Using the new Solution Explorer in LS-PrePost to set up an LS-DYNA ICFD simulation

Erik Svenning, PhD
DYNAmore Nordic AB
erik.svenning@dynamore.se

Purpose

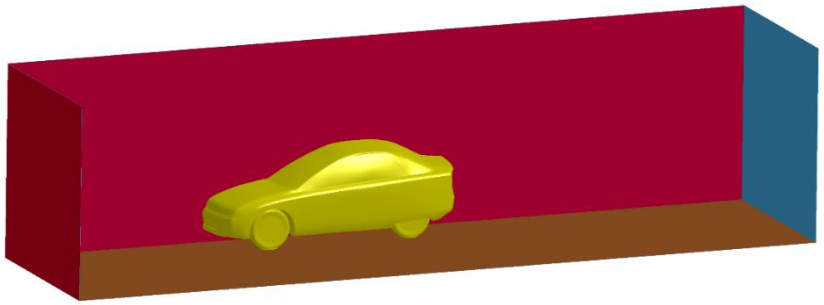
- Set up a fluid flow simulation using the ICFD solver in LS-DYNA and the new Solution Explorer GUI in LS-PrePost
  - Easier and more intuitive compared to the conventional keyword based setup

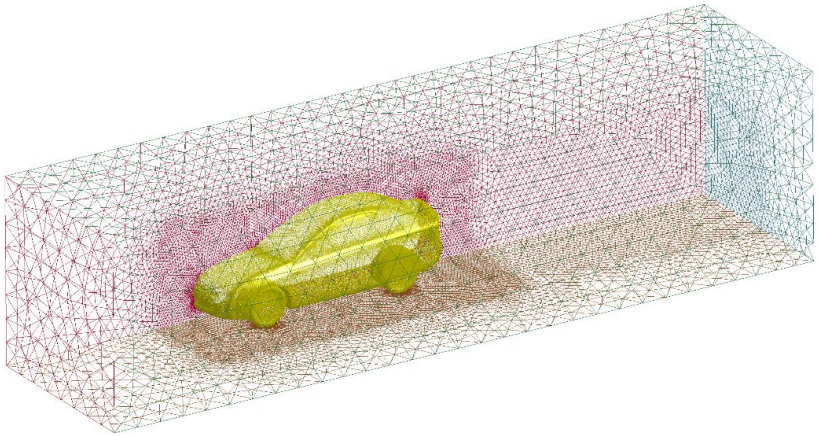
Outline

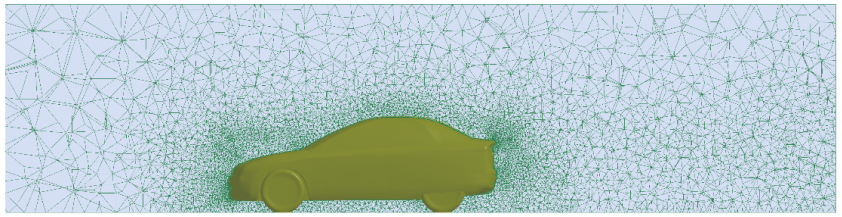
- Intro to the ICFD solver in LS-DYNA
- The new GUI in LS-PrePost
- Simulation setup: pipe flow
- Post-processing
- Summary

Solver introduction

- Features of the ICFD solver in LS-DYNA
  - An implicit solver for incompressible fluids
  - Recommended to use version R9.2 or R10.1
  - Uses the Finite Element Method (FEM)
  - 2D and 3D calculations
  - Automatic volume mesh generation
  - Coupled simulations are supported
    - Fluid-Structure Interaction (FSI)
    - Conjugate heat transfer
  - Non-Newtonian fluids
  - Turbulence models
  - Porous media
  - Free surface flows (level set)
  - Steady state solver (from R10)



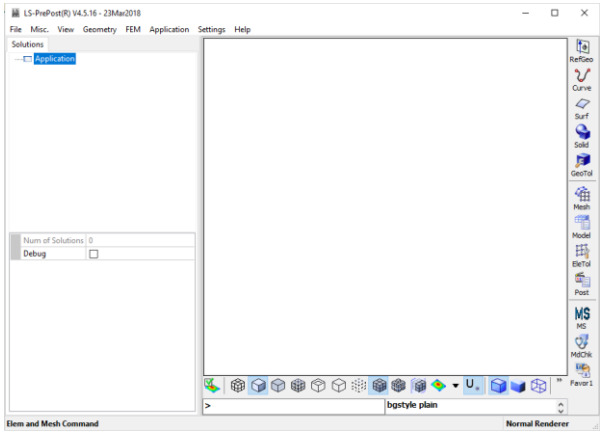


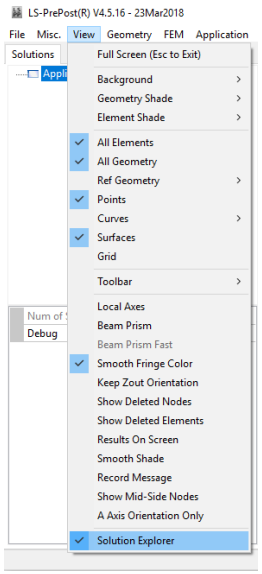


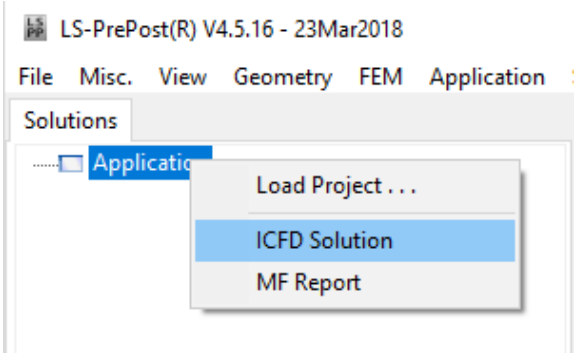
Model created by BETA CAE systems

The Solution Explorer in LS-PrePost

- LS-PrePost version 4.5 is used in this tutorial
- Alternative to the conventional keyword based setup
- Located to the left in the LS-PrePost window
- Activated by clicking View->Solution Explorer
- A new simulation can be created by right-clicking in the solution explorer window

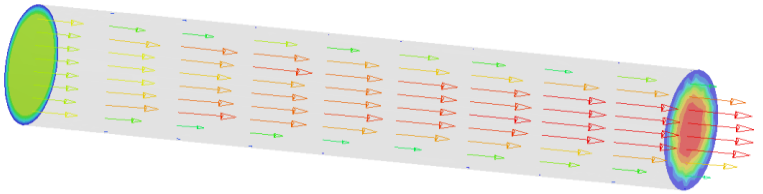


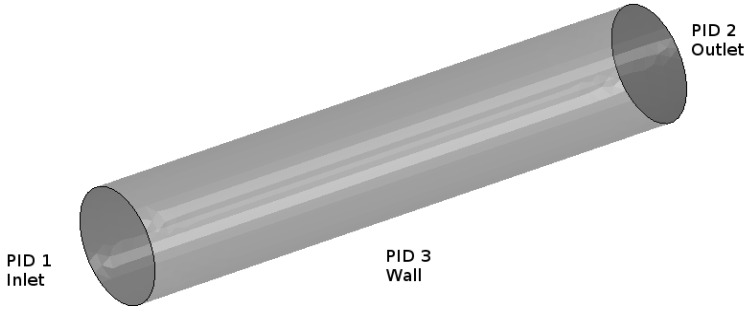




Example problem: pipe flow

- Flow through a cylindrical pipe
  - Length L = 5 cm
  - Diameter D = 1 cm
  - Inlet with prescribed velocity of 0.1 m/s
  - Outlet with zero pressure
  - No-slip conditions on the pipe wall
  - Water with density ρ = 1000 kg/m^3 and viscosity μ = 1 mPas
  - Re = ρUD/μ = 1000
  - The task is to compute the velocity and pressure fields of the fluid in the pipe.
  - PIDs for the different parts according to the figure
    - PID 1: Inlet
    - PID 2: Outlet
    - PID 3: Wall





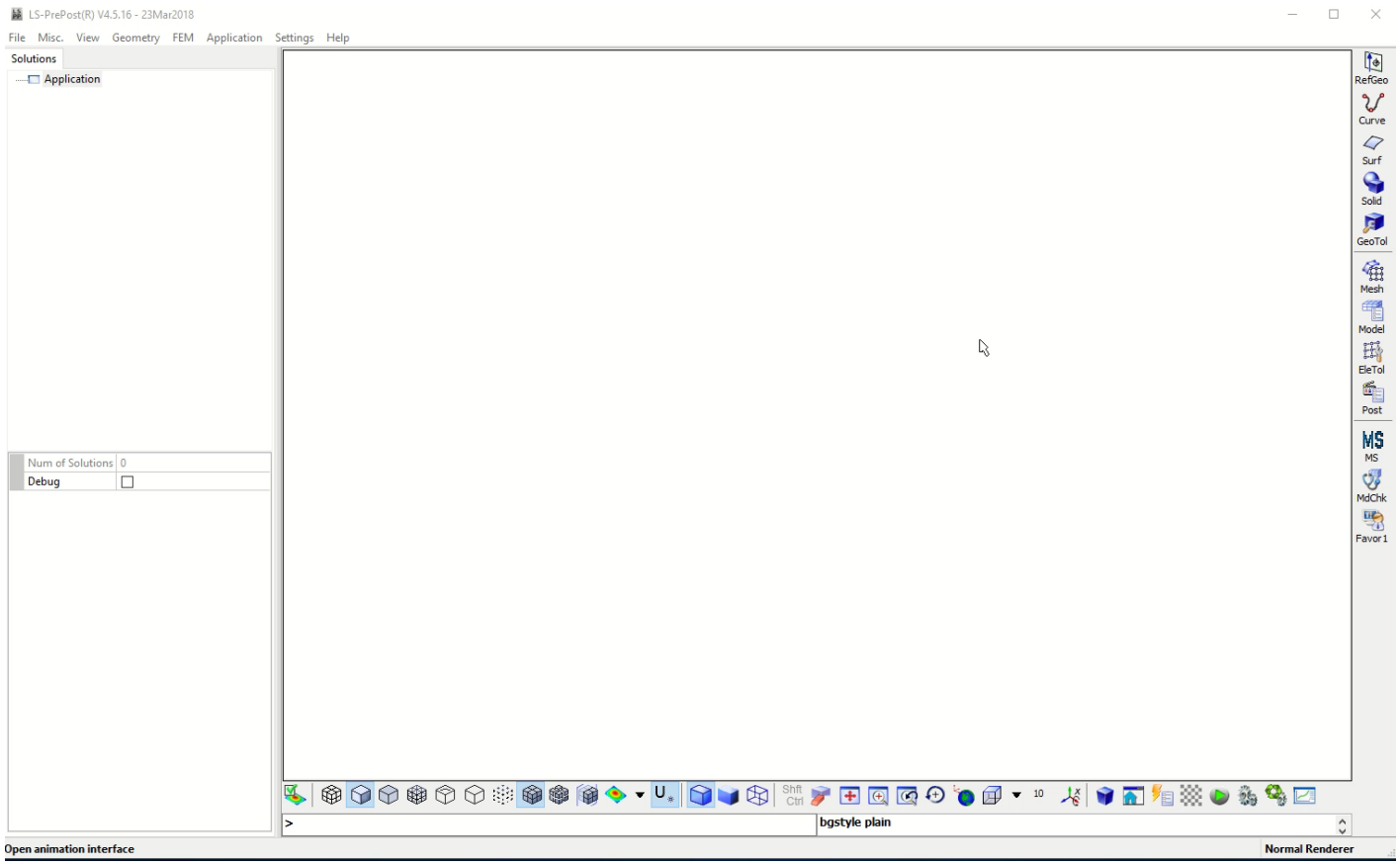
Creating the geometry

Create the cylinder

- Click **Solid**
- Click **Cylinder**
  - **Radius: 0.005**
  - **End Pos: 0.05**
  - Click **Apply** and **Close**

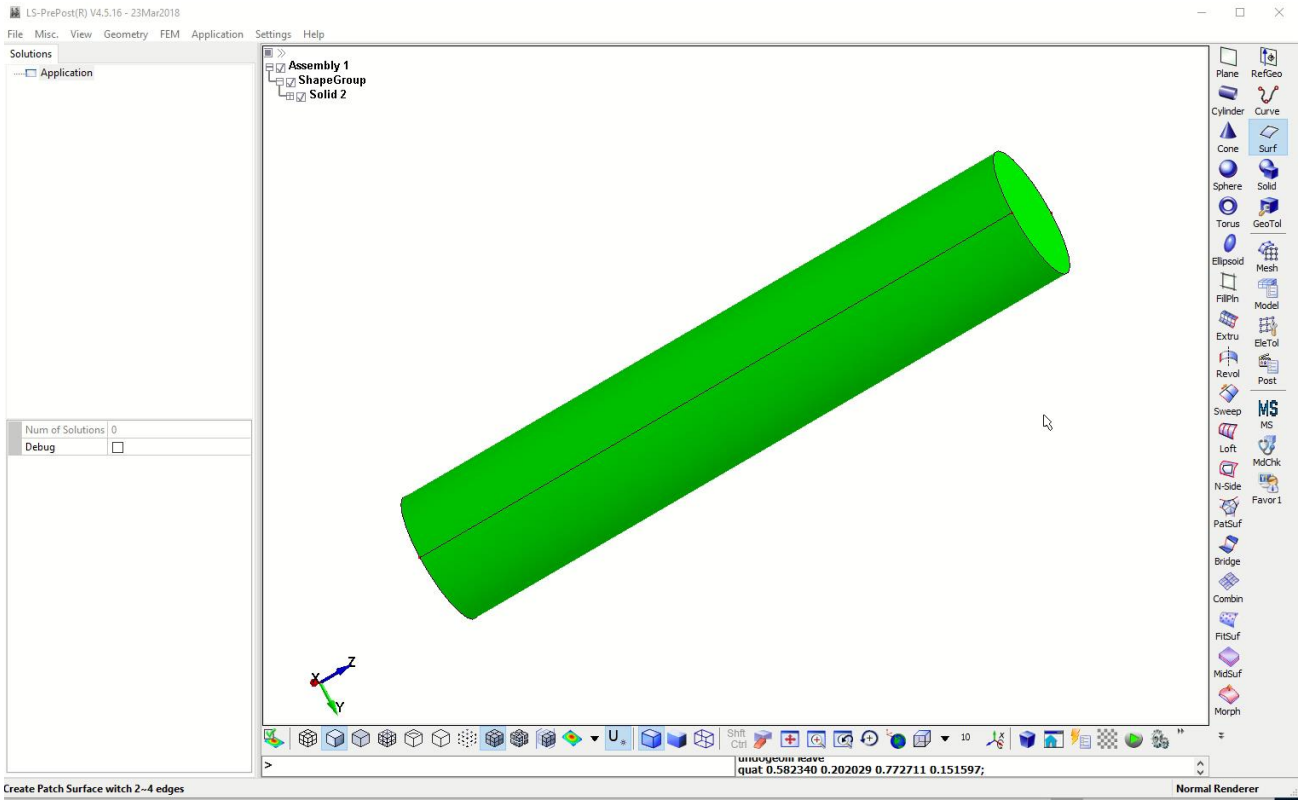
Split the cylinder surface:

- Click **Surface**
- Click **Break Surface**
  - For **U Parameters**, enter **0.5**
  - Clear **V Parameters**
  - Click on the axial edge on the cylinder
  - Click **Apply** and **Close**



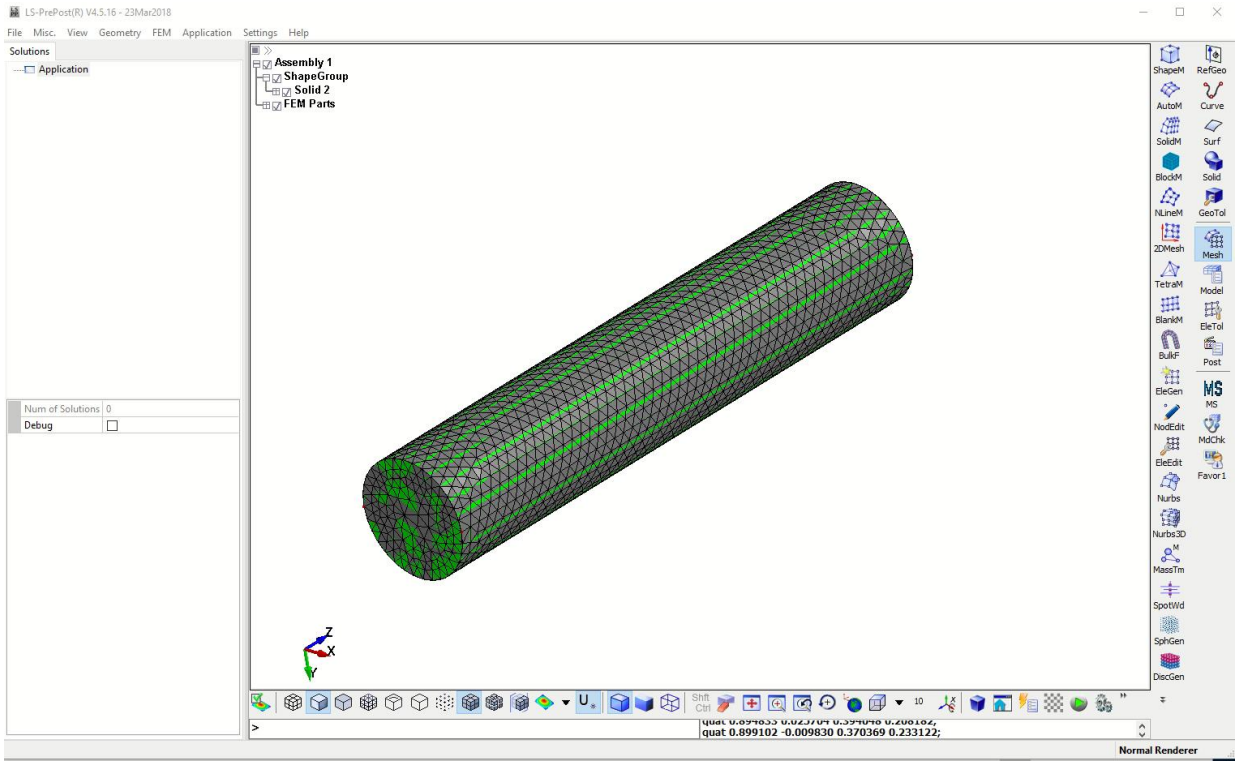
Creating the surface mesh

- Click **Element and Mesh**
- Click **Auto Mesher**
  - Select **Mesh Mode: Size**
  - Select **Mesh Type: Triangle**
  - Select **Elem Size: 0.001**
  - Select **Connect Boundary Nodes**
  - Unselect **Mesh by Gpart**
  - Pick the inlet face
  - Click **Mesh** and **Accept**
  - Pick the outlet face
  - Change **Part ID** to **2**
  - Click **Mesh** and **Accept**
  - Pick the pipe wall faces
  - Change **Part ID** to **3**
  - Click **Mesh**, **Accept** and **Done**



Creating the surface mesh

- Check that there are no duplicate nodes.
  - Click **Element Tools**
  - Click **Duplicate Nodes**
    - Click **Show Dup Nodes**
    - Click **Merge Dup Nodes**
    - Click **Accept** and **Done**
- Rename the parts to *Inlet*, *Outlet*, and *Wall*
  - In the Feature tree, expand **FEM Parts**
  - Rename **LSHELL1** to **Inlet** by right clicking on **LSHELL1** and choosing **Rename**
  - Rename **LSHELL2** to **Outlet**
  - Rename **LSHELL3** to **Wall**



Creating the mesh

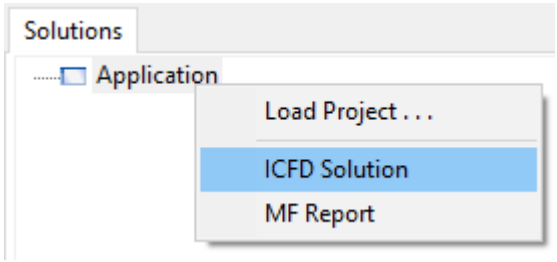
- The next step is to save the mesh.
  - Choose **File** -> **Save** -> **Save Keyword.**
  - Browse to the folder where you want to store the file.
  - As file name, enter **mesh_fluid.k**
  - Click **Save.**

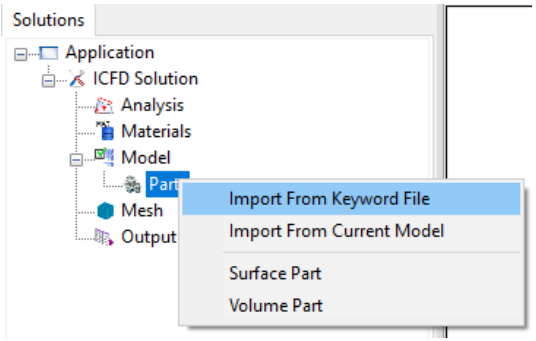
- Converting the mesh to ICFD format.
  - Here, we will convert the mesh manually. Open the mesh file and perform the following steps:
    - Change *ELEMENT_SHELL to *MESH_SURFACE_ELEMENT.
    - Change *ELEMENT_BEAM to *MESH_SURFACE_ELEMENT.
    - Change *NODE to *MESH_SURFACE_NODE.
    - Change *PART to *ICFD_PART_TITLE.

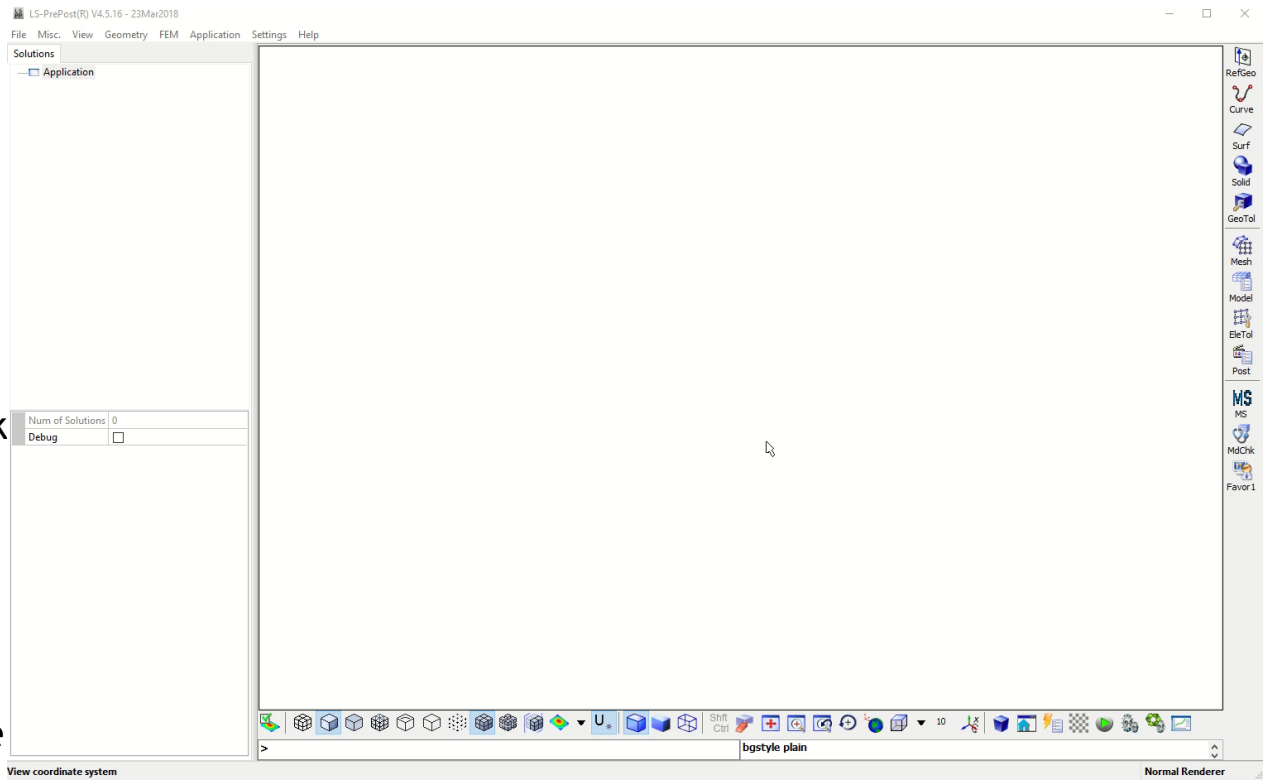
- Now, the mesh has been created, and we can proceed by setting up the CFD solution using the Solution Explorer in LS-PrePost.

Simulation setup

- Open a new LS-PrePost window
  - Make sure that the Solution explorer is visible by selecting **View->Solution explorer**
  - In the solution explorer, right click **Application->ICFD Solution**
  - Under **Model**, right click on **Parts** and choose **Import From Keyword File**. Pick **mesh_fluid.k**
  - The mesh created previously is now visible

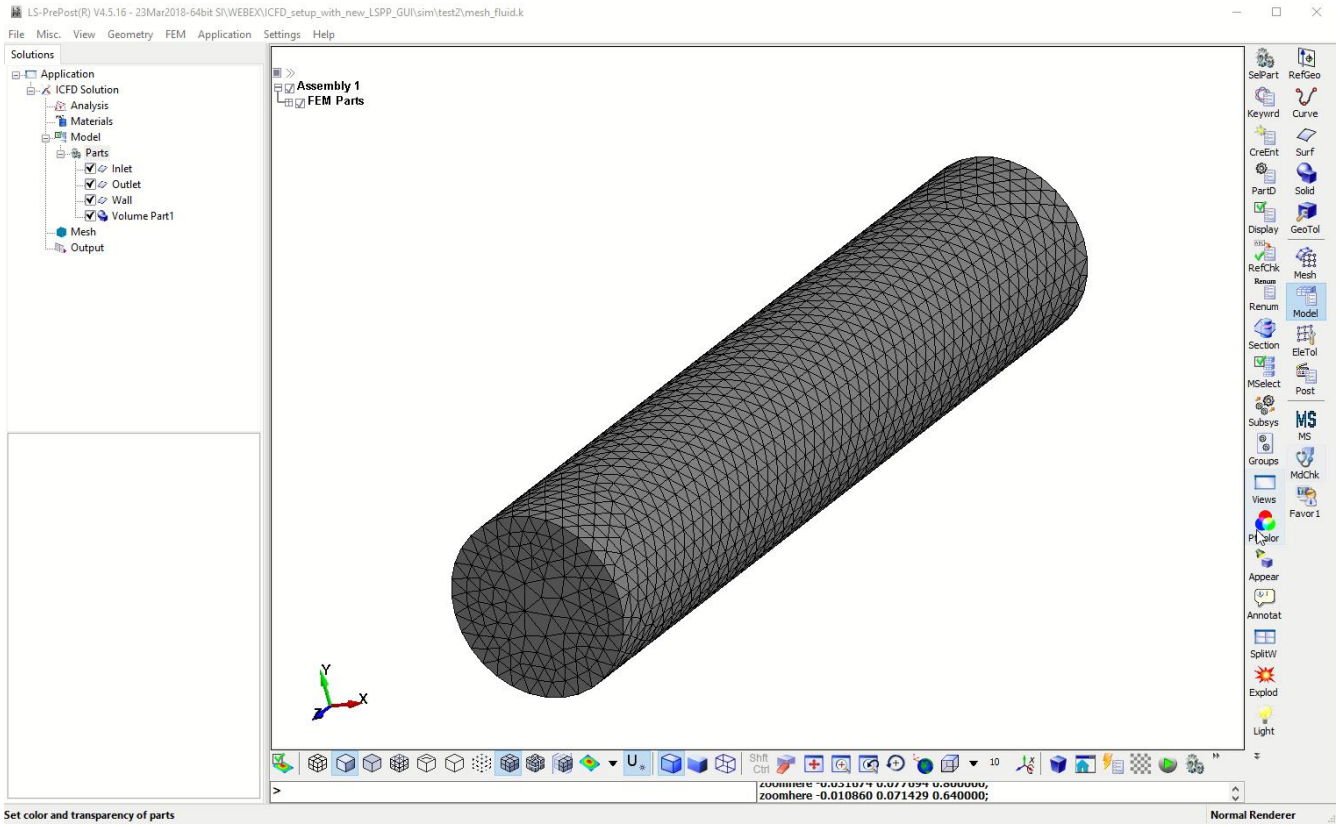






Fluid material

- Right click **Materials** and choose **Create material.**
- Click on the newly created Material1
  - Set the **density** to **1000**
  - Set the **viscosity** to **0.001**
- Alternative: Right click and choose Set As Water.

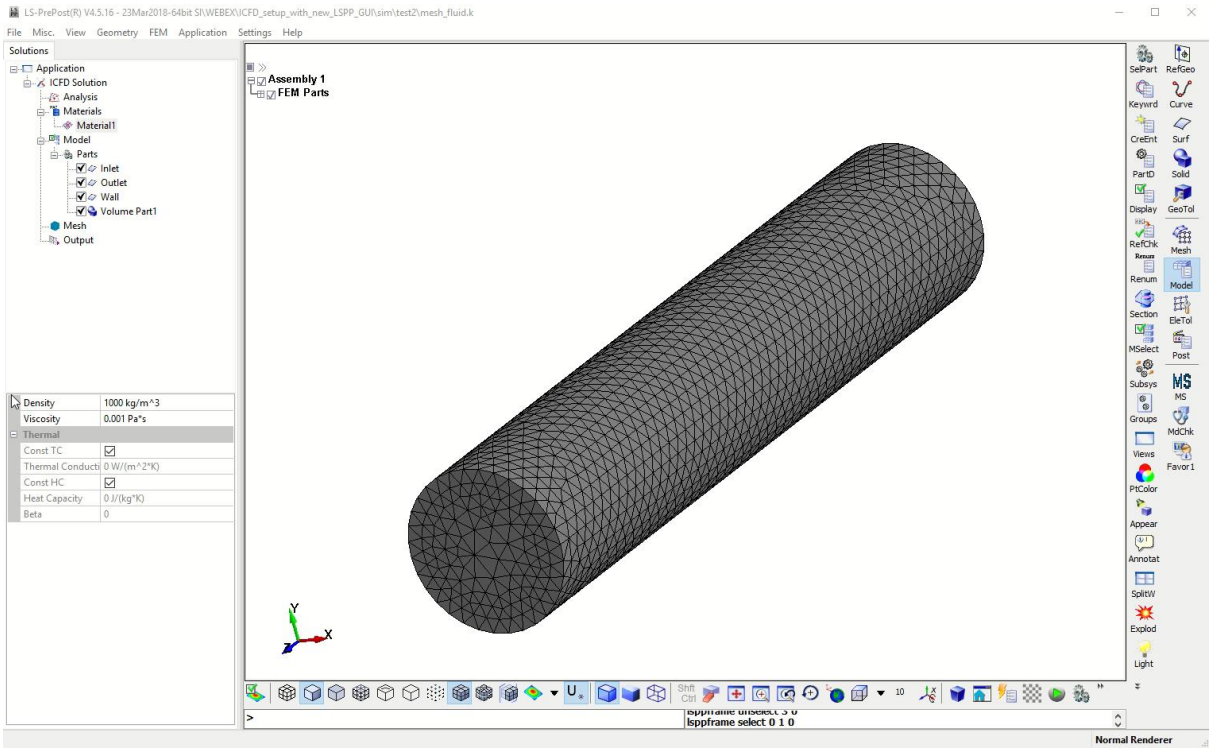


Boundary conditions

- Create boundary conditions:
  - Right click on **Inlet** and choose **Inflow**
  - Set **Value Type** to **Velocity**
  - Set the **Velocity** to **0.1** m/s
  - Right click on **Outlet** and choose **Outflow**
  - Zero pressure outlet is default, so no further changes are needed for the outlet
  - Right click on **Wall** and choose **Wall**
- Setting mesh options:
  - By clicking on Mesh, options for the automatic volume mesher can be set. Here, we will leave the meshing options as default.

Material	Material1
Boundary	
Velocity	☑
Value Type	**Velocity**
Defined	Const
Velocity	**0.1 m/s**
DOF	Normal direction degree of freedom
Advanced Options	☐
Pressure	☐

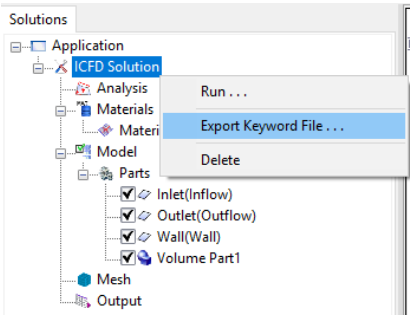
Material	Material1
Boundary	
Velocity	☐
Pressure	☑
Defined	Const
Pressure	0 Pa
Advanced Options	☐



Output and time step

- Click **Output**
  - Set **MSGL** to **full output, copied to messag file**
  - Set **D3plot** output frequency to **0.1**

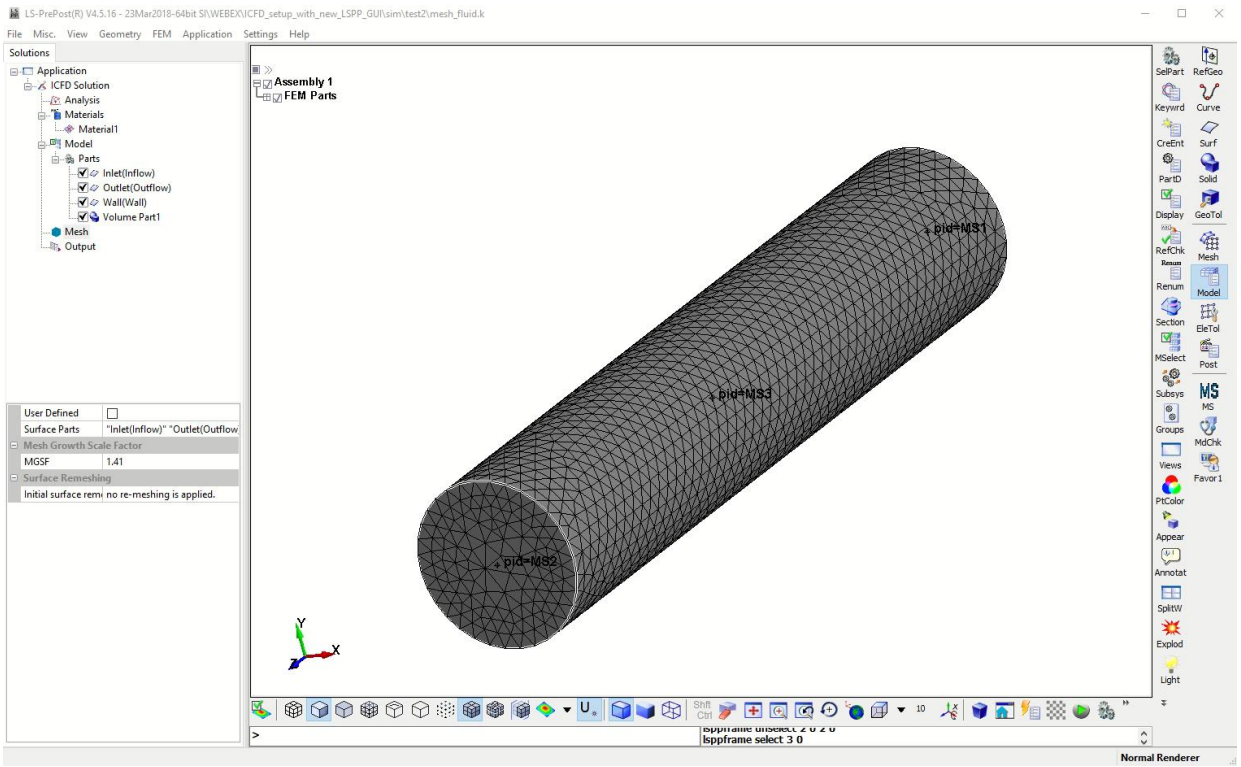
End Time	**1 s**
Unit	SI
Time Step (Fluid)	
Automatic	☐
Time Step	**0.1**
Scale Factor LCID	☐
Min Timestep	0 s
Max Timestep	0 s
Advanced	
Solver	☐



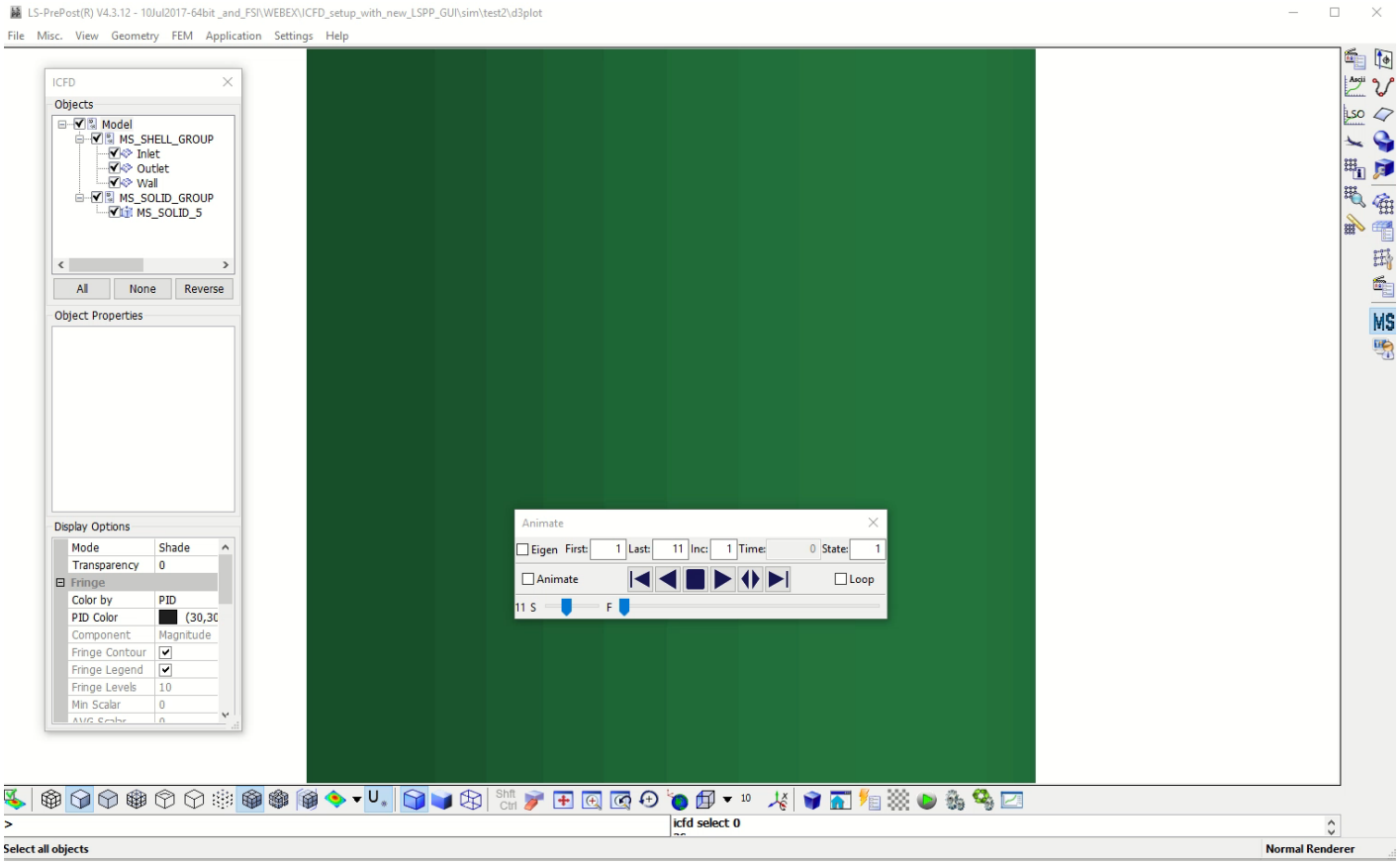
- Click **ICFD Solution**
  - Set **End Time** to **1.0**
  - Unselect **Automatic time stepping**
  - Set the **Time Step** to **0.1**

- Right click **ICFD Solution**
  - **Export Keyword File**
  - Save the file as **main.k**

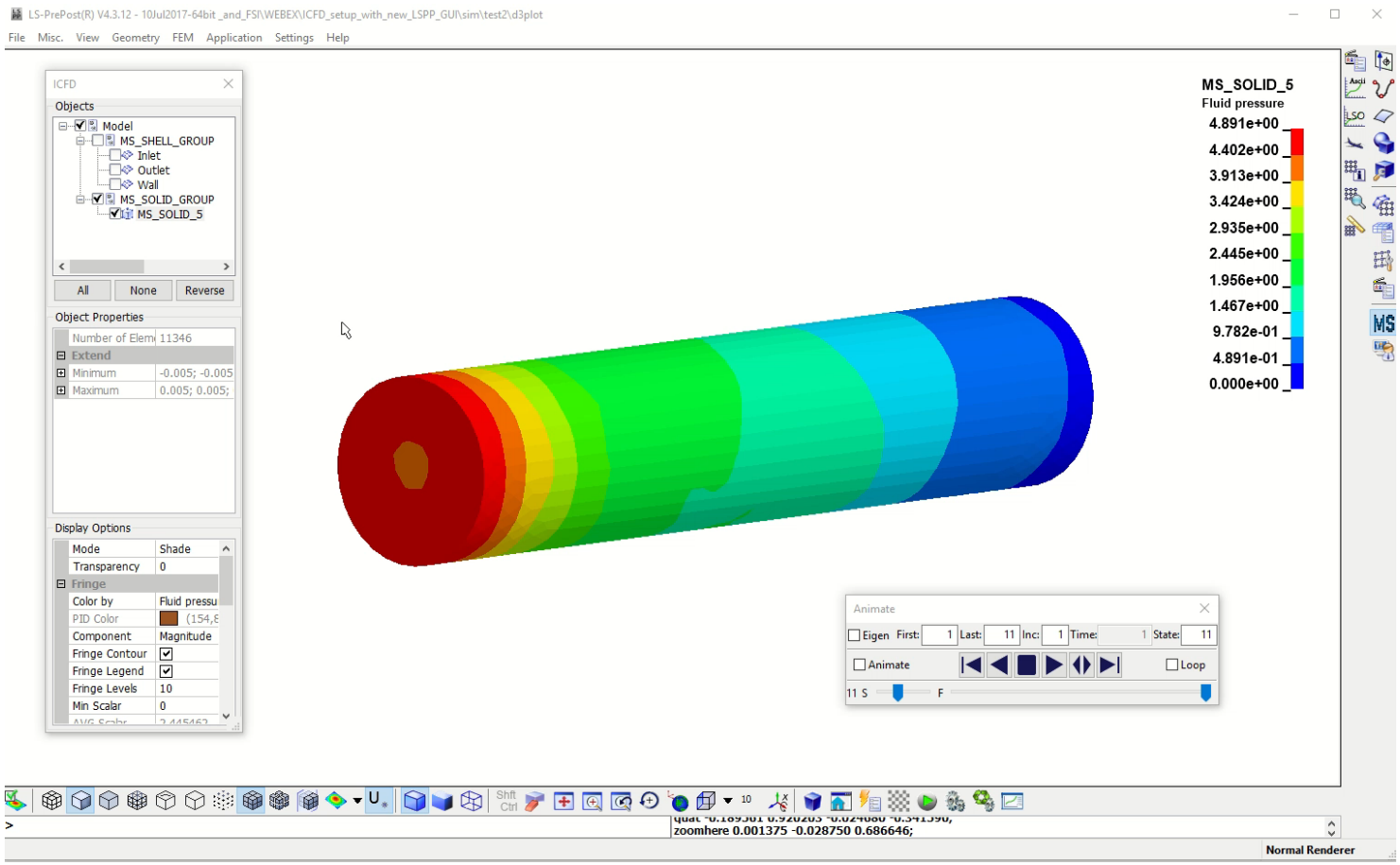
- Run the simulation using LS-Run.



- Open the **d3plot** file with **LS-PrePost**
  - Unmark the **MS_SHELL_GROUP**, so that only the **MS_SOLID_GROUP** is active
  - Highlight **MS_SOLID_5**
  - Change **Color by** to **Fluid pressure**
  - In the **Animate** window, change to the last state
  - Now, we can see the pressure drop along the pipe



- Right click **MS_SOLID_5** and choose **Section**.
  - Unselect **MS_SOLID_5**, so that only the section plane is active.
  - Change **Color by** to **Fluid velocity**.
  - Now, we can see the velocity profile inside the pipe.



Summary

- In this webinar, we have performed an ICFD simulation using the Solution Explorer in LS-PrePost
  - Easier simulation setup compared to the conventional keyword based setup
  - The geometry and mesh still needs to be created as before

- Questions and comments
  - erik.svenning@dynamore.se

Thank you!

DYNA MORE

Your LS-DYNA distributor and more