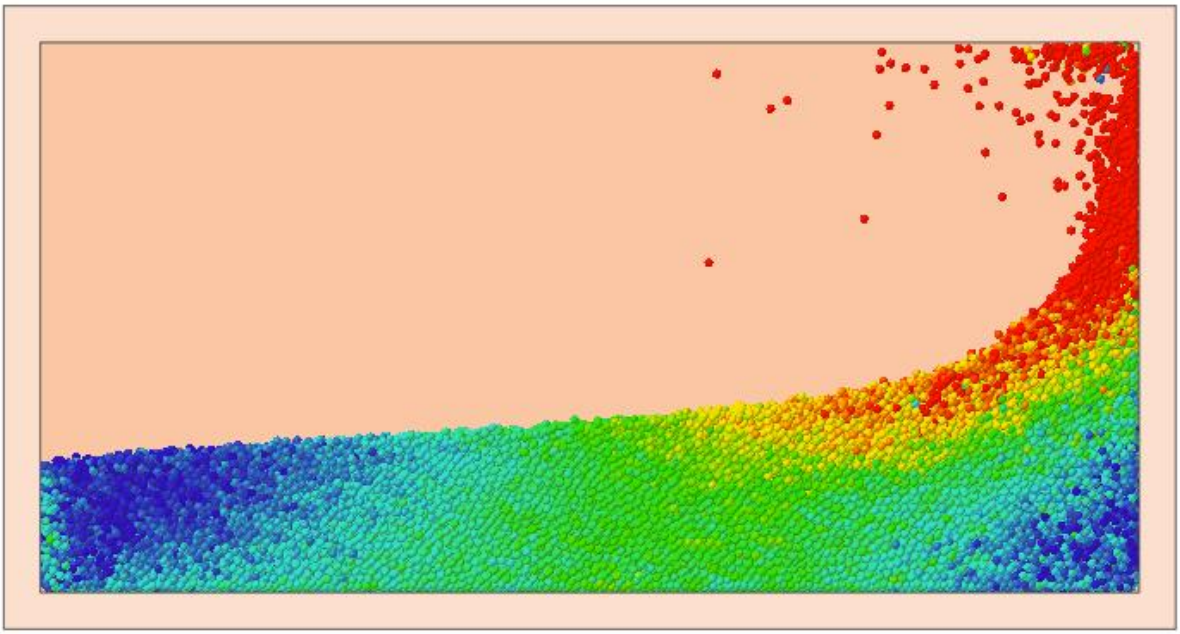
RECURDYN

Water Sloshing (Particleworks)



FunctionBay

Edition Note

This document describes the release information of **RecurDyn V9R4**.

Table of Contents

Chapter 1

Getting Started

In this tutorial, you will learn how to perform a simulation in which **Particleworks** is co-simulated with **RecurDyn**. Co-simulation of **Particleworks** with **RecurDyn** allows fluids to be co-simulated with a multibody dynamics system. **RecurDyn** is used to analyze the MBD (Multi Body Dynamics) system, and **Particleworks** is used to analyze a fluid. The fluid dynamics formulation that **Particleworks** is based on is the MPS (Moving Particle Simulation) method.

The model that is used in this tutorial is based on a model used in a scientific journal paper in which fluid sloshing behavior and pressure in a rectangular tank is analyzed. The issue of sloshing is one of the most studied phenomena of fluid motion. There has been extensive research performed in this area as intense sloshing flow could result in significant damage to a Wall and its structure from the impact caused by fluid. The paper that is used as the basis of this tutorial compares the result analyzed using the CIP (Constrained Interpolation Profile) method, one of the fluid dynamics modeling methods, to the result gained from experimental trials. This tutorial will demonstrate how to compare and analyze the result from the MPS method in **Particleworks** and the results from the paper.

Reference: Numerical simulation of violent sloshing by a CIP-based method, Kishev etal (2006)

Objective

In this tutorial, you will learn how to:

- Create a rigid body model in and export *.wall files from **RecurDyn**
- Create particles in **Particleworks**
- Set fluid properties in **Particleworks**
- Co-simulate **Particleworks** with **RecurDyn**
- Post-process in **Particleworks**

Prerequisites

- This tutorial is intended for users who are familiar with the basic RecurDyn tutorials. Prior knowledge of the aforementioned tutorials will help you better understand this tutorial.
- Particleworks have to be installed to proceed with this tutorial. And this tutorial is progressed with Particleworks 6.2.0 version.
- This tutorial is progressed with the graphic card(NVIDIA GeForce GTX TITAN Black). The computer specification or software version can change the result of simulation.

Procedures

The tutorial is comprised of the following procedures. The estimated time to complete each procedure is shown in the table.

Procedures	**Time (minutes)**
Register Particleworks GUI in RecurDyn	10
Model creation in RecurDyn	15
Model creation in Particleworks	15
Co-simulation	120
Particleworks Post-processing	30
Analysis and review	5
Total	**195**

Chapter

2

Register Particleworks GUI in RecurDyn

By default, the **External SPI (Particleworks)** GUI is not visible in the **RecurDyn**. You must add the GUI to **RecurDyn** using the configuration XML file provided by **Particleworks**.

Task Objective

Learn how to register **External SPI(Particleworks)** tab in **RecurDyn** ribbon using the configuration XML and to set a particle solver DLL.

Estimated Time to Complete

10 minutes

Copy Configuration XML

To copy Particleworks.xml:

Copy the **Particleworks.xml** from install path of **Particleworks** software.

- <Particleworks Install Path>\module\Particleworks.xml

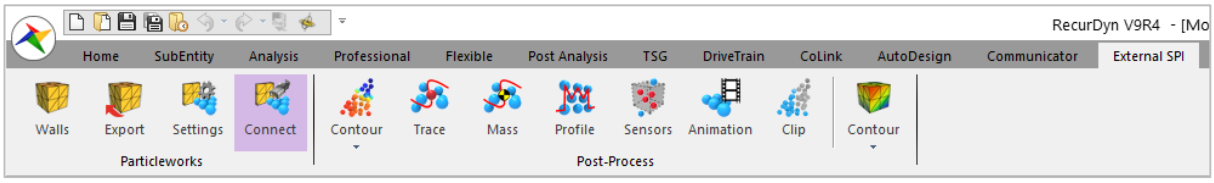
To paste XML in RecurDyn:

Paste the **Particleworks.xml** copied above to **RecurDyn** install path.

- <RecurDyn Install Path>\Bin\Solver\CoSim\StdParticleInterface\Particleworks.xml

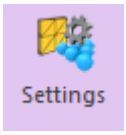
To confirm RecurDyn GUI:

Run RecurDyn and check a ribbon GUI, you can see the External SPI tab and there is a Particleworks group.

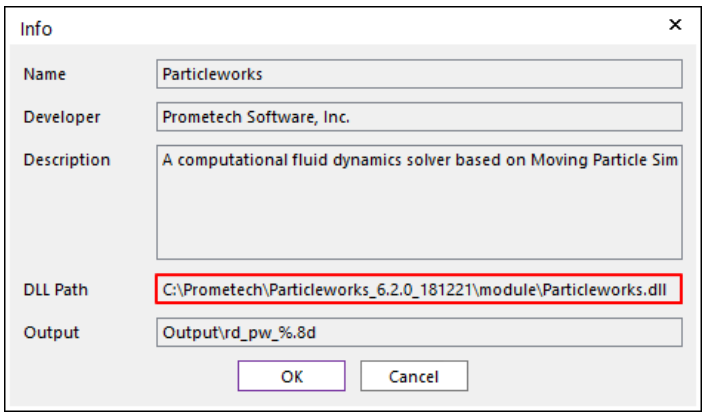


To check a path of particle solver DLL:

Must check where particle solver DLL file should be located.



1. From the **Paticleworks** group in the **External SPI** tab, click **Settings**.
2. In the **Settings** dialog box, click the **Info**.



You can check the **DLL Path** in the **Info** dialog window. This path is set to the default installation location of Particleworks. So if you installed Particleworks in a different path, you must modify the DLL path in the Configuration XML file.

Tip: Modifying the Particle Solver DLL path in Configuration XML file

(Perform it when the DLL path in the Info dialog window is different from the Particleworks installation path.)

1. Open the Particleworks.xml file.
2. Modify the DLL path following **<Path>** as shown below.

```
<?xml version="1.0"?>
<!-- The first letters of names of elements are capitalized and names of attributes are v
<!-- Any element for which support is optional must have the attribute "supported" with t
<!-- If there are multiple supported options of an element, then the element can have an
<!-- The names of elements in this file cannot be changed. -->

<!-- Configuration : required -->
<!-- Requires the attribute "type" and "version"-->
<!-- The value of the attribute "type" must be "Embedded" or "Independent" -->
<!-- External particle solvers should use 'type="Independent"'. -->
<Configuration type="Independent" version="1010">
    <!-- Details : requred -->
    <Details>
        <!-- Name : required -->
        <!-- This text will be used as the name of Ribbon Group icon in RecurDyn to ident
        <Name>Particleworks</Name>

        <!-- Developer : optional -->
        <Developer>Prometech Software, Inc.</Developer>

        <!-- Description : optional -->
        <Description>A computational fluid dynamics solver based on Moving Particle Simul

        <!-- Path : required -->
        <!-- The path of the dll that RD will load to connect to the particle solver -->
        <Path>C:\Prometech\Particleworks_6.2.0_181221\module\Particleworks.dll</Path>
```

3. Save the Configuration XML file.
4. After confirming that the Configuration XML has been modified successfully, restart **RecurDyn**.

Creating a Model in RecurDyn

For co-simulation of Particleworks with RecurDyn, first, it is necessary to create a multibody dynamics model in RecurDyn. Then it is necessary to export certain data from that model that is required in Particleworks. In the second chapter, you will create a multibody dynamic model of the tank in which the fluid will slosh from side to side and then export the necessary files for Particleworks.

Task Objective

Learn how to create a multibody dynamic model necessary for analysis and how to export the necessary files for Particleworks.

Estimated Time to Complete

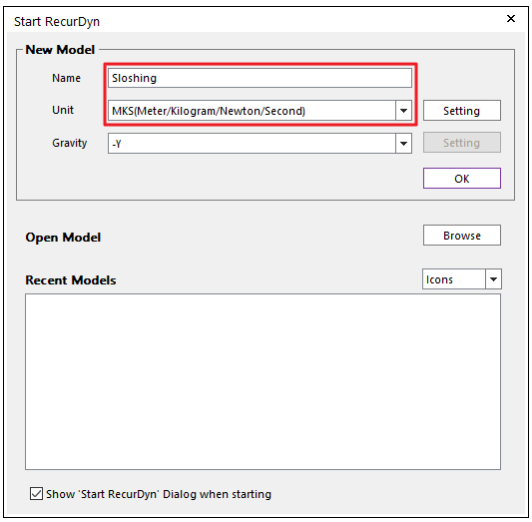
15 minutes

Starting RecurDyn

To start RecurDyn and create a new model:

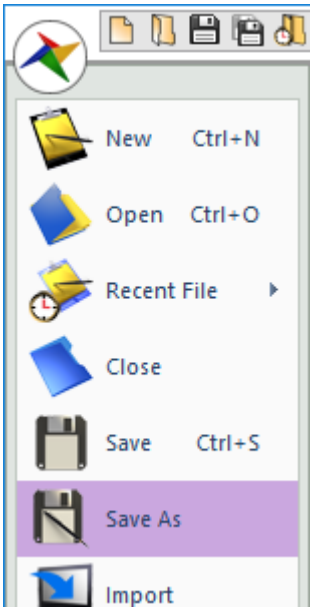
1. On your desktop, double-click the **RecurDyn** icon.

   RecurDyn starts and the **Start RecurDyn** dialog box appears.
2. Enter **Sloshing** for the **Name** of the new model.
3. Set **Unit** to **MKS**.
4. Click **OK**.



To save the model

- From the File menu, click Save As to save Sloshing.rdyn in the desired location.
- (In chapter 3, this model will be moved to the project folder in Particleworks.)



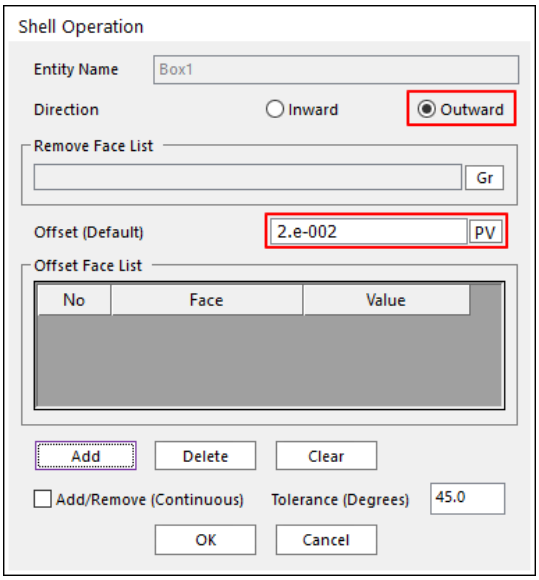
Creating the Geometry

Now, you will create the Container Geometry.

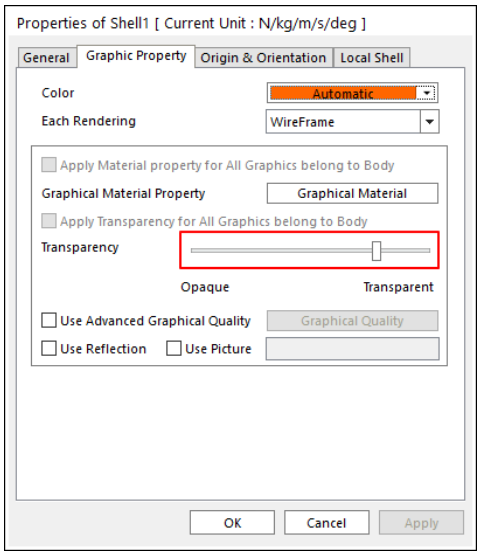
1. Change the **Working Plane** to the **XY Plane**.
2. From the **Body** group in the **Professional** tab, click **Box**.
3. Set the Creation Method toolbar to **Point, Point, Depth**.
4. Enter the following in the **Input Window** toolbar:
   - **Point1: -0.3, 0, 0**
   - **Point2: 0.3, 0.3, 0**
   - **Depth: 0.05**

5. Change the name of **Body1** to **Container**
6. Right-click on **Container** in the database window and then enter Edit Mode for the **Container** body.

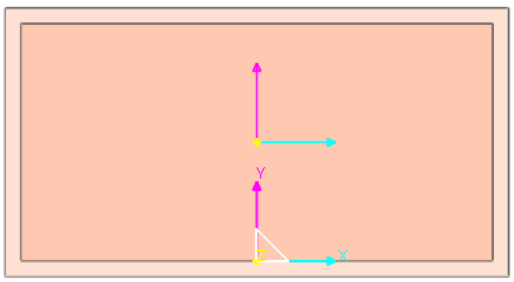
7. From the **Local** group in the **Geometry** tab, click **Shell**.
8. Select **Box1** on the **Working Window**.
9. Display the **Shell Operation** dialog box and do the following as shown in the figure to the right.
   - **Value: 2.e-0.02**
   - **Direction: Outward**
10. Click **OK** to exit the **Shell Operation** dialog box.

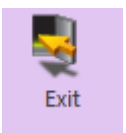


11. Raise the **Transparency** of **Shell1** in the **Graphic Property** tab in the **Properties of Shell1** dialog box.



You will see that the shell geometry of 0.02m in thickness has been created.

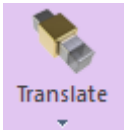




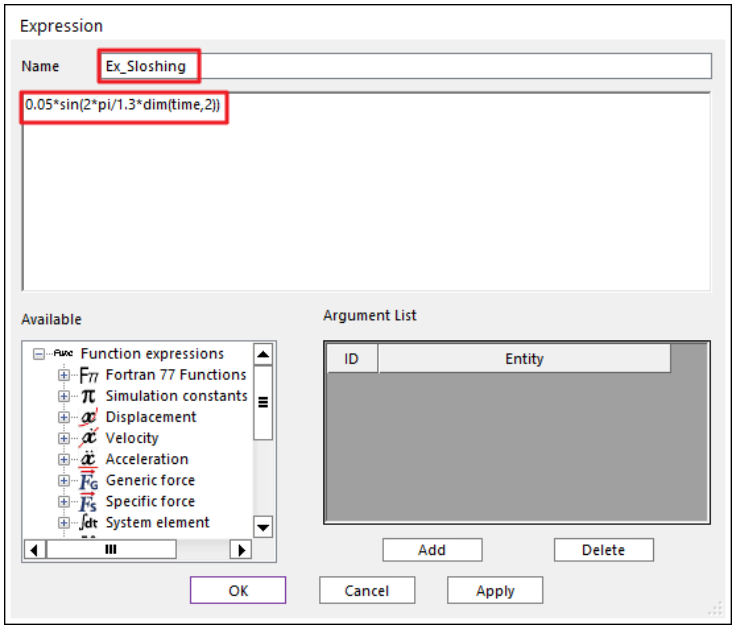
12. Exit Edit Mode.

Creating the Translational Joint

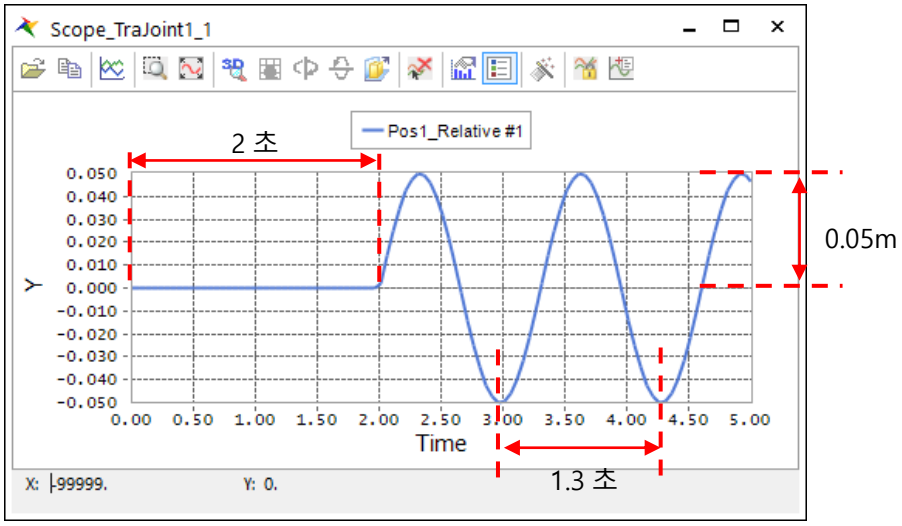
Now, you will create the joint to swing the **Container** body from side to side.



1. From the **Joint** group in the **Professional** tab, click **Translate**.
2. Set the Creation Method toolbar to **Body, Body, Point, Direction**.
3. Enter the following in the Input Window toolbar:
   - **Body 1: Ground**
   - **Body 2: Container**
   - **Point: 0, 0. 15, 0**
   - **Direction: 1, 0, 0**
4. Display the **Property** dialog box of **TraJoint1**.
5. Check the **Include Motion checkbox**, and then click the **Motion** button.
6. Set **Displacement (Time)** as type, click the **EL** button to enter the expression.
7. Enter the following by clicking **Create** to create an expression.
   - **Name: Ex_Sloshing**
   - **Value: 0.05*sin(2*pl/1.3*dim(time,2))**

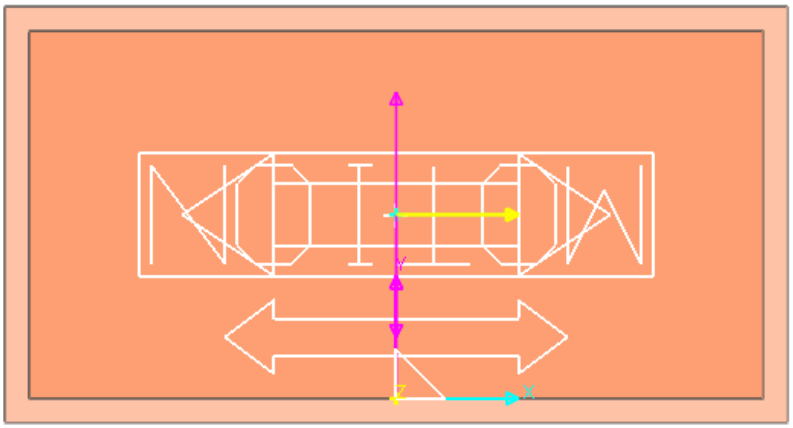


The expression creates a motion of the translational joint as shown in the figure below, with an amplitude of 0.05m and a period of 1.3 seconds starting from 2 seconds. The initial delay of 2 seconds is used for the purpose of allowing the fluid particles in the tank to settle before the forced oscillations begin.



8. Click **OK** to close all the dialog boxes.

Now you have completed the creation of the all necessary RecurDyn elements for a simulation that does not include co-simulation with Particleworks. Next, you will create what is called a Wall in RecurDyn, which is required for co-simulation with **Particleworks**.

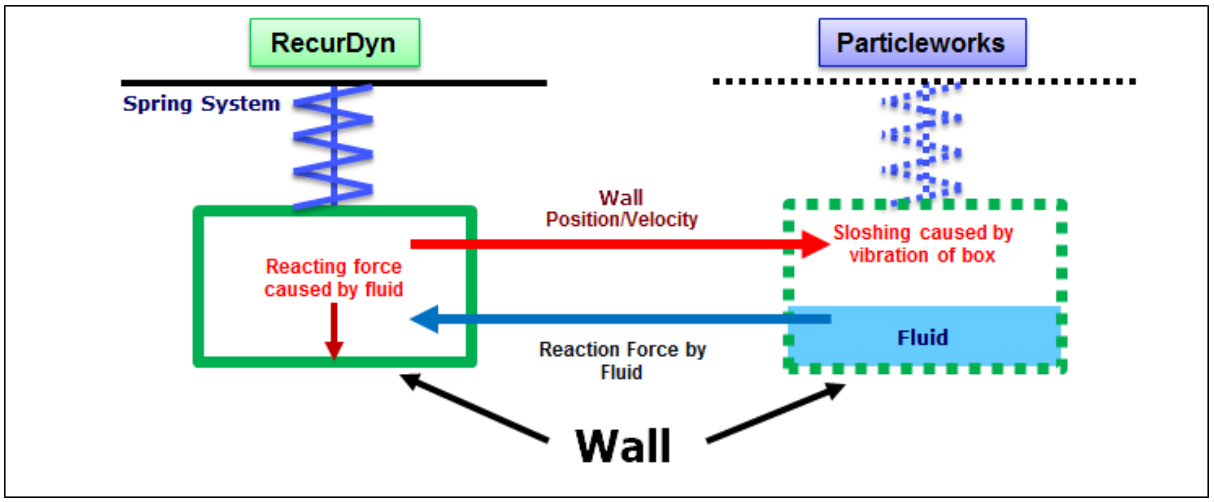


Creating the Wall

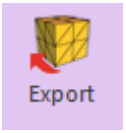
Next, you will create a Wall on the **Container** body you just created.

1. From the **Particleworks** group in the **External SPI** tab, click **Walls**.
2. Click the **Container.Shell1** on the **Working Window** as solid.

Note: What is Wall?
A Wall is a rigid body surface in a RecurDyn model that can make contact with the fluid in Particleworks during co-simulation. The Wall is the primary element of interaction between RecurDyn and Particleworks. RecurDyn passes the positions and velocities of the Wall surfaces to Particleworks. In Particleworks, the surfaces can make contact with the fluids. Particleworks computes the forces of interaction between the surface and the fluid. These forces are then sent back to RecurDyn to influence the motion of the body to which the Wall surface is attached.



Exporting the *.wall



1. From the **Particleworks** group in the **External SPI** tab, click **Export**.
2. Display the dialog box to save the file by clicking **OK**.

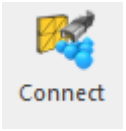
   The **rd_pw.wall** file and **WallGeometries** folder are created in saved folder. These will be moved to the Particleworks project folder created in chapter **4.**

Note:
In the **WallGeometries** folder, the geometry files (*.obj or *.stl) for Wall is saved. And in the **rd_pw.wall** file there is information about the position of each wall geometries.

Analyzing the RecurDyn Model

Executing the Simulation Without Co-Simulation:

Before running the co-simulation between **RecurDyn** and **Particleworks**, you may want to verify that the model which you just created works properly. Since **RecurDyn** will automatically run co-simulation with **Particleworks** when there is a Wall in the RecurDyn model, you should change several options to allow RecurDyn to execute the simulation without co-simulation with Particleworks.



1. From the **Particleworks** group in the **External SPI** tab, uncheck the **Connect** icon.

2. From the **Simulation Type** group in the **Analysis** tab on the main RecurDyn ribbon control, click **Dyn/Kin**.
3. Enter the following:
   - **End Time**: 14
   - **Step**: 1400
4. Click the **Simulation** button to analyze.

To display an animation of the simulated motion:

- After the completion of analysis, click **Play** on the **Analysis** tab of the main **RecurDyn** ribbon control to view the animation.
- You can animate the Container body moving from side to side. The modeling in RecurDyn has been completed. Now you will exit RecurDyn after saving the model, then proceed to modeling in Particleworks.

Chapter

4

Creating a Model in Particleworks

Fluids are modeled in **Particleworks** using a particle method, where each particle represents a certain mass of fluid. Creating a model in Particleworks involves defining a collection of these fluid particles. There are various methods for defining the particles in Particleworks. In this tutorial, you will use the method of filling particles from the defined plane to the boundary surface of a Wall.

Task Objective

Learn how to create fluid particles, enter a property of matter, and configure the analysis environment.

Estimated Time to Complete

15 minutes

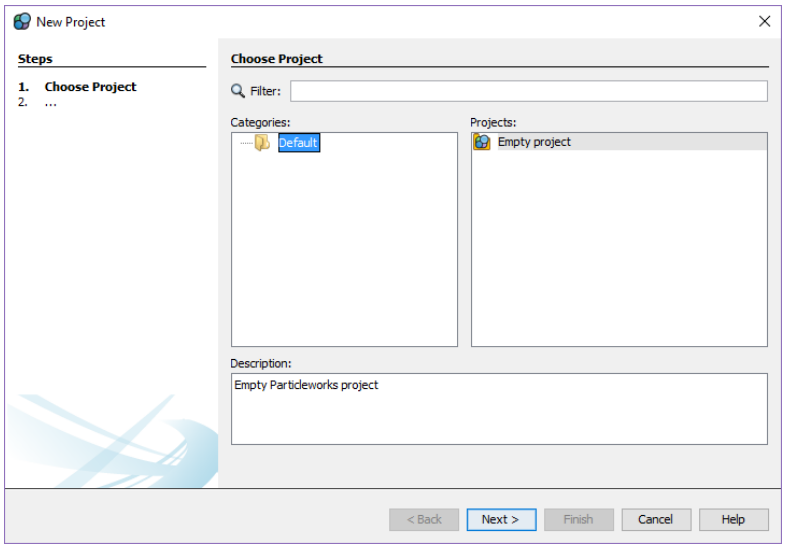
Starting Particleworks

To create a new model:

1. On your desktop, double-click the **Particleworks** icon.

2. Click **New Project** from the **File** tab.

   Display the **New Project** dialog box.



3. Click **Next**.
4. Enter **Sloshing** for the name of the **Project Name**.
5. Specify the folder in which the project files will be created in **Project Location**.
6. Click **Finish**, and then Particleworks will create the new model that you specified.

Copy the RecurDyn files:

The **RecurDyn** model and **Wall** files should be manually copied into the **Particleworks** project folder for co-simulation.

- You must move the **Sloshing.rdyn** file, **rd_pw.wall** file and WallGeometries folder which you just created in chapter 3, to the scene folder for the Particleworks project.
- (Folder path: <ProjectLocation>/Sloshing/scene)

Setting up the Preprocess

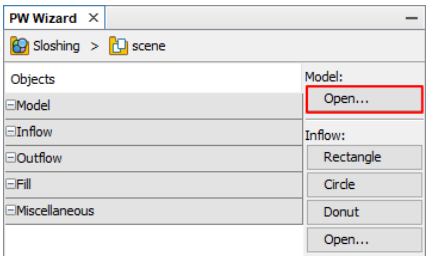
Preprocess is the process to define the overall system of modeling from the particle creation to the environment settings.

To import the Wall from the Wall file:

1. Click **Start wizard** from the **Simulation** tab.

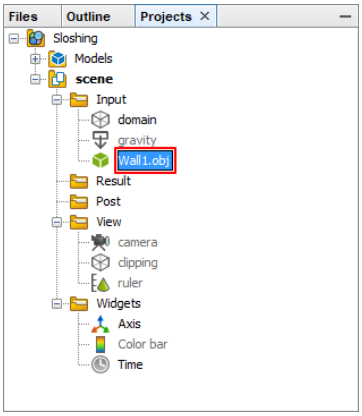
   The **PW Wizard** dialog box appears on the right side.

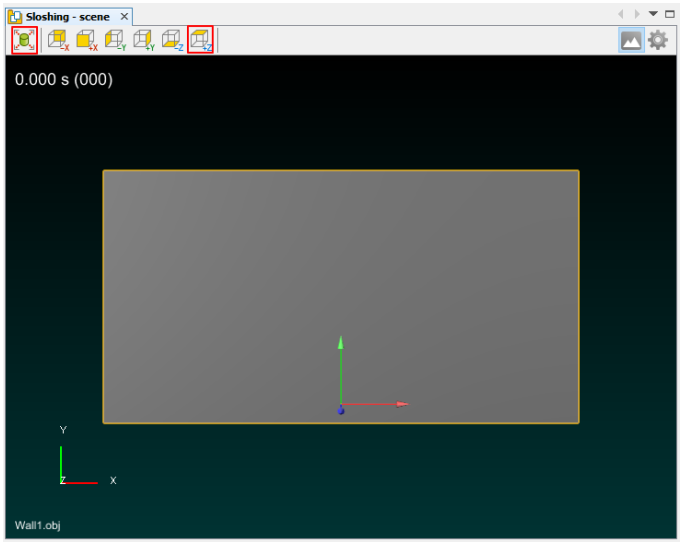
2. Click the **Open** button to import **rd_pw.wall** file.
   - <ProjectLocation>/Sloshing/scene/rd_pw.wall)



3. Click **Fit** and **+Z** to take a close look at the Wall which you just imported.

   If the Wall has been properly imported, then an item titled **Wall1.obj** will be visible in the database tree under Sloshing > scene > Input, as shown in the figure below. The imported Wall should be visible in the **Working Window** as well.



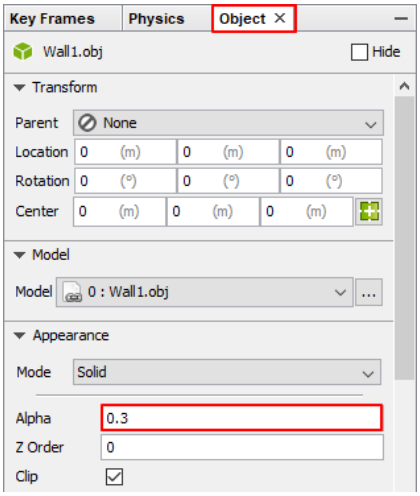


To set the transparency of a Wall:

It is recommended that you set the **Wall1.obj** object to be transparent. This will be useful when creating the fluid particles.

1. Double-click **Wall1.obj** on the **Projects** dialog box (scene>Input).

   Display the **Object** dialog box about the entity you have clicked.



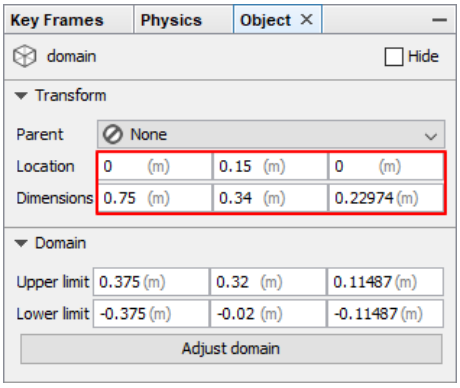
2. Change **Alpha** to **0.3** under the **Appearance** and press the enter key.

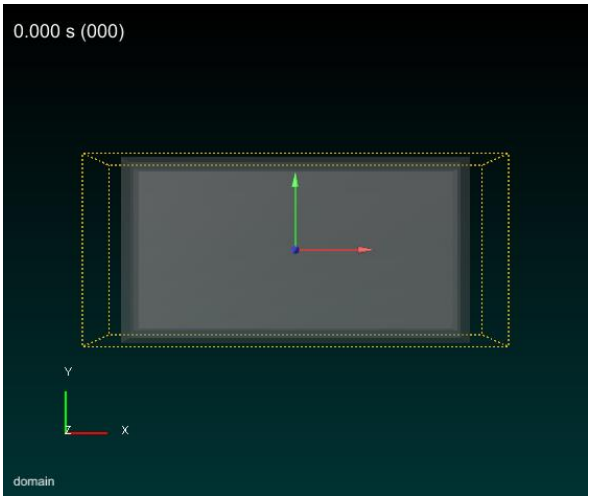
   You can see that the **Wall1.obj** object becomes transparent.

To set the domain:

The **Domain** in Particleworks signifies absolute volume of space in which the fluid particle can be permitted to move during the fluid simulation in Particleworks. This volume should include the entire volume of space that the **Wall1**.obj object is expected to pass through during the simulation. It is acceptable for the Domain volume to be larger than the volume through which the Wall1.obj actually passes through during the simulation.

1. Double-click on **domain** in the **Projects** database tree. (Sloshing > scene > Input)
2. Enter the following in the **Location** under the **Transform**.
   - **Location: 0, 0.15, 0**
3. Enter the following in the **Dimensions** under the **Domain**.
   - **Dimensions: 0.75, 0.34, 0.22974**



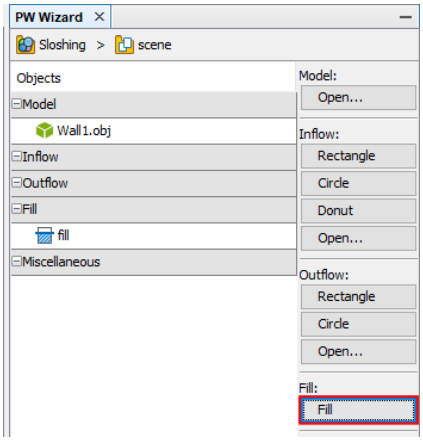


To create the particles:

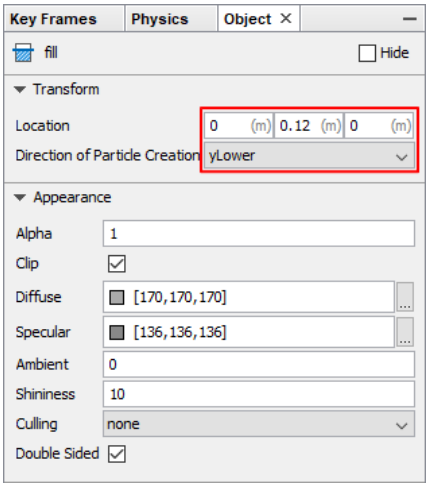
Fill has a function of filling both the defined plane and closed space of **Wall1** with particles. The plane is limited to the inside of the domain.

1. Click **Fill** in the **PW Wizard** dialog box.

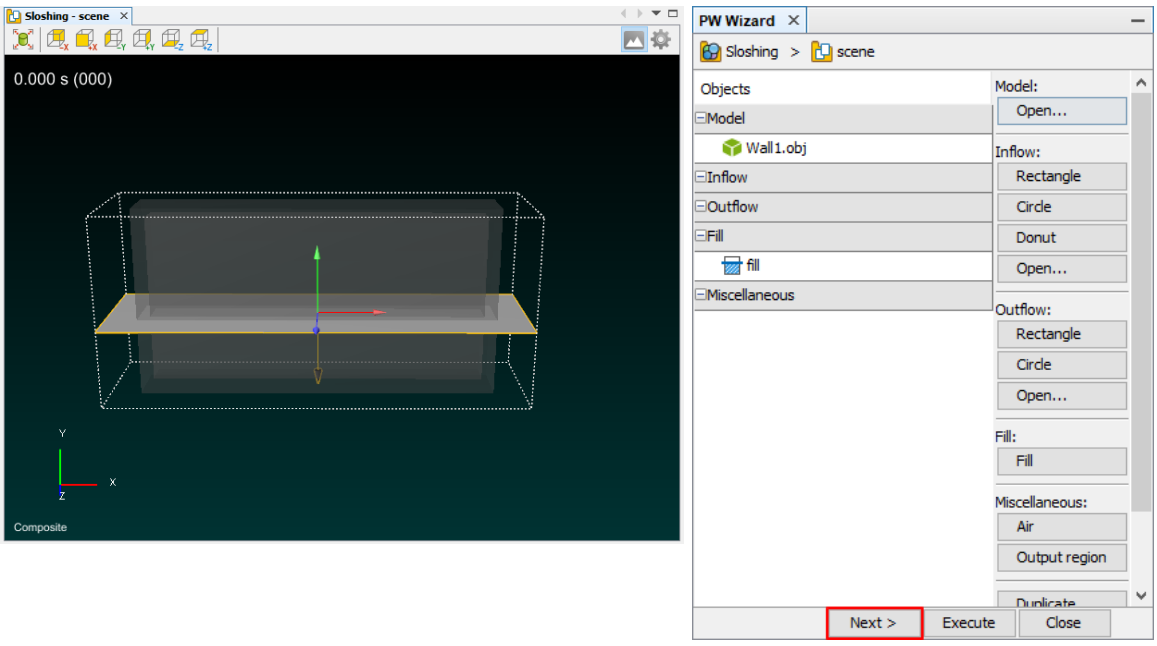
   An item titled **Fill** will be created in the **Projects** database tree. (Sloshing > scene > Input)



2. Double-click **Fill** in the **Projects** database tree.
3. Enter the following under the **Transform** in the **Object** dialog box.
   - **Location: 0, 0. 12, 0**
   - **Direction of Particle Creation: yLower**

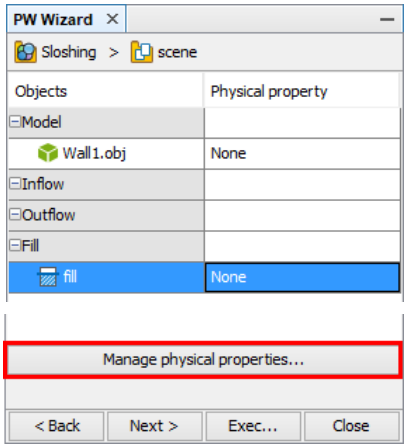


4. Click **Next** in the **PW Wizard** dialog box to go on to the next page after verifying that **Fill** is defined as shown in the figure below.



To create and set the physcial properties of the fluid:

1. Click **Manage physical properties...** to create the property of matter in the **PW Wizard** dialog box.



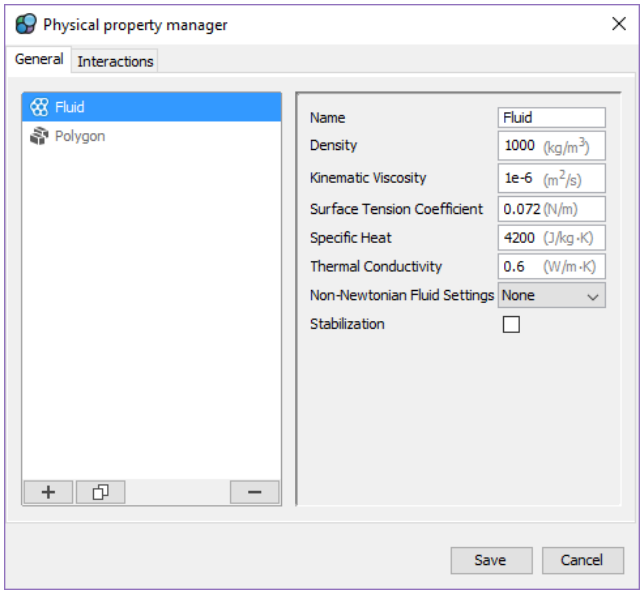
   The **Physical property manager** dialog box appears.

2. Click **+** to create **Fluid**.

   The default properties are the physical properties of water. Use these properties for the tutorial model.

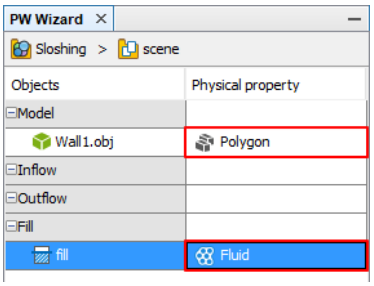
3. Click **+** to create **Polygon**.

   Polygon is used to define the material properties of the contact surface of the Wall1.obj object.

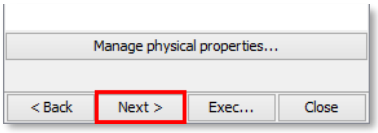


4. Click **Close** to exit the **Physical property manager** dialog box.

5. Enter the following in the **Physical property** in the **PW Wizard** dialog box from None to:
   - **Wall1.obj: Polygon**
   - **Fill: Fluid**



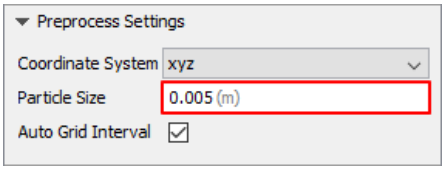
6. Click **Next** in the **PW Wizard** dialog box.



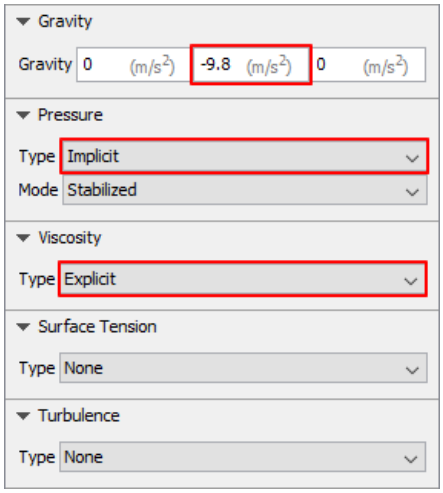
To configure the particles and environment:

You will change the environment options in the **PW Wizard** dialog box.

1. Enter **0.005** in the **Particle Size**.
2. Click **Next** in the **PW Wizard** dialog box to go to the next page.

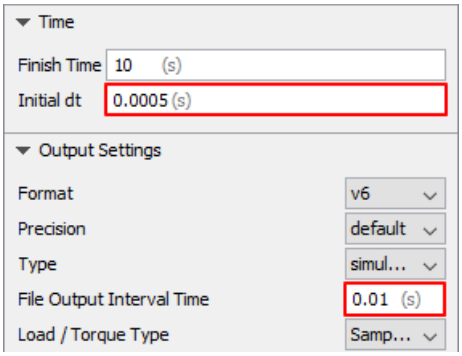


3. Enter **-9.8** in the **Y** of **Gravity**, the same value with **RecurDyn**.
4. Enter the following under the **Pressure**:
   - **Type: Implicit**
   - **Gamma: 0.5**
5. Select the **Explicit** option under the **Viscosity**.
6. Click **Next** to go to the next page.
7. Click **Next** on the **Thermal** page without any change to go to the next page.



To configure the analysis conditions:

You will change analysis options in the **PW Wizard** dialog box.



1. Enter **0.0005** in the **Initial dt[s]** under the Time.

   The above value signifies the initial step size and the maximum step size. It also performs same role as **Initial Time Step** and **Maximum Time Step** in **RecurDyn**.

2. Enter **0.01** in the **File Output Interval Time[s]** under the **Output Settings**.

   The above option value signifies time (second) interval between steps. This value should be identical with that of **EndTime/Step** in **RecurDyn**.

To create the particles:

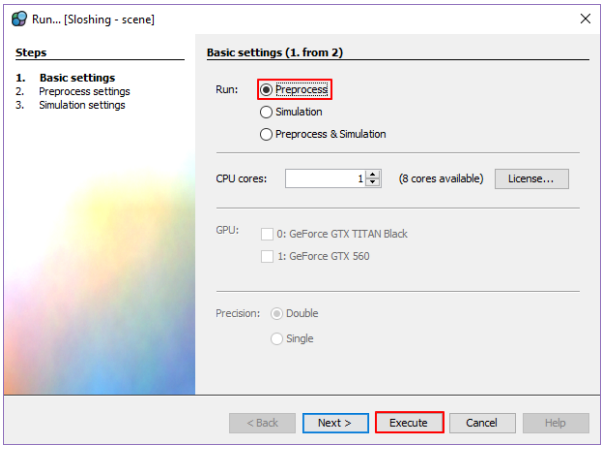
1. Click **Execute** after completing the simulation configuration.

   Display the **Run** dialog box.

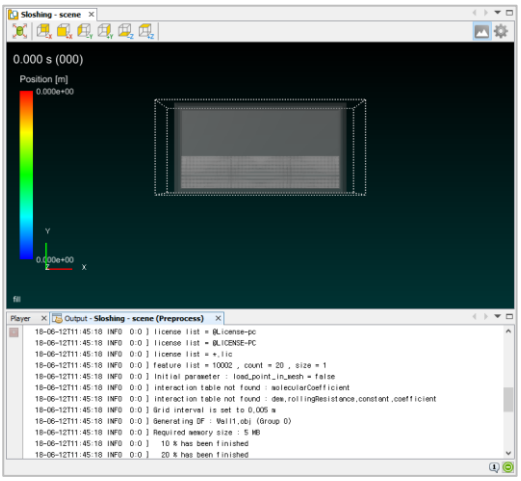
2. In the category Run, select the radio button **Preprocess**.

3. Click **Execute**.

   The fluid particles in the **Wall1.obj** object will be created after clicking the Execute button. They should look like the image. A message will be displayed in the Output window indicating that the particles have been created.



The necessary files for co-simulation will be created in the **df, pre** folder in the project folder (<ProjectLocation>/Sloshing/scene) during the analysis.



Preparing for Co-Simulation with RecurDyn

A number of files that are generated by Particleworks are required by RecurDyn for the co-simulation of RecurDyn with Particleworks. These files are automatically generated by Particleworks when Particleworks executes a simulation independent of RecurDyn. Therefore, it is required that you run a simulation first in Particleworks using the model that you have just created, but without co-simulation with RecurDyn.

Simulation in Particleworks Independent of RecurDyn:

1. Click **Run** in the **Simulation** tab.

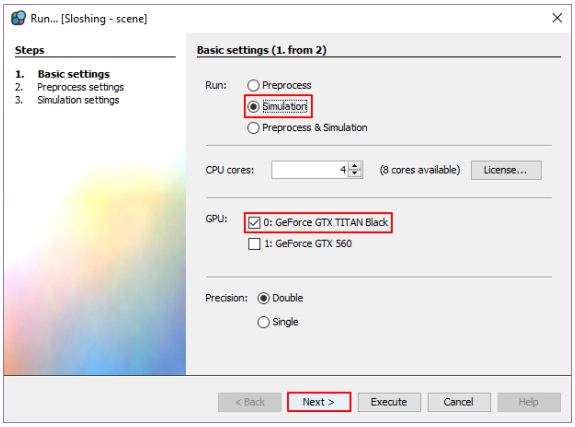
   The **Run** dialog box appears.

2. In the category Run, select the **Simulation** radio button.

3. Enter the number of **CPU cores** that you wish to use during this simulation.

   Compatible GPUs for your computer will be listed in the GPU category on this page of the Run dialog box. If you wish to use one of the GPUs for this simulation, select the GPU from this list.

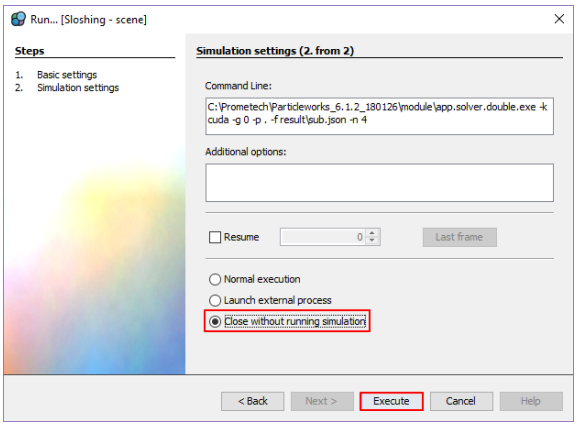
4. Click **Next**.



5. Click **Close without running simulation** radio button.

6. Click **Execute**.

   All the necessary files for co-simulation will be created in the **result** folder in the project folder (<ProjectLocation>/Sloshing/scene) during the analysis



   Now you are ready for co-simulation between RecurDyn and Particleworks.

7. Save the project.

It is not necessary to run **Particleworks** while you are working on chapter 5. But in the next chapter, you should run Particleworks to check the result in real time.

Chapter

5

Co-Simulation

In this chapter, RecurDyn and Particleworks will run co-simulation for behavior analysis between rigid bodies and particles.

Task Objective

Learn how to run co-simulation in RecurDyn.

Estimated Time to Complete

120 minutes

Co-Simulation

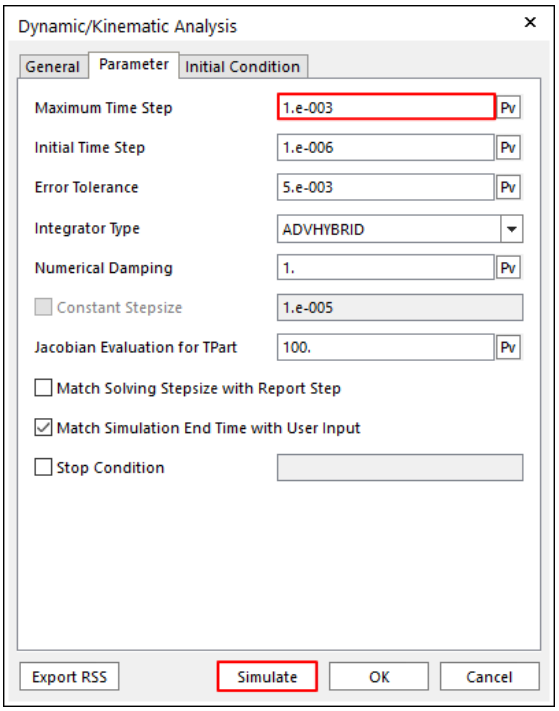
To run co-simulation in RecurDyn:

1. Start **RecurDyn** and open **Sloshing.rdyn** file which you just copied in chapter 3.
   (File path :<ProjectLocation>/Sloshing/scene/Sloshing.rdyn)

2. From the **Particleworks** group in the **External SPI** tab, check the **Connect** icon.

3. From the **Simulation Type** group in the **Analysis** tab of the main RecurDyn ribbon, click **Dyn/Kin**.
4. In the **Parameter** tab, enter **1.e-003** in the **Maximum Time Step** input field.
5. Click the **Simulation** button.

   RecurDyn will start a co-simulation with Particleworks. (The estimated time depends on the specifications of both CPU and GPU.)



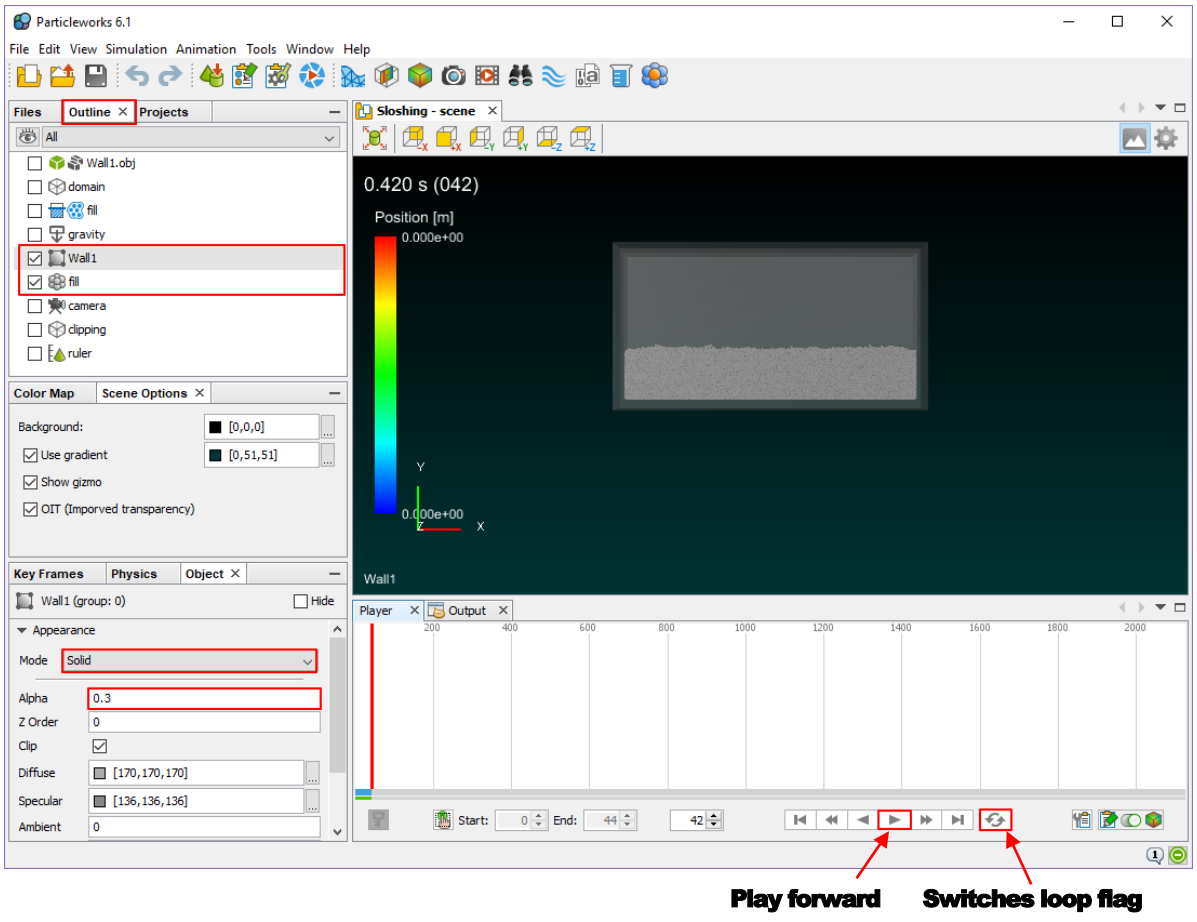
To view the progress of analysis:

You can view the progress of analysis of the fluid particles being simulated in Particleworks through the Particleworks GUI.

1. Select the **Wall1** and **fill** checkboxes in the **Outline** dialog box.
2. Click the **Wall1** in the **Outline** dialog box.
3. Change the below options in the **Object** dialog box of **Wall1**.
   - **Mode: Solid**
   - **Alpha: 0.3**
4. Turn off the **Switches loop flag** option in the **Player** dialog box.

   The **Switches loop flag** option is for endless loop of animation.
5. Click the **Play forward** button in the **Player** dialog box.

   You can view the animation of completed time steps up to the present time.

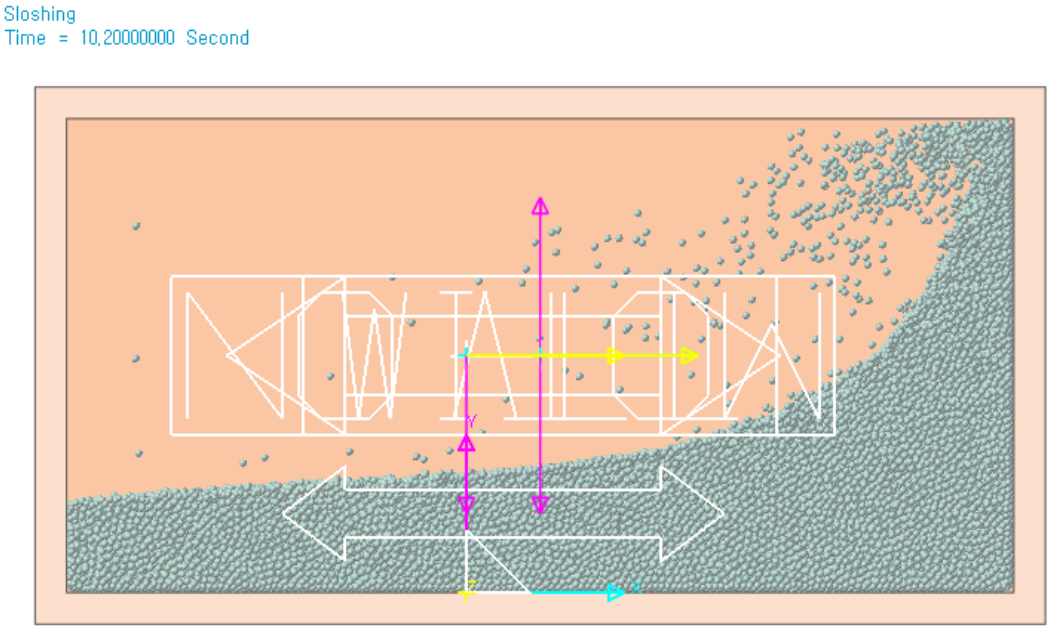


Working with Animations

To play an animation:

- From the **Animation Control** group in the **Analysis** tab, click **Play/Pause**.

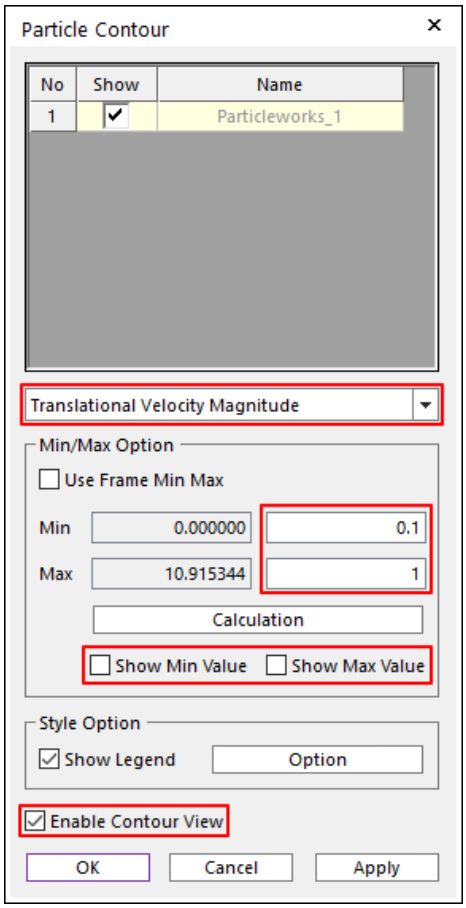
The Wall motion results in both the generating of reaction force and the particle behavior by the impact between the particles created inside of the Wall and the inside walls of the Wall. In the beginning of area of shaking the Wall, the period form of particles is irregular. But you will notice that the particles gradually start moving with a regular period form as the cycle is repeated several times.



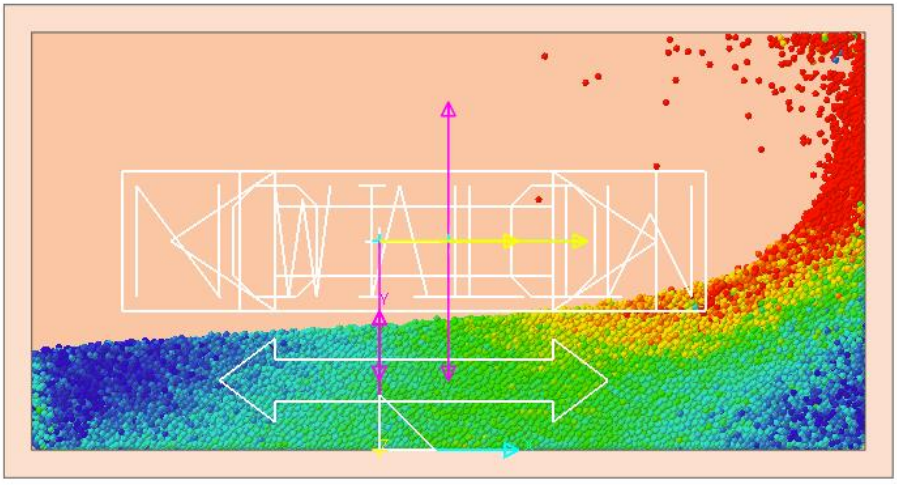
To display a particle contour:

Color can be used to display various properties of the fluid during the animation.

1. From the **Post-Process** group in the **External SPI** tab, check the **Particle Contour** icon
2. Change the **Contour Type** to **Translational Velocity Magnitude**.
3. In the **Min/Max Option**, set **Min/Max** values.
   - **Min: 0.1**
   - **Max: 1**
4. Uncheck the **Show Min Value** and the **Show Max Value**.
5. Check the **Enable Contour View** and click **OK**.



Now you have set contour to display the particles using colors to represent particle speeds relative to the reference coordinate system, as shown in the figure below.



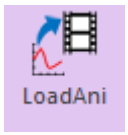
Working with Plots

You will see the cyclic fluid forces in RecurDyn acting on the tank body for caused by the sloshing of the fluid.

To view the driving torque:

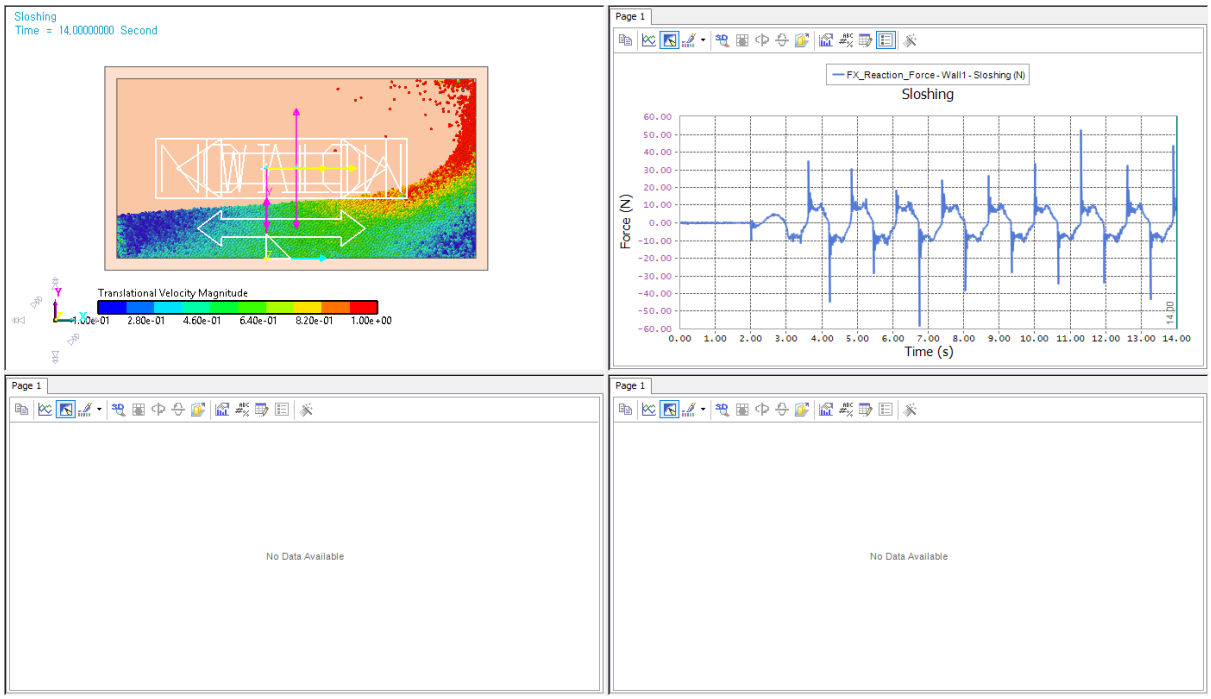
1. From the **Plot** group in the **Analysis** tab of the main RecurDyn ribbon, click **Result**.

2. From the **View** group in the **Home** tab of the ribbon, click **Show All Windows**.



3. From the **Animation** group in the **Tool** tab, click **Load Animation** in the upper left side of the window.
4. Plot the **FX_Reaction_Force** of the **Wall1** in the upper right side of the window.

   The result should look like the figure below. In the plot on the right, the reaction force acting in the x-direction on the body Wall1 can be seen. The reaction force has a repeating cyclic shape that matches the animation shown in the left window.



Chapter

6

Particleworks Postprocessing

In this chapter, you will use Particleworks to calculate fluid pressure and create an animation of fluid behavior by utilizing post-processing functionality in **Particleworks**.

Task Objective

Learn how to view the results of the fluid simulation and how to create a surface which looks like an actual fluid.

Estimated Time to Complete

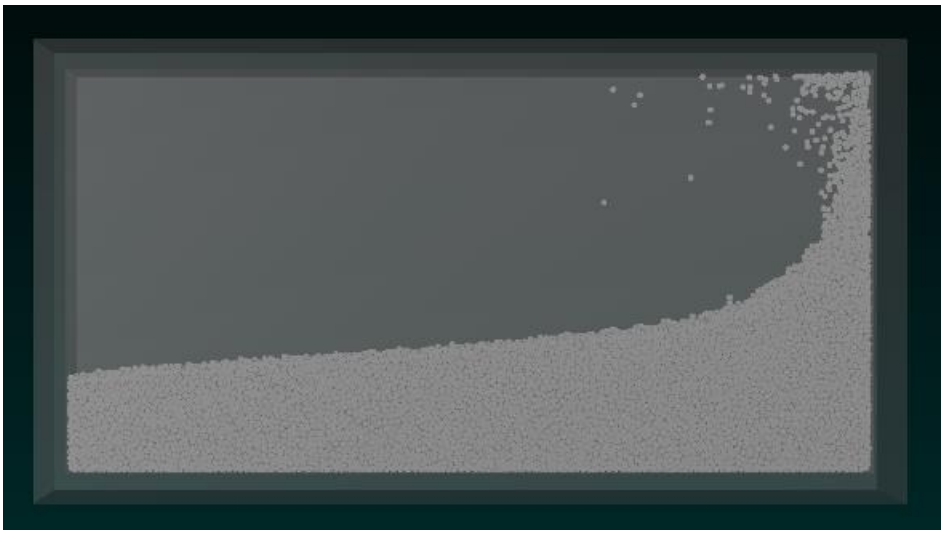
30 minutes

Working with Animations

You can view the animation of analysis result in Particleworks.

To play an animation:

- Click **Play** to view the animation.



To create the probe:

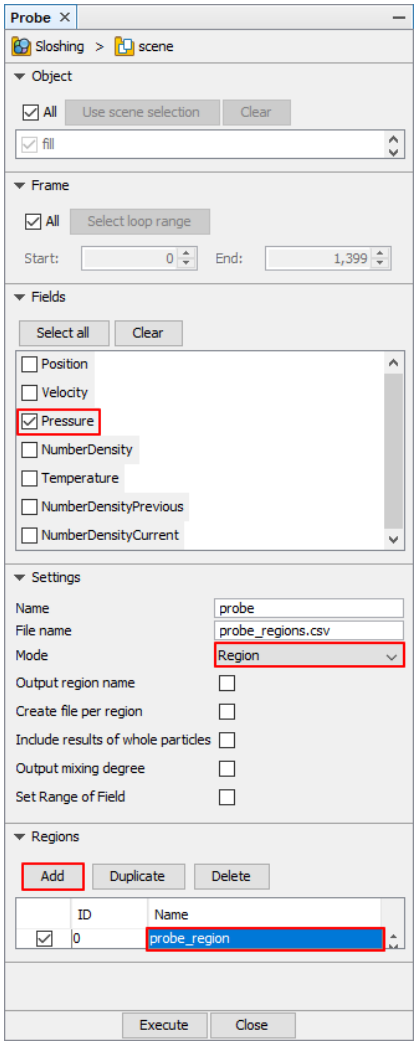
A probe is a particle whose result data can be plotted. You will create a probe and then plot the pressure experienced by that particle during the simulation.

1. Click **Probe** in the **Tools** tab.

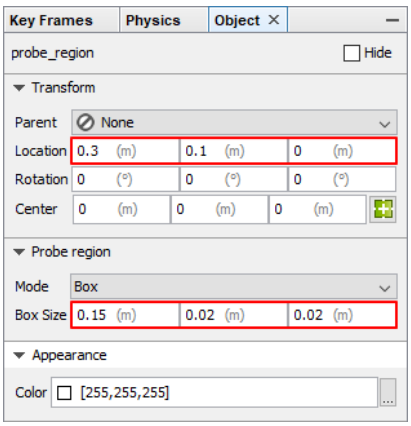
   The **Probe** dialog box appears on the right side.

2. Select **Pressure** under the **Fields**.
3. Set **Mode** to **Region** under the **Settings**.
4. Click **Add** under the **Regions**.
5. Click the list of **ID 0** the **Name** is **prove_region**

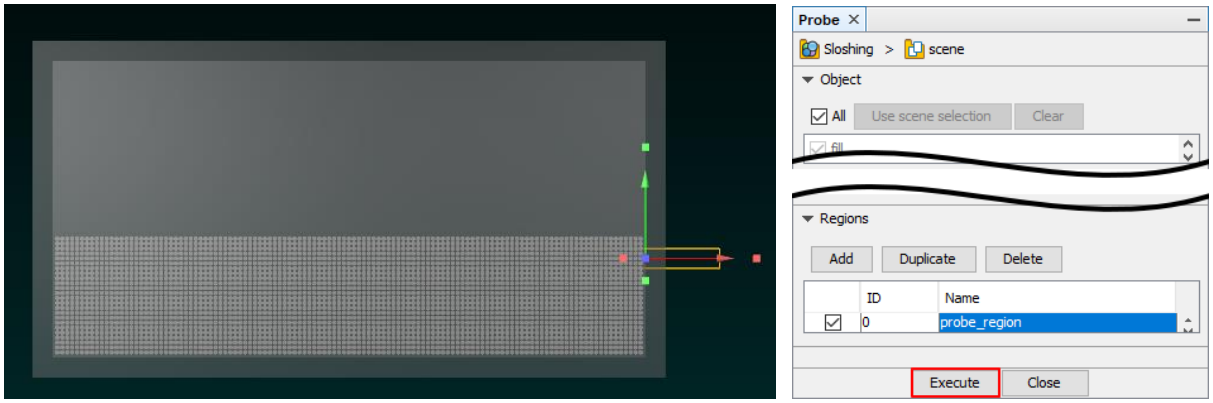
   The **Object** dialog box for **probe_region** appears on the left side.



6. Enter the following under the **Transform** in the **Object** dialog box.
   - **Location: 0.3, 0.1, 0**
7. Enter the following under the **Prove region** in the **Object** dialog box
   - **Box Size: 0.15, 0.02, 0.02**



Now you have created the probe within the bounds of motion to measure the wall on the right side of the Wall.



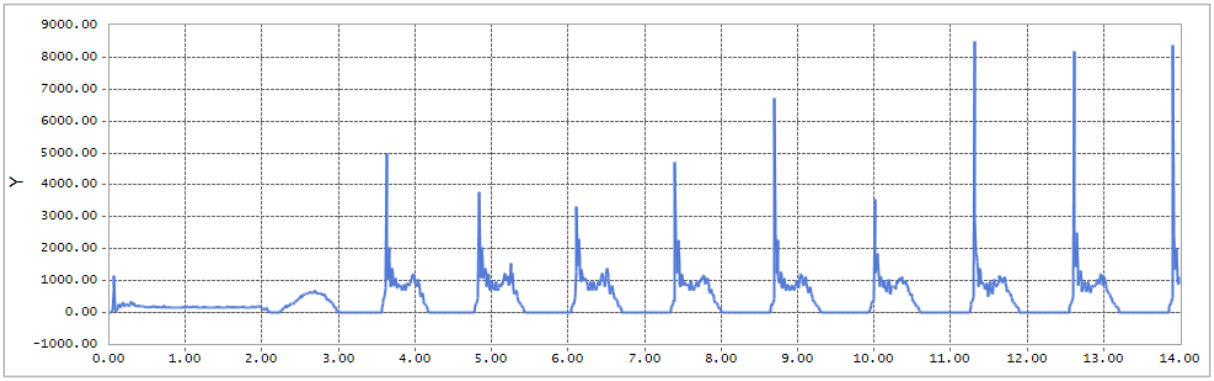
8. Click **Execute**.

   The probe result has been created as a CSV file type. The folder where the file is saved will appear following the completion of the CSV file creation.

   (Folder path: <ProjectLocation>/Sloshing/scene/prove)

The CSV output file can be opened by any appropriate program, such as Microsoft Excel, to view and plot its contents. The figure below was generated by Excel. It shows a plot of the pressure of the probe particle over time.

- **X : SimulationTime**
- **Y : Pressure_max**



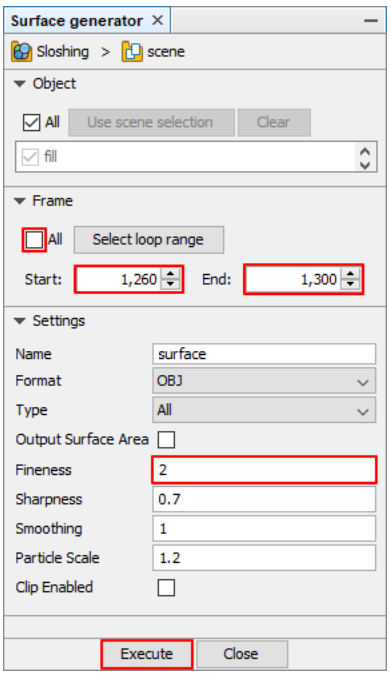
Creating the Surface

You can create a smooth surface of fluid behavior presented like as actual surface in Particleworks.

To create the surface:

1. Click **Surface generator** in the **Tools** tab.

   The **Surface generator** dialog box appears on the right side.



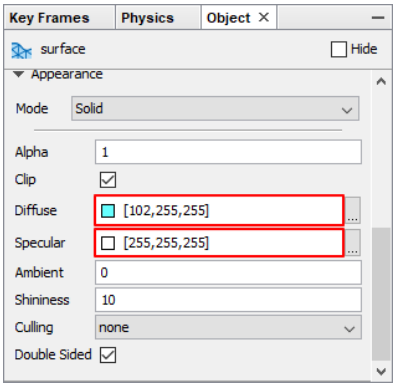
2. Enter and check the following in the **Surface generator** dialog box.
   - **Start Frame: 1260**
   - **End Frame: 1300**
   - **Fineness: 2**

3. Click **Execute**.

   You can check the current situation of progress in the **Output** dialog box.

4. Double-click **surface** in the **Project** dialog box after completion of surface creation. (scene>Post)

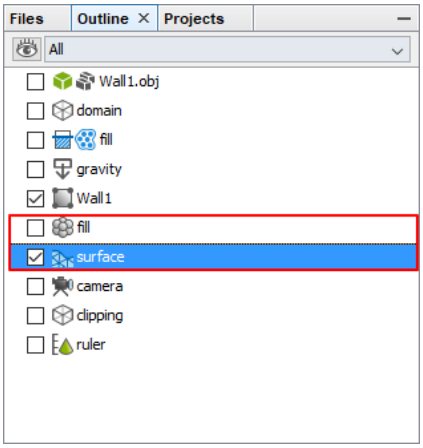
5. Change the values of **Diffuse** and **Specular** in the surface options as below.
   - **Diffuse: 102, 255, 255**
   - **Specular: 255, 255, 255**



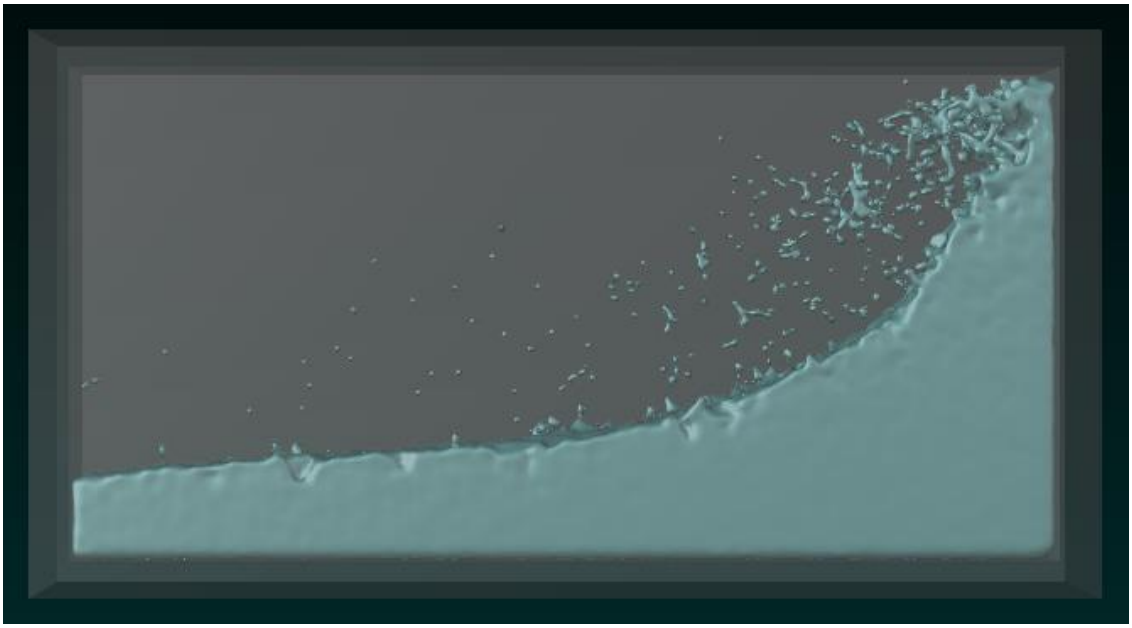
To hide the particle animation:

You should hide the existing particles of animation in Particleworks through the **Particleworks GUI**.

- Turn off the fill checkbox and select the surface checkboxes in the Outline dialog box.



You will see the result as shown in the figure below when playing the animation again.



In the period step1260 to step1300, this animation shows the result close to the flow of actual fluid by creating the smooth surface. Your presentation and report writing will be more convincing by using the images and animations created by this function.

Chapter 7

Result Analysis and Review

Task Objective

Learn how to do a comparative analysis between the trial result value from paper and the analysis result value from co-simulation between RecurDyn and Particleworks.

Estimated Time to Complete

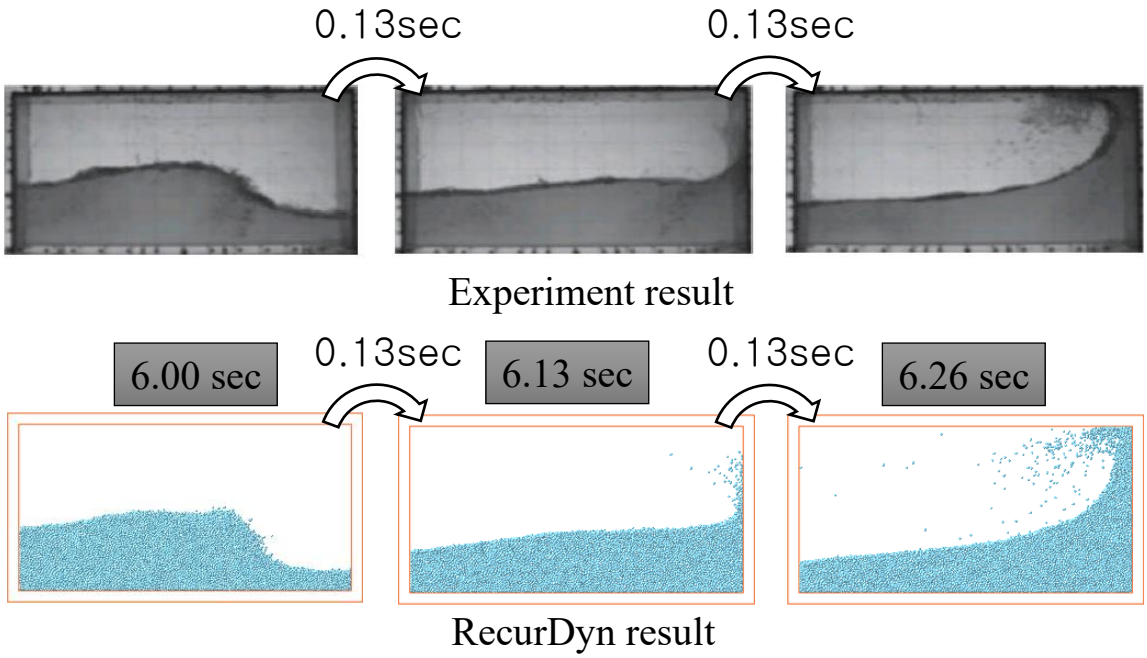
5 minutes

A Comparative Analysis of Results

You will judge the reliability of the MPS analysis method by doing a comparative analysis between the result from reference paper and the trial result according to this tutorial.

A comparative analysis of fluid behavior results:

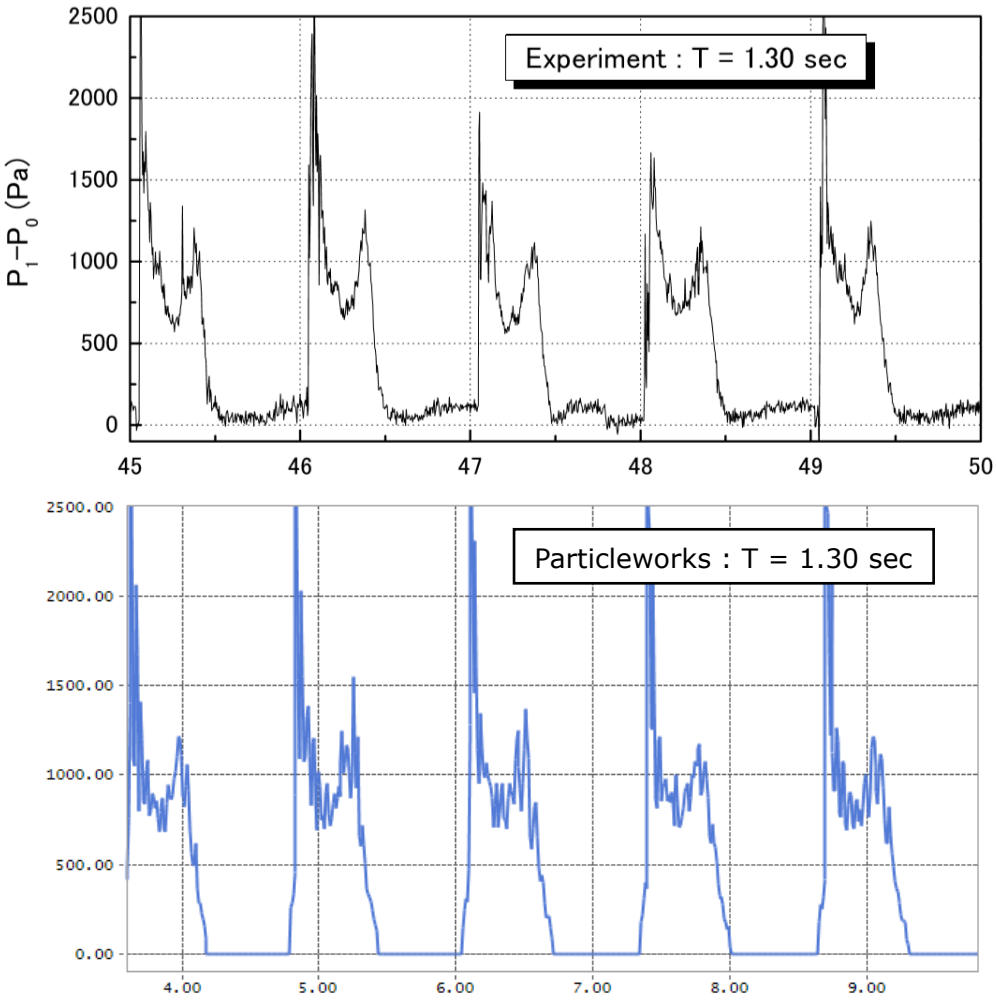
The figure below shows the profile of fluid behavior at intervals of 0.13 second. As you can see in the figures below, the fluid waveform created from the left side breaks hitting against the wall while moving to the right. The analysis result from trial shows that the waveform started moving to the right breaks hitting against the wall presenting the same waveform as the profile of trial.



A comparative analysis of fluid pressure result:

The figure below, as a pressure graph against time, shows both the data extracted from the trial and the data gained from the analysis from chapter 6 in this tutorial. Both pressure data values are measured under the same condition of 10cm above the ground.

Through a comparative analysis of two graphs, the similar pressure values in the similar pattern are created at the moment when the fluid hits against the wall with the periodic motion of 1.3 seconds.



Thanks for participating in this tutorial.