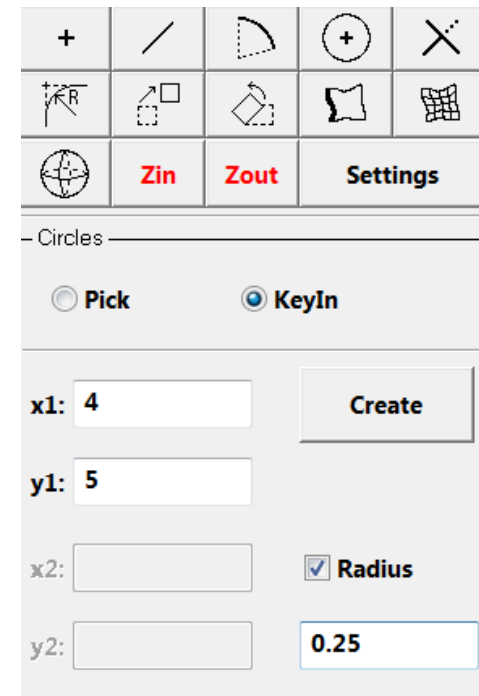
## ■ The 2D Mesh Generator

### ■ Creating the circle.

1. Press the circle tool in the toolbar.
2. Select KeyIn and enter the coordinates (4,5).
3. Click the Radius option and enter 0.25.
4. Then press create and if the geometry looks like the picture below you can press exit to leave the 2D Mesher.



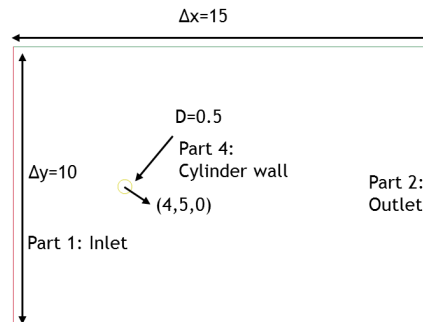
Part 3: Domain boundaries



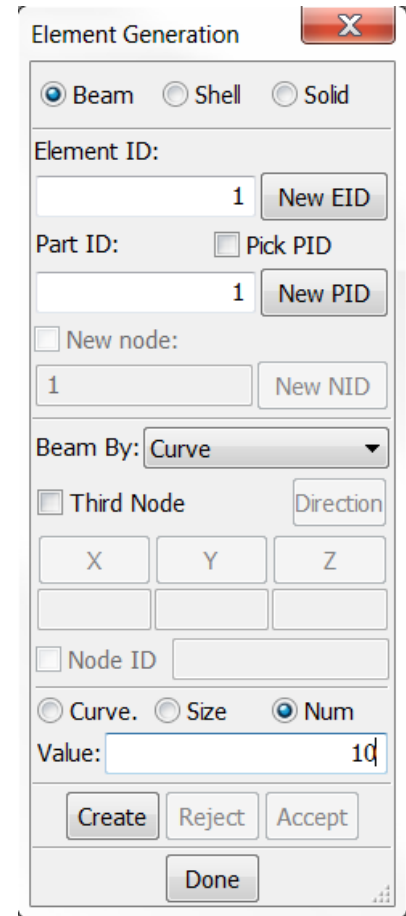
# Mesh generation

## ■ Convert the curves to beam elements.

1. To create the beam elements press Mesh/EleGen.
2. Select Beam in the Element Generation popup.
3. Change 'Beam By:' to Curve.
4. Select Num and enter 10 then select Part 1. Now press Create and the Accept.
5. Do the same for part 2 but change the Part ID to 2.
6. For Part 3 change the Num value to 15, remember to change the Part ID.
7. For the circle part change to Size and enter 0.01, use Part ID 4.
8. Check for duplicate nodes and merge them by using EleTol/DupNod then save the mesh as a keyword file called mesh.k.



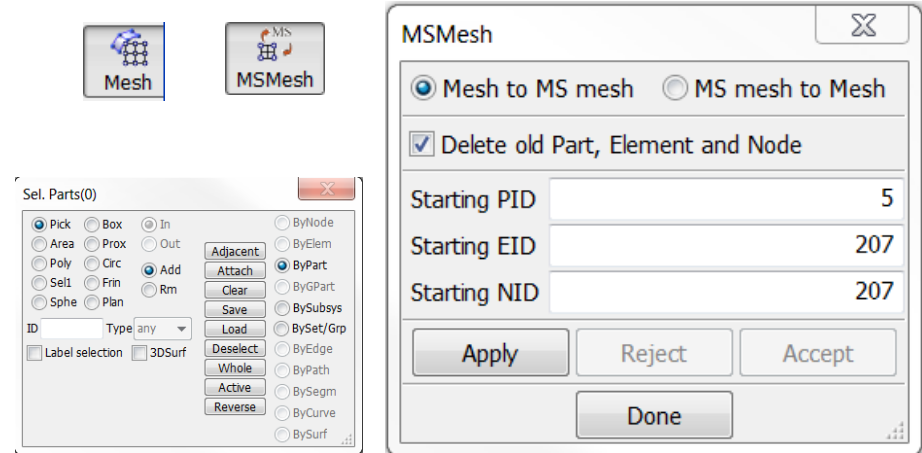
Part 3: Domain boundaries



# Mesh generation

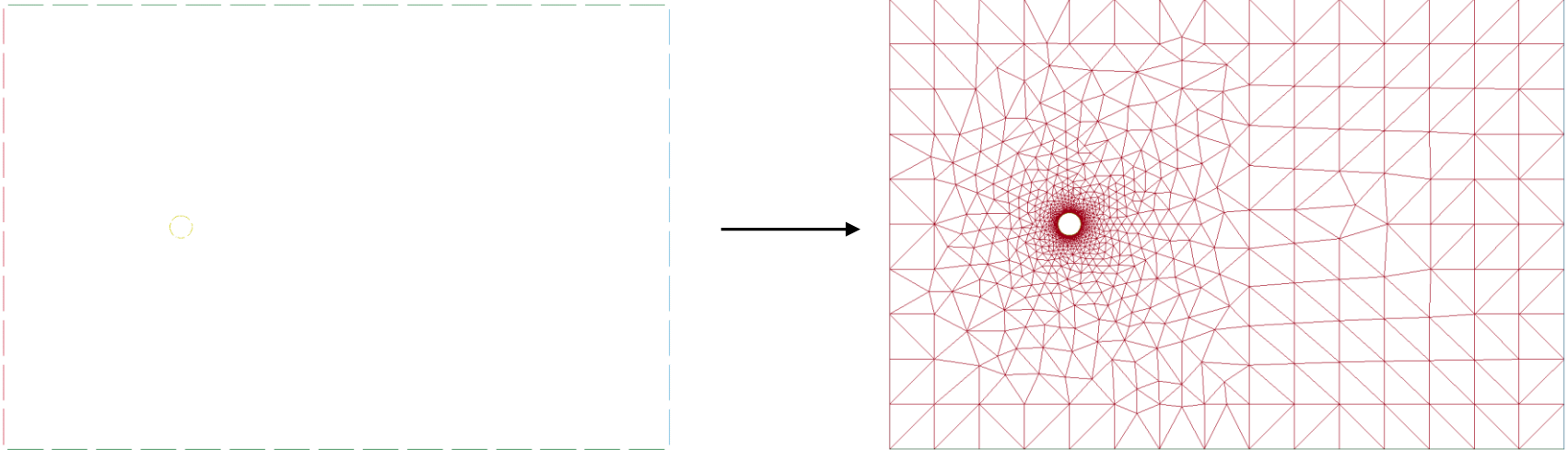
## ■ Convert the mesh to a ICFD mesh.

- First option is to use a text editor and change \*ELEMENT\_BEAM and \*NODE to \*MESH\_SURFACE\_ELEMENT and \*MESH\_SURFACE\_NODE respectively.
- The second option is to use LS-PrePost:
  - Click on Mesh/MSMesh
  - In the Sel. Parts window select all parts.
  - In the MSMesh window set Starting PID/EID/NID to one.
  - Then press Apply and if the model is highlighted press Accept.
  - Now save the mesh in a file called mesh.k.



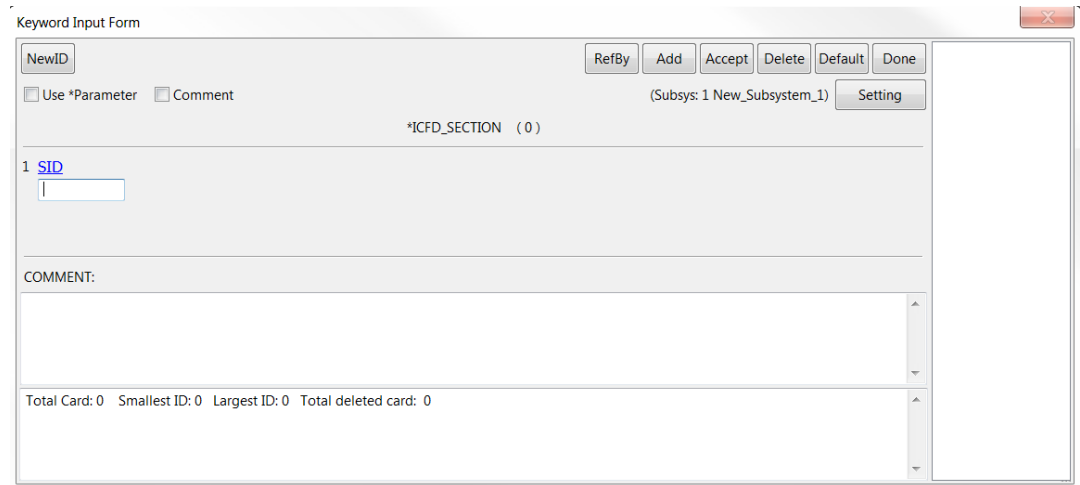
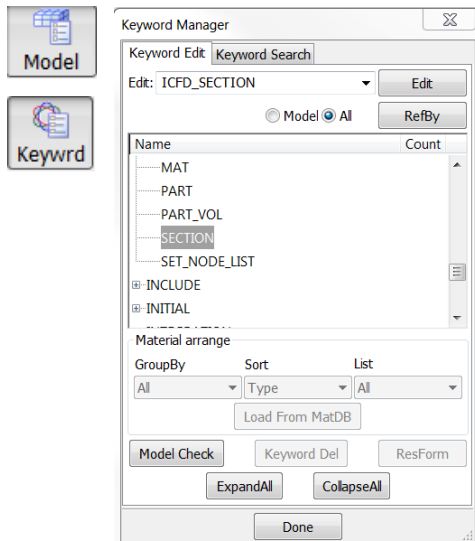
# Mesh generation

- The solver generates the following mesh from the beam mesh.
  - As you can see it is a coarse mesh.
  - If you want to use this case to compare to other results it is recommended to use a finer mesh, especially for the outer boundaries.



# Parts and fluid properties

- The first keyword is the \*ICFD\_SECTION.
  - A good practise is to have one file for the mesh and one file for all other ICFD keywords.
  - Start by including the mesh.k file by using \*INCLUDE and then save the model as main\_fluid.k for example.
  - Then close LSPP and open the new file you created, now you will see the mesh.
  - To add \*ICFD\_SECTION click on Model/Keywrđ and then make sure that All is marked in the Keyword Manager. Then go to ICFD and select SECTION, at the moment this keyword do not do anything but it must be defined so just click Accept and Done.



# Parts and fluid properties

- The next keyword is \*ICFD\_MAT
  - Double click on MAT under the ICFD family.
  - This keyword controls the fluid properties.
  - Set the density to one, the parameter RO, and then set the dynamic viscosity to 0.005, the parameter VIS.
- Then modify the ICFD\_PART keyword.
  - This keyword is automatically created when opening a ICFD mesh if it do not exist already.
  - Check that the SECID and MID has the same ID's as the created ICFD\_MAT and ICFD\_SECTION keyword.
  - Do this for all parts and remember to press Accept before changing to the next part.

Keyword Input Form

NewID [ ] RefBy [ ] Pick [ ] Add [ ] Accept [ ] Delete [ ] Default [ ] Done [ ]

Use \*Parameter  Comment (Subsys: 1 New\_Subsystem\_1) Setting

\*ICFD\_MAT,(TITLE) (0)

TITLE [ ]

1	MID	FLG	RO	VIS	ST
	[ ]	1	0	0	0

A  AB

2	HC	IC	BETA	PRT	HCSFLCID	ICSFLCID
	0	0	0	0.85	[ ]	[ ]

3	NNMOID	PMMOID
	[ ]	[ ]

Total Card: 0 Smallest ID: 0 Largest ID: 0 Total deleted card: 0

Keyword Input Form

NewID [ ] RefBy [ ] Pick [ ] Add [ ] Accept [ ] Delete [ ] Default [ ] Done [ ]

Use \*Parameter  Comment (Subsys: 1 New\_Subsystem\_1) Setting

\*ICFD\_PART,(TITLE) (0)

TITLE [ ]

1	PID	SECID	MID
	[ ]	[ ]	[ ]

COMMENT: [ ]

Total Card: 0 Smallest ID: 0 Largest ID: 0 Total deleted card: 0

# Parts and fluid properties

- ICFD\_PART\_VOL is used to associate the fluid properties with the fluid volume.
  - Now click on PART\_VOL under the ICFD family in the Keyword Manager.
  - Make sure that you are in the right Subsys, look below the Delete button, if not press Setting and select the file main\_fluid.k or the filename you used.
  - Select the section and material id that we created before.
  - Then select all parts that enclose the fluid volume, parts 1-4 in this case and press Insert followed by Accept.

Keyword Input Form

NewID: 10

Use \*Parameter  Comment

(Subsys: 1 i.k) Setting

\*ICFD\_PART\_VOL(TITLE) (1)

1 PID SECID MID

10 1 1

Repeated Data by Button and List

SPID1	SPID2	SPID3	SPID4	SPID5	SPID6	SPID7	SPID8
1	2	3	4	0	0	0	0

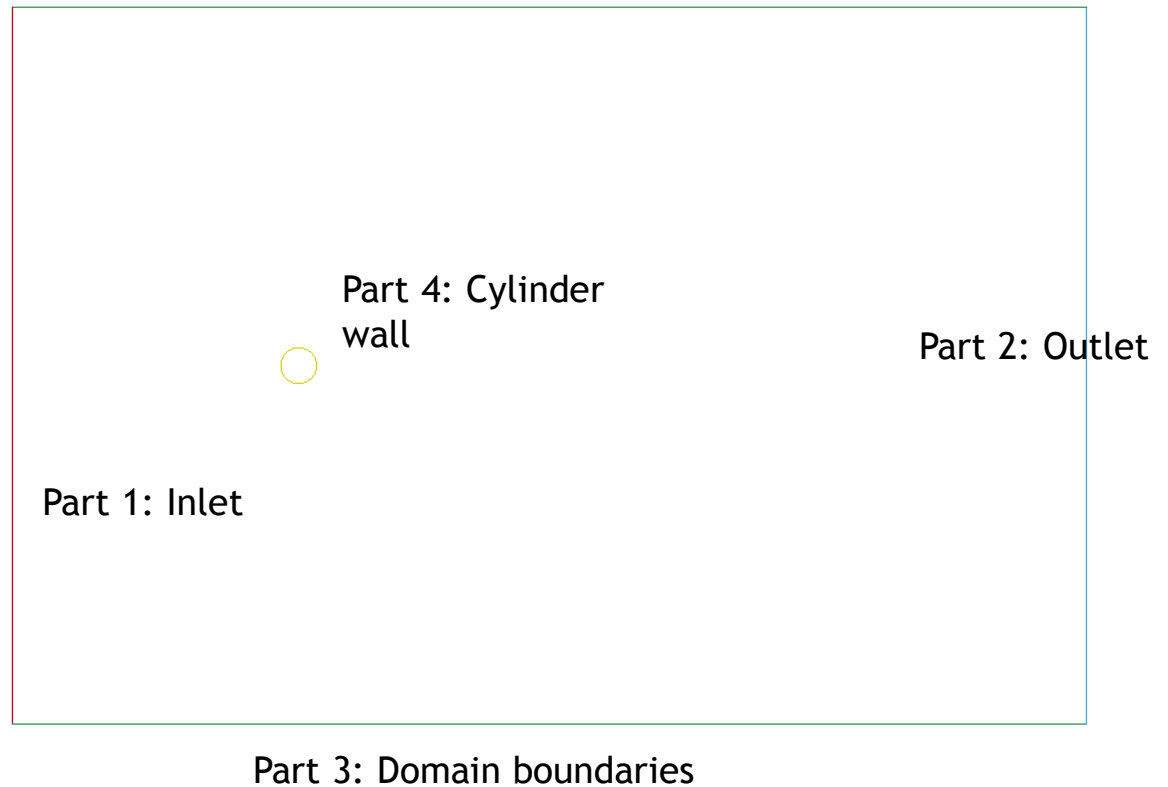
Data Pt. 1

Replace Insert

Total Card: 1 Smallest ID: 10 Largest ID: 10 Total deleted card: 0

# Boundary conditions

- Prescribed velocity
  - Part 1
- No-slip
  - Part 4
- Free-slip
  - Part 3
- Prescribed pressure
  - Part 2





# Boundary conditions

- Load curve for the inlet and outlet.
  - First create a load curve by clicking on DEFINE and then CURVE in the Keyword Manager.
  - Create a curve with a constant value of 1 by setting O1 to one for time zero and press insert, do the same for A1 value of 10000.
  - Then press Accept and press NewID to create a similar curve but with a constant value of zero.
- Inlet prescribed velocity.
  - Double click on BOUNDARY\_PRESCRIBED\_VEL under ICFD in the Keyword Manager.
  - Set DOF to 1, the x-direction, and VAD to 1, a linear profile, then select LCID 1. Next press Insert and accept then press NewID and set the y-direction to 0, use load curve 2.

Keyword Input Form

NewID Draw RefBy Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 ik) Setting

\*DEFINE\_CURVE\_(TITLE) (2)

Velocity inlet

LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	LCINT
1	0	1.0000000	1.0000000	0.0	0.0	0	0

Repeated Data by Button and List

A1	O1
0.0	1.0
1 0.0 1.0	
2 10000.0 1.0	

Data Pt. 1 Load XYData

Replace Insert Plot Raise

Total Card: 2 Smallest ID: 1 Largest ID: 2 Total deleted card: 0

Keyword Input Form

NewID Pick Add Accept Delete Default Done

Use \*Parameter  Comment (Subsys: 1 ik) Setting

\*ICFD\_BOUNDARY\_PRESCRIBED\_VEL (2)

Repeated Data by Button and List

PID	DOF	VAD	LCID	SE	VID	DEATH	BIRTH
1	1	1	1	1.0	0	1.00000E28	0.0

Data Pt. 1

Replace Insert

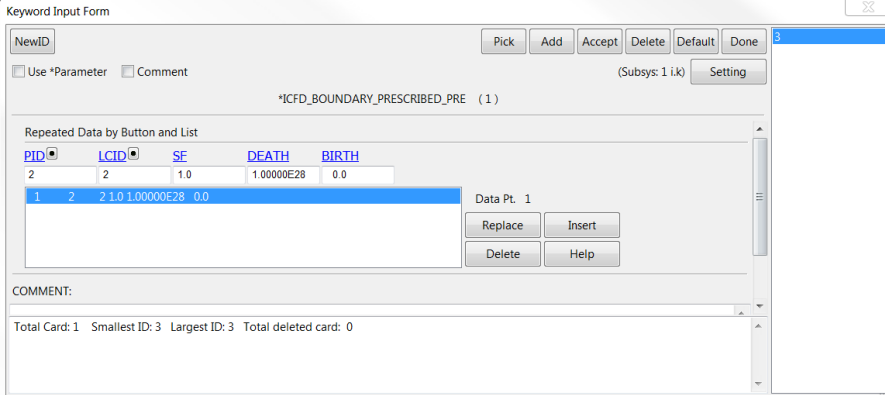
Delete Help

COMMENT:

Total Card: 2 Smallest ID: 1 Largest ID: 2 Total deleted card: 0

# Boundary conditions

- A zero pressure outlet condition.
  - Find \*ICFD\_BOUNDARY\_PRESCRIBED\_PRE in the Keyword Manager and select PID 2. Use the load curve with a constant value of zero, LCID 2.
  - Then press Insert and Accept.
- No-slip condition for the circle.
  - The velocity at a wall surface with friction is zero, to get that the keyword ICFD\_BOUNDARY\_NONSLIP is used.
  - Select the part id 4 which is the circle and press Insert. Then press accept and Done.
- Free-slip condition for the boundaries of the fluid domain.
  - Locate the keyword ICFD\_BOUNDARY\_FREESLIP and use PID 3.
  - This boundary condition allow the fluid to have a velocity along the boundary, used for far field boundaries.



Keyword Input Form

NewID: [ ] Pick Add Accept Delete Default Done 3

Use \*Parameter  Comment (Subsys: 1 i.k) Setting

\*ICFD\_BOUNDARY\_PRESCRIBED\_PRE (1)

Repeated Data by Button and List

PID	LCID	SF	DEATH	BIRTH
2	2	1.0	1.00000E28	0.0
1	2	1.0	1.00000E28	0.0

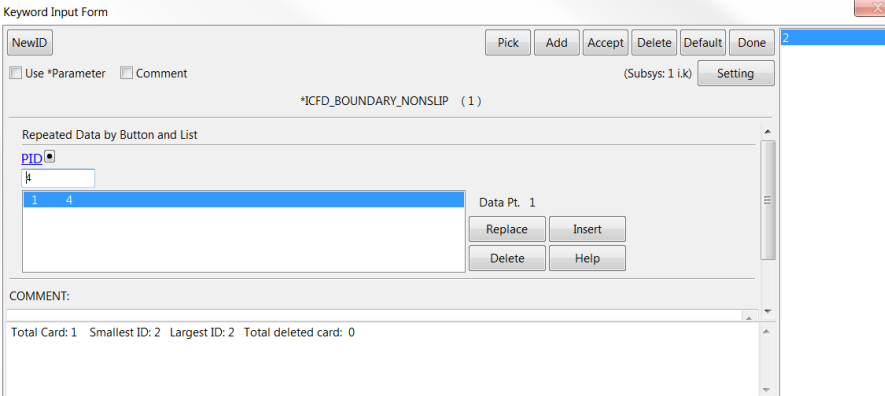
Data Pt. 1

Replace Insert

Delete Help

COMMENT:

Total Card: 1 Smallest ID: 3 Largest ID: 3 Total deleted card: 0



Keyword Input Form

NewID: [ ] Pick Add Accept Delete Default Done 2

Use \*Parameter  Comment (Subsys: 1 i.k) Setting

\*ICFD\_BOUNDARY\_NONSLIP (1)

Repeated Data by Button and List

PID	
4	
1	4

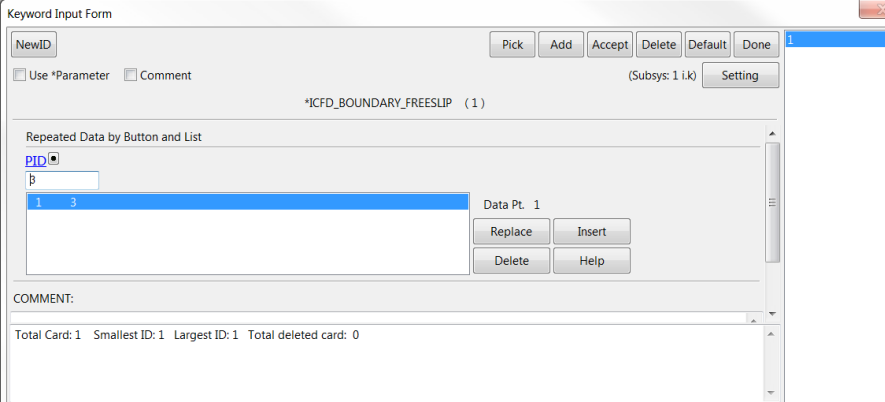
Data Pt. 1

Replace Insert

Delete Help

COMMENT:

Total Card: 1 Smallest ID: 2 Largest ID: 2 Total deleted card: 0



Keyword Input Form

NewID: [ ] Pick Add Accept Delete Default Done 1

Use \*Parameter  Comment (Subsys: 1 i.k) Setting

\*ICFD\_BOUNDARY\_FREESLIP (1)

Repeated Data by Button and List

PID	
3	
1	3

Data Pt. 1

Replace Insert

Delete Help

COMMENT:

Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0

# Control cards

- To control the time step and end time the keyword `*ICFD_CONTROL_TIME` is used.
  - Set the end time to 100, the parameter TTM and keep the remaining parameters as default.
  - The time step will automatically be calculated by the solver based on the CFL criterion.
- Another good keyword to use is `*ICFD_CONTROL_OUTPUT`.
  - It can output the fluid volume mesh and the results in another format than d3plot, for example in a Paraview format.
  - It also controls the message level details from the solver that are printed in the terminal and message files.

Keyword Input Form

NewID  Use \*Parameter  Comment (Subsys: 1 i.k)

\*ICFD\_CONTROL\_TIME (1)

1	TTM	DT	CFL	LCIDSE	DTMIN	DTMAX
	100.000000	0.0	1.0000000	0	0.0	0.0

COMMENT:

Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0

Keyword Input Form

NewID  Use \*Parameter  Comment (Subsys: 1 i.k)

\*ICFD\_CONTROL\_OUTPUT (0)

1	MSGL	OUTL	DTOUT	LSPPOUT
	4	0	0.0	1

COMMENT:

OUTL=Output the fluid results in other file formats apart from d3plot.  
EQ: 0: only d3plot output  
EQ: 2: output a file with mesh statistics and the fluid results in GMV format. A directory named output/gmv has to be created one level above the executable.

# Mesh cards

- To generate the fluid volume mesh the keyword **\*MESH\_VOLUME** is used.
  - Locate the keyword under MESH in the Keyword Manager.
  - Select all parts that encloses the fluid volume and press Insert. Then press Accept and Done.
- To generate the boundary layer mesh the keyword **\*MESH\_BL** is used.
  - Select which PID that will have a boundary layer, in this case PID 4, then enter 2 in the NELTH parameter.
  - This means that the element closest to the surface will be divided two times and generating three boundary layer elements.
  - Keep the remaining parameters as default and press Insert and then Accept.

Keyword Input Form

NewID [ ] RefBy [ ] Pick [ ] Add [ ] Accept [ ] Delete [ ] Default [ ] Done [ ]

Use \*Parameter  Comment (Subsys: 1 i.k) Setting [ ]

\*MESH\_VOLUME (1)

1 **VOLID**

Repeated Data by Button and List

PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
1	1	2	3	4	0	0	0
1	1	2	3	4	0	0	0

Data Pt. 1

Replace [ ] Insert [ ]

Delete [ ] Help [ ]

Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0

Keyword Input Form

NewID [ ] Pick [ ] Add [ ] Accept [ ] Delete [ ] Default [ ] Done [ ]

Use \*Parameter  Comment (Subsys: 1 i.k) Setting [ ]

\*MESH\_BL(TITLE) (1)

**TITLE**

Repeated Data by Button and List

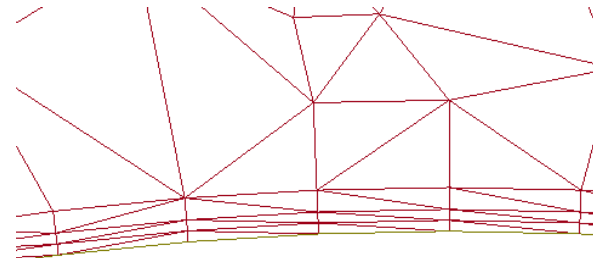
PID	NELTH	BLTH	BLFE	BLST
1	2	0.0	0.0	0
1	4	2	0.0	0

Data Pt. 1

Replace [ ] Insert [ ]

Delete [ ] Help [ ]

Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0



# Output results

- To output results the keyword **\*DATABASE\_BINARY\_D3PLOT** is used.
  - Locate the keyword under DATABASE and double click on it.
  - Enter 1 in the DT parameter field this means that you will get 100 result states since the end time is 100.
  - Then press Accept and Done.
- ICFD database keywords that are useful for this case.
  - The **\*ICFD\_DATABASE\_DRAG** outputs the drag force on a given boundary, set PID 4 in this case.
  - The **\*ICFD\_DATABASE\_TIMESTEP** outputs a file called icfd\_tsout.dat and outputs the taken time steps and the CFL time step.
- Save the model as a keyword file.

Keyword Input Form

Use \*Parameter  Comment

(Subsys: 1 (k))

\*DATABASE\_BINARY\_D3PLOT (1)

1	DT	LCDT	BEAM	NPLTC	PSETID
	1.0000000	0	0	0	0

2 IOOPT  
0

COMMENT:

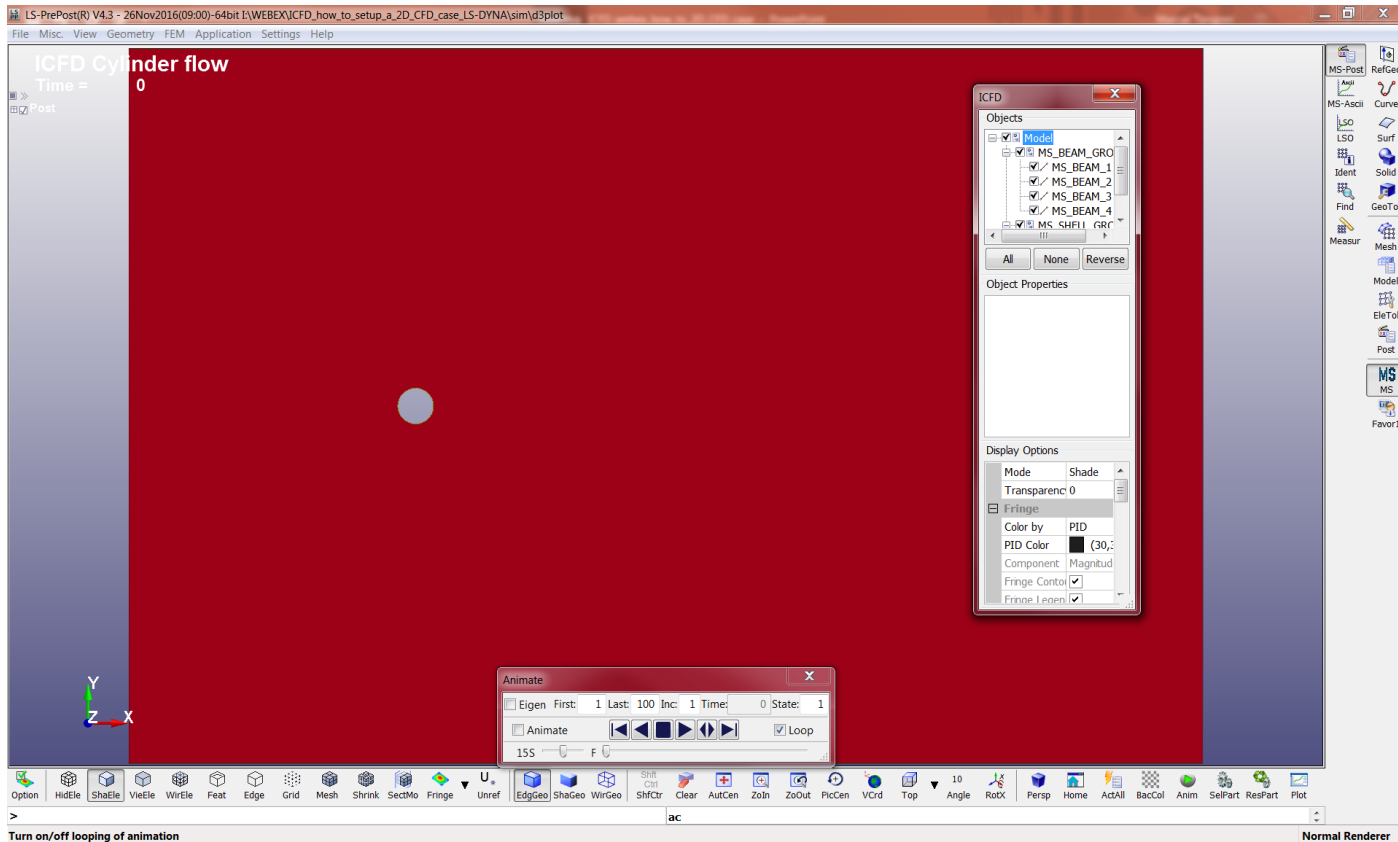
IOOPT=Flag to govern behavior of plot frequency load curve:  
EQ.1: At the time each plot is generated, the load curve value is added to the current time to determine the next plot time.(this is the default behavior).  
EQ.2: At the time each plot is generated, the next plot time T is computed so that T = the current time plus the load curve value at time T.

# Run the model

- You have to use a double precision version of LS-DYNA since the ICFD solver do not exists in the single precision version.
- I recommend to use R9.0.1 or higher LS-DYNA version.
- For windows:
  - I recommend to use LS-RUN, can be downloaded for free from <https://ftp.dynamore.se/public/lsprepost/LS-Run/>.
  - To get scalability use a MPP version since the SMP version do not have any scalability implemented.
  - The ICFD solver uses dynamic memory so you could start it with 1M.
- For Linux:
  - Our command looks like this:
    - `mppdyna 4 ls-dyna_mpp_d_r9_0_1_109912_x64_redhat54_ifort131_sse2_platformmpi i=i.k memory=1m`
    - mppdyna is our script to run a mpp simulation.
    - Check the manual for more information.
  - LS-RUN works on Linux as well if you want to use it.
- The model takes about 5 min on 1 CPU.

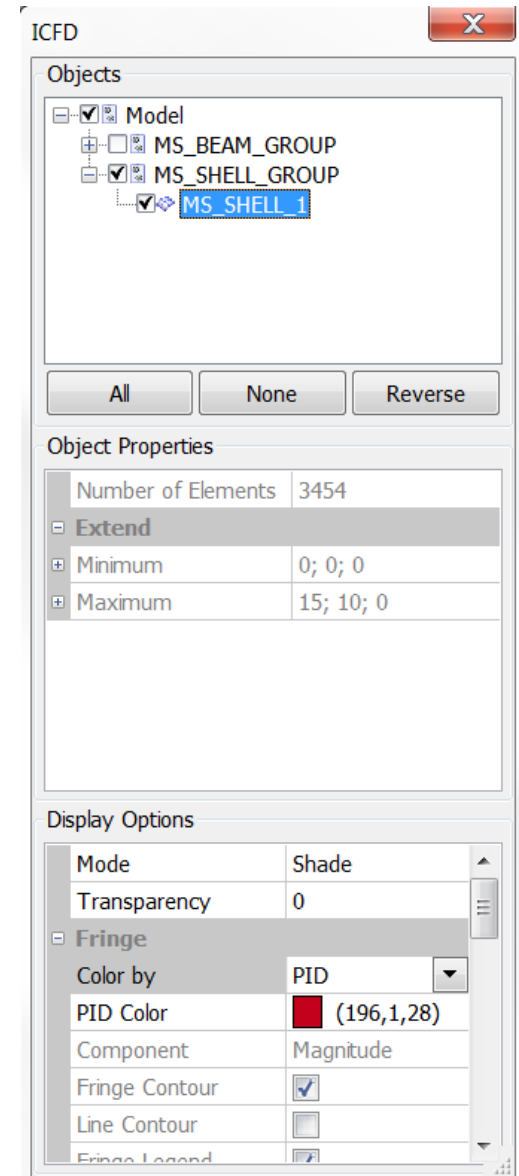
# Results

- To post-process the results I recommend to use LS-PrePost 4.3 or a higher version.
  - When opening the d3plot file it will look like this in LS-PrePost 4.3.
  - The time is controlled by the Animate bar.



# Results

- The objects window determines the visibility of the objects and it is also here that you create more objects by right clicking on a object.
- In a 2D simulation most of the interesting results can be found on the MS\_SHELL object.
- The easiest way to present results is to change the PID colour in the Display Options.
  - Select the Object MS\_SHELL and then change the “Color by” in the Display Option window to Fluid velocity.
  - If you scroll down in the Display Option window there exists some control parameters for the fringe legend.
- Create streamlines by right clicking on the MS\_SHELL object and choose Streamline.





# Results

- The source of the streamlines is controlled in the Object Properties.
  - Click on P0 and select a point on the model or type the coordinates, do the same for P1. The point P2 is not used since the source is set to a line.
  - The number of lines is controlled by the NumXpt and set it to 15 to start with.
  - As you can see in the Objects window the streamline has a \* in front of it which means that it needs to be updated. Right click on the Streamline object and select update, now you should see the streamlines if not check that you have moved from state 1.
  - How the streamlines is presented is controlled under the Display Options.
- To plot results from a ICFD\_DATABASE keyword use the MS-Ascii button and load the correct database.
- This gives you an basic understanding of how to post-process the results.



The image shows a screenshot of the ICFD software interface. The 'Objects' window displays a tree structure with the following items: Model, MS\_BEAM\_GROUP, MS\_SHELL\_GROUP, MS\_SHELL\_1, and \*StreamLine\_8. The \*StreamLine\_8 object is selected. Below the tree are three buttons: All, None, and Reverse. The 'Object Properties' section contains a table with the following data:

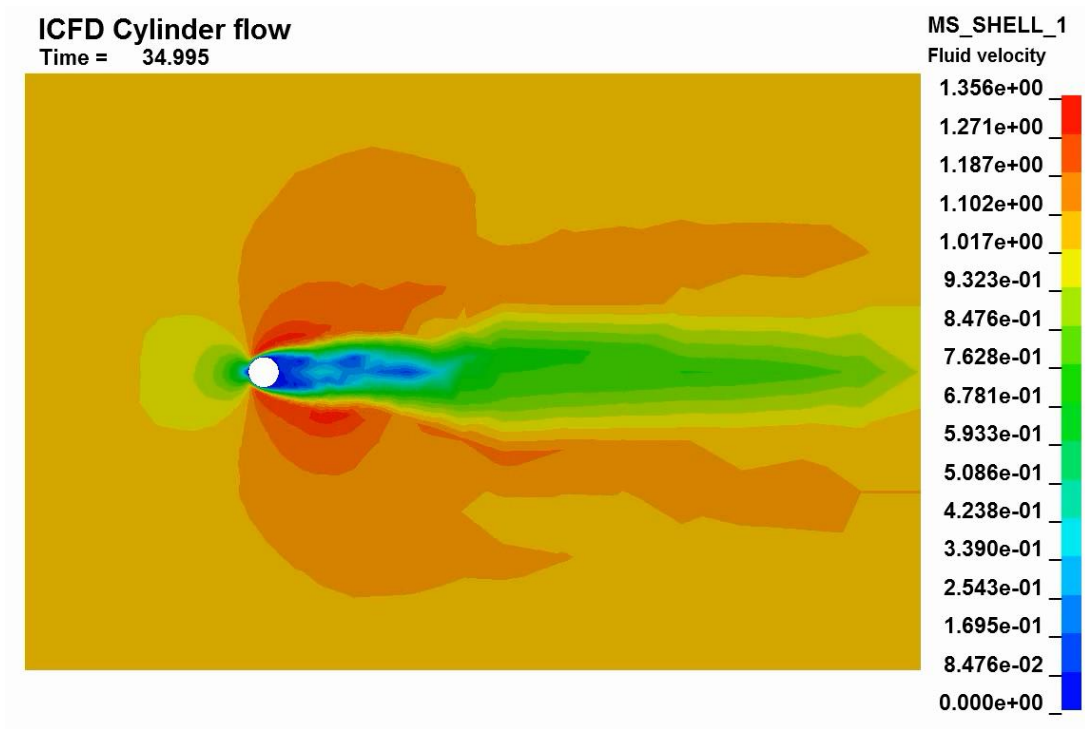
Property	Value
Source	Line
Show Source	<input checked="" type="checkbox"/>
Direction	Forward
Vector	Fluid velocity
P0	0; 0; 0
P1	0; 0; 0
P2	0; 0; 0
NumXpt	0
NumYpt	0
Radius	1

The 'Display Options' section contains a table with the following data:

Property	Value
Mode	Shade
Transparency	0
Fringe	
Color by	PID
PID Color	(51,159,96)
Component	Magnitude
Fringe Contour	<input checked="" type="checkbox"/>
Fringe Legend	<input checked="" type="checkbox"/>
Fringe Levels	10
Min Scalar	0

# Results

- Fluid velocity

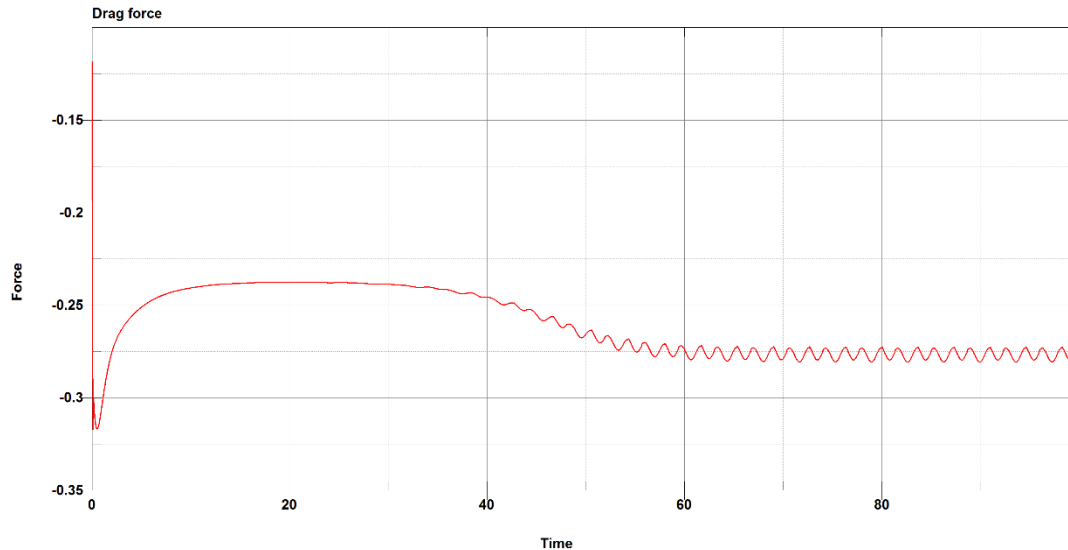


# Results

## ■ Drag force for the circle

### ■ Analytical formula for the drag force.

- $F_D = C_D A \frac{1}{2} \rho U^2$  where  $C_D$  is the drag coefficient and can be set to 1.2 for a circle and laminar flow.
- When using the values from the simulation it gives a force of 0.1178 compared to force of 0.27 from the simulation.
- The difference can be explained by the coarse mesh and the vortex shedding. For a benchmark study of this case look at: [ftp://ftp.lstc.com/outgoing/inaki/docs/pdf\\_icfd/Cylinder.pdf](ftp://ftp.lstc.com/outgoing/inaki/docs/pdf_icfd/Cylinder.pdf)



Thank you!

**DYNA**  
MORE

Your LS-DYNA distributor and  
more

