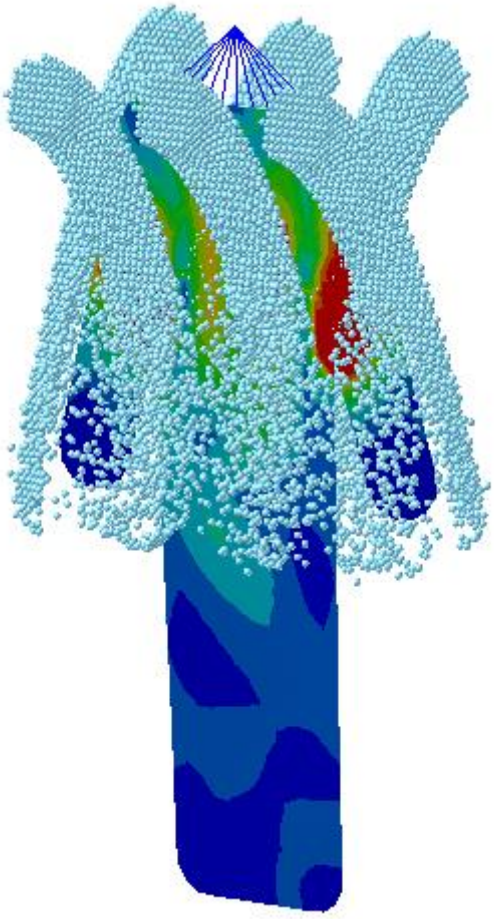
RECURDYN

Styler (Flex – Particleworks)



FunctionBay

Edition Note

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

Table of Contents

Overview 4
Task Objectives 4
Prerequisites 5
Tasks 5
Register Particleworks GUI in RecurDyn 6
Task Objective 6
Estimated Time to Complete 6
Copy Configuration XML 7
Modifying a RecurDyn Model 9
Task Objectives 9
Estimated Time to Complete This Task 9
Defining a patch set 10
Creating walls 11
Exporting a *.wall file 11
Creating a Particleworks Model 12
Task Objectives 12
Estimated Time to Complete This Task 12
Starting Particleworks 13
Configuring a preprocess 14
Preparing for Co-simulation 22
Co-simulation 23
Task Objectives 23
Estimated Time to Complete This Task 23
Co-simulation 24
Checking the Analysis Results 25
Task Objectives 25
Estimated Time to Complete This Task 25
Checking Contour 26

Chapter 1

Overview

This tutorial covers how to perform co-simulation between RecurDyn and Particleworks. In this tutorial, co-simulation is performed on the dynamic interaction between the flexible bodies of RecurDyn and the fluid particles of Particleworks.

The model that will be covered in this tutorial is the Styler mechanism. The Styler is used to shake off the dust attached to cloth and to remove its wrinkles with steam. This tutorial analyzes the dynamic interaction between the clothes, which is expressed as a flexible body, and the steam, which is expressed as particles. After the co-simulation, we will also check the stress formed as the particles touch the flexible body, using the Contour function of RecurDyn.

Task Objectives

This tutorial covers the following topics:

- How to export *.wall files of rigid and flexible bodies in a RecurDyn model
- How to create particles in Particleworks
- How to set fluid properties in Particleworks
- How to perform co-simulation in RecurDyn
- How to perform post-processing in RecurDyn

Prerequisites

- This tutorial is intended for users who have completed the basic tutorials provided with RecurDyn. If you have not completed these tutorials, then you should complete them before proceeding with this tutorial.
- Particleworks software must be installed to proceed with this tutorial. This tutorial was created based on Particleworks 6.2.0.
- In this tutorial, the analysis was carried out using the NVIDIA GeForce GTX TITAN graphics card. Depending on the specifications of the computer and the version of the software, there may be slight differences in the results of the analysis.

Tasks

This tutorial consists of the following tasks. The following table outlines the time required to complete each task.

(* The time required may vary depending on the specifications of the computer and the proficiency of the user.)

Task	**Duration (minutes)**
Registering Particleworks GUI on the RecurDyn Ribbon	10
Modifying a RecurDyn Model	10
Creating a Particleworks Model	15
Co-simulation	10
RecurDyn Post-processing	5
Total	**50**

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

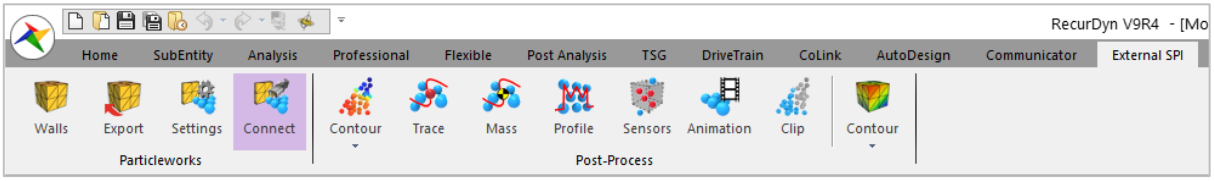
To paste XML in RecurDyn:

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

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

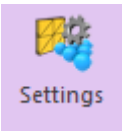
To confirm RecurDyn GUI:

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

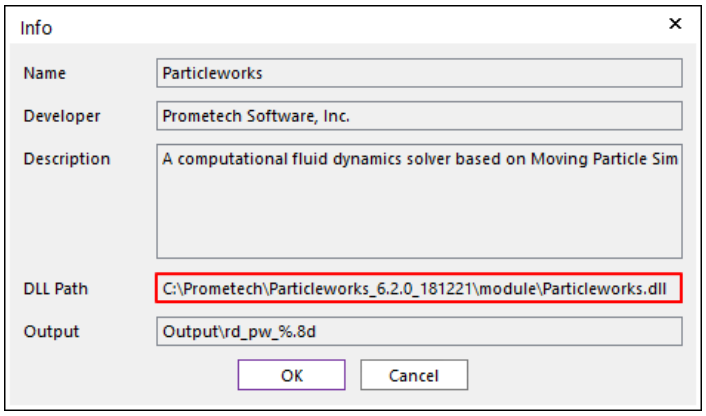


To check a path of particle solver DLL:

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**.

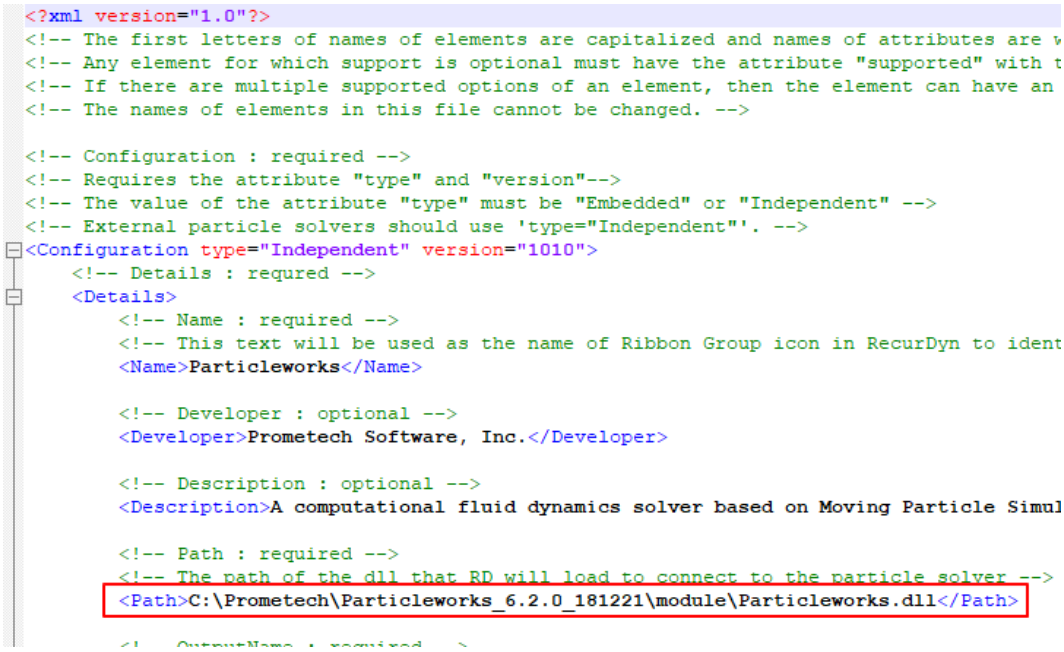


You can check the **DLL Path** in the **Info** dialog window. This path is set to the default installation location of Particleworks. So, if you installed Particleworks in a different path, you must modify the DLL path in the Configuration XML file.

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.



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

Chapter

3

Modifying a RecurDyn Model

Task Objectives

In this chapter, you will learn how to create a flexible wall for analysis and how to export the necessary files in Particleworks.

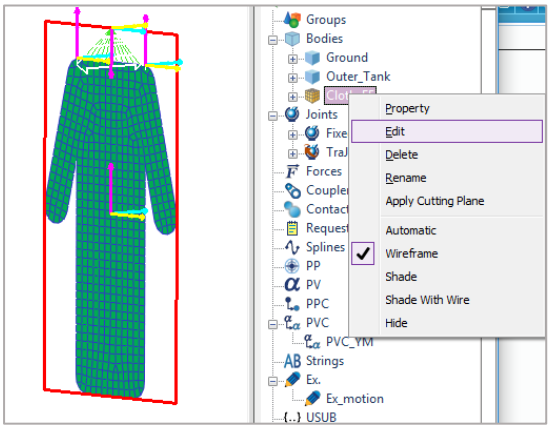
Estimated Time to Complete This Task

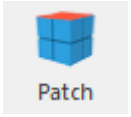
10 minutes

Defining a patch set

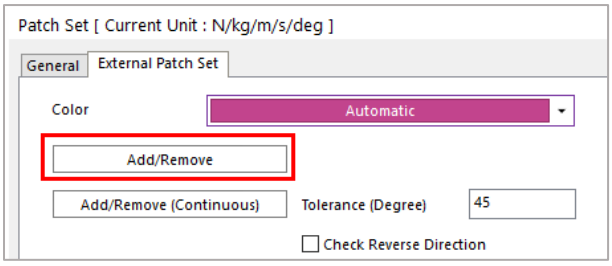
For co-simulation between flexible bodies and particles of Particleworks, the portion of the flexible body that contacts the particles must be defined as a patch set.

1. Open the **Styler_flex_model_cloth_start.rdyn** file that was provided as an example model for this tutorial.
2. Enter the Edit mode of the Cloth_FE body.

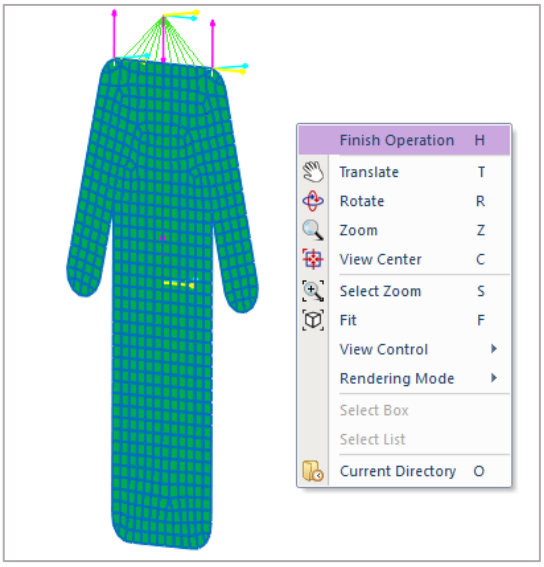


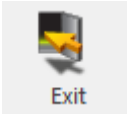


3. In the **Set** group of the **FFlex Edit** tab, click **Patch**.
4. When the **Patch Set** dialog box appears, click the **Add/Remove** button.



5. Select all **patches** by **dragging** them, right-click the mouse, and click **Finish Operation**.
6. Click the **OK** button.





7. **Exit** the **Edit** mode.

Creating walls

Create walls for Outer_Tank and Cloth_FE.

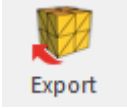
1. In the **Particleworks** group of the **External SPI** tab, click **Walls**.
2. In the **Working Window**, click **Outer_Tank**.
3. Click **Walls** again, and click **Cloth_FE.SetPatch1** in the **Working Window**.

Tip: You can select Cloth_FE by using **Select Box** or **Select List**, which are existing methods for selecting internal bodies. In this model, however, Outer_Tank is set as the layer 2 in a multi-layer. So, if you press "Ctrl + 2", which is a shortcut key combination for **Layer hide/show**, Outer_Tank will be hidden so that you can choose inner Cloth_FE more easily.

---

4. Modify the names of the walls as follows:
   - Wall1 → Wall_Outer_Tank
   - Wall2 → Wall_Cloth_FE

Exporting a *.wall file



1. In the **Particleworks** group of the **External SPI** tab, click **Export**.
2. When the **Export** dialog box appears, locate the folder you want to save the file to and click **OK** to save it.

   The **rd_pw.wall** file and the **WallGeometries** folder are created in the saved folder. The files created in this chapter are moved to the **Particleworks** project folder in Chapter 4.

TIP: If you want to save the rd_pw.wall file somewhere else, you may do so. Pressing the Export button and clicking **OK** immediately saves the file in the default folder where the rdyn files are saved.

Note: The rd_pw.wall file contains information about the position and orientation of **wall geometries**. If you import this file into **Particleworks** later, several wall geometries are imported at once. In the Wall Geometries folder, the files (*.obj or *.stl) related to wall geometry are automatically saved.

---

3. Save and exit **RecurDyn**.

Chapter 4

Creating a Particleworks Model

Task Objectives

In this chapter, you can import wall files from Particleworks and create fluid particles by using inflow as a model for steam.

Estimated Time to Complete This Task

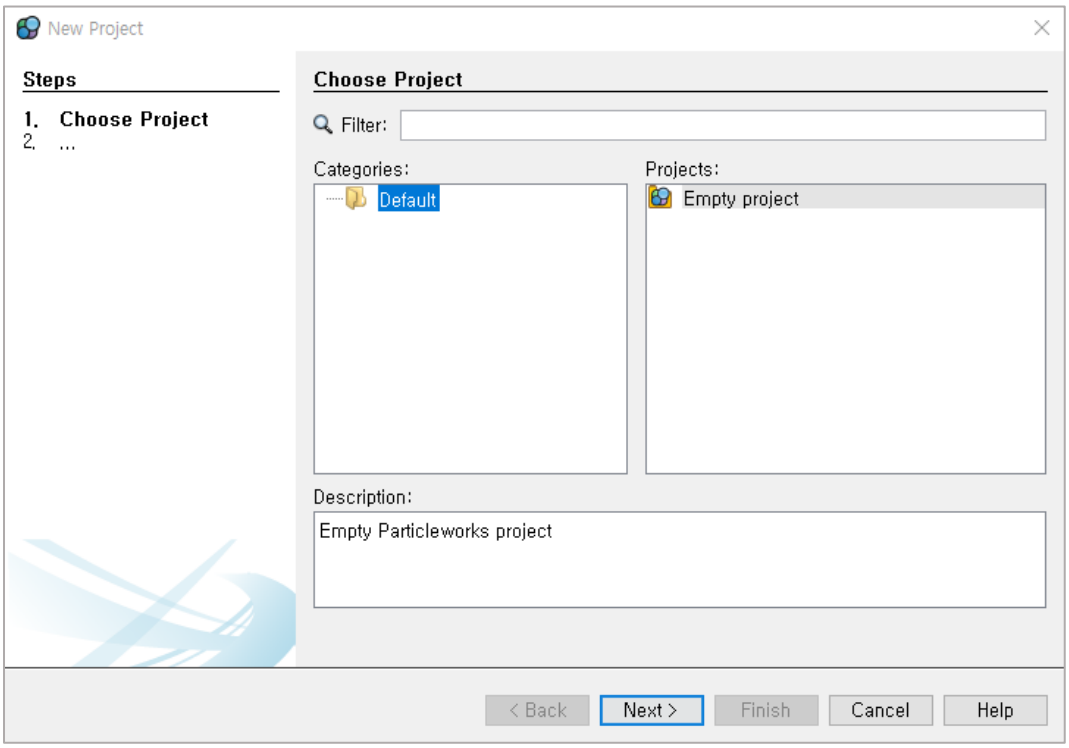
15 minutes

Starting Particleworks

Create a new model

1. Double-click the **Particleworks** icon on the Desktop.

2. When you click **New Project** on the **File** tab, the **New Project** dialog box appears.



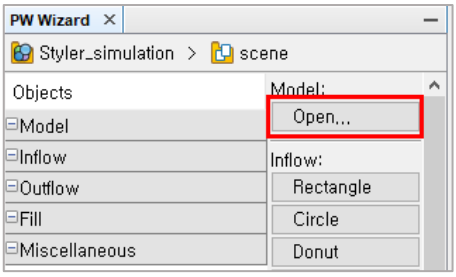
3. Click **Next**.
4. In the **Project Name** field, type **Styler_simulation**.
5. In the **Project Location** field, enter the location where the project will be created.
6. Click **Finish** to create the new model.

Copying a RecurDyn model and wall-related files

- In order for both software to perform co-simulation, the RecurDyn model and the Wall file must be in the Particleworks project folder.
- Move the Styler_flex_model_cloth_start.rdyn file, rd_pw.wall file, and WallGeometries folder you used in Chapter 3 to the scene folder in the folder where the Particleworks project was created. (Folder path: <Project Location>/Styler_simulation/scene)

Configuring a preprocess

Import a wall file

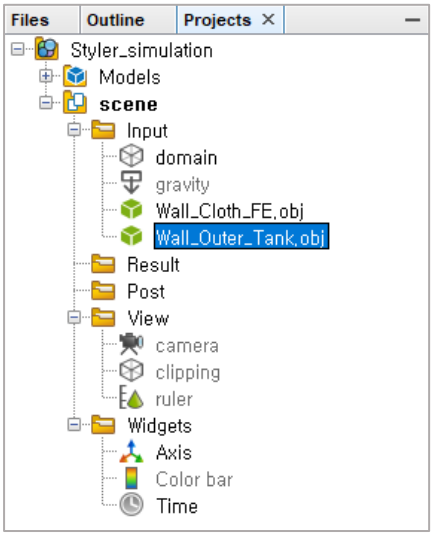


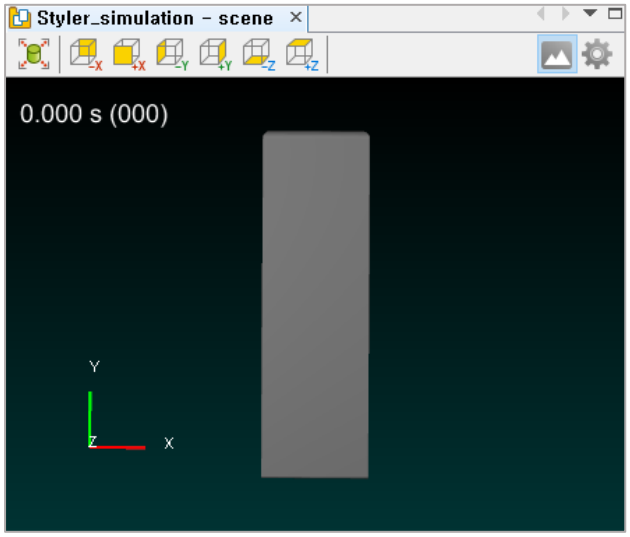
1. Double-click the **scene** folder in the **Projects** dialog box.
2. Click **Start wizard**.

   The **PW Wizard** dialog box appears on the right.
3. Click the **Open** button to import the rd_pw.wall file.

   <Project Location>/Styler_simulation/scene/rd_pw.wall
4. Click the **Fit** button and the **+Z** plane button at the top of the **Working Window** to view the imported wall.

   As shown on the left, you can check whether the **Wall_Cloth_FE.obj** and **Wall_Outer_Tank.obj** are properly **imported** into the **scene** tree and check the shape of the imported wall files as shown on the right.



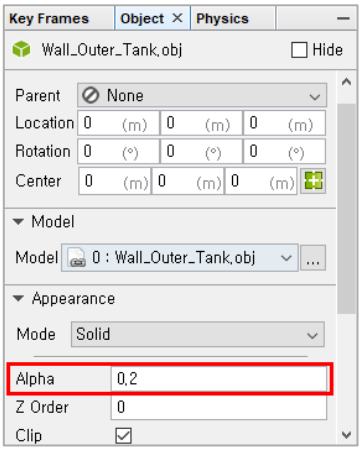


Set Wall_Outer_Tank.obj transparency

You need to adjust the transparency because the inner **cloth** is not visible due to **Outer_Tank**.

1. In the Projects dialog box, double-click **Wall_Outer_Tank.obj** under **scene > Input**.

   The **Object** dialog box associated with the **entity** you clicked is shown in the figure to the right.



2. Change the **Alpha** value in the **Appearance** panel to **0.2** and press **Enter**.

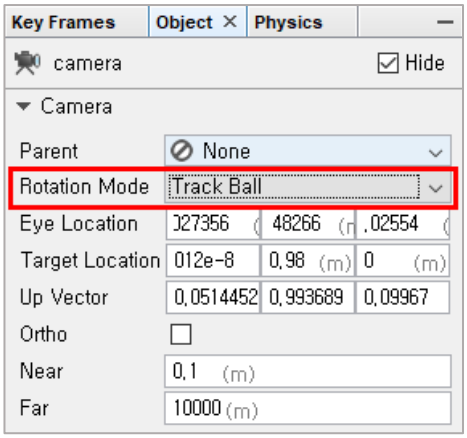
   Wall_Outer_Tank.obj becomes transparent, allowing you to look its inside.

Configure the camera

The default GUI setting of **Particleworks** is **Turn Table**, which is inconvenient to use. Switch the setting to a **Track ball** so that you can rotate it conveniently.

1. In the **Projects** dialog box, double-click **Camera** under **scene > View**.

   The **Object** dialog box associated with the **entity** you clicked is shown in the figure to the right.

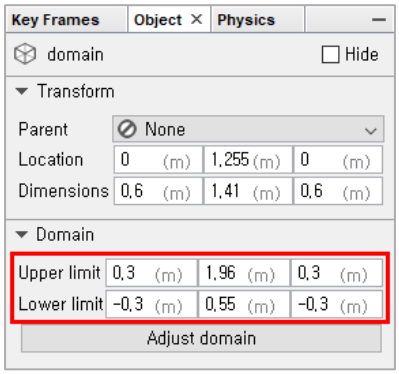


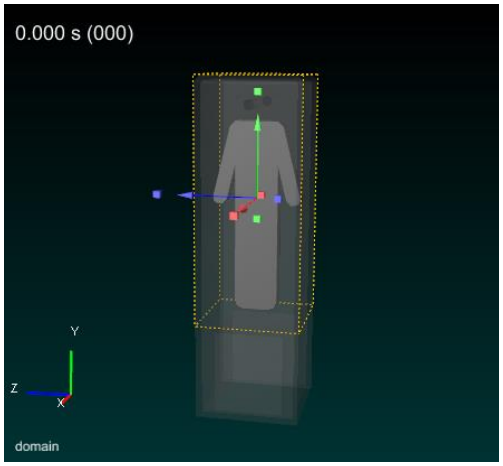
2. Change the Rotation Mode to Track Ball.

Set a domain

Domain is the area where fluid particles are analyzed. When the particles used as the steam fall down to the bottom, a reservoir at the bottom receives the water particles in the actual Styler. However, this analysis does not need to simulate the particles entering the reservoir. So, we will use a domain to delete the particles that fall down to the bottom.

1. In the **Projects** dialog box, double-click a **domain** under **scene > Input**.
2. Set the size of the **domain** as follows.
   - **Upper limit: 0.3, 1.96, 0.3**
   - **Lower limit: -0.3, 0.55, -0.3**



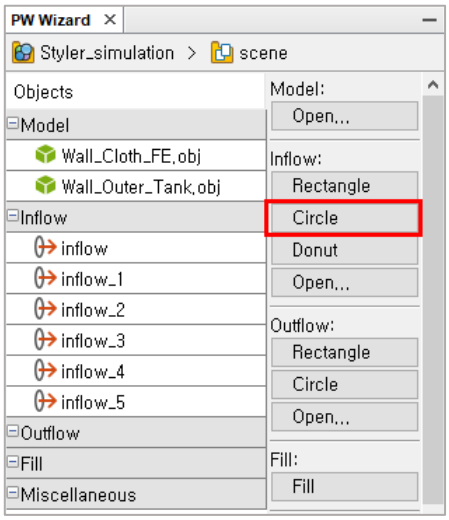


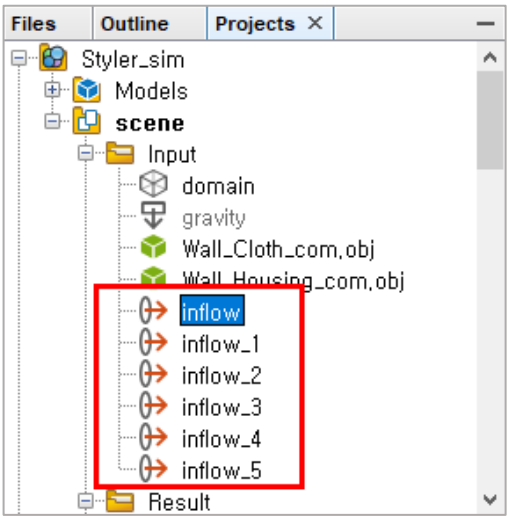
Create an inflow

Inflow is a function that lets particles flow to the user-defined zone at the user-defined speed or flux.

1. In the **PW Wizard** dialog box, click the **Circle** button six times.

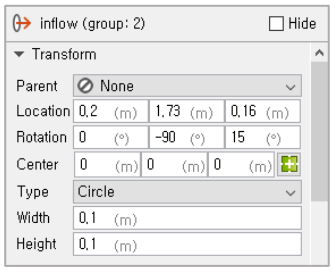
   In the **Projects** dialog box, six **inflows** are created in **scene > Input**.

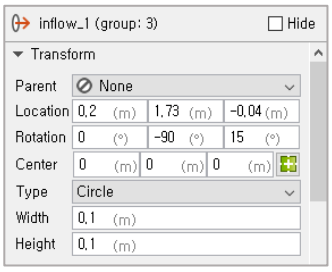


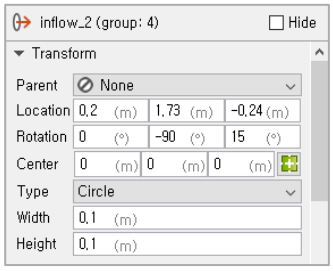


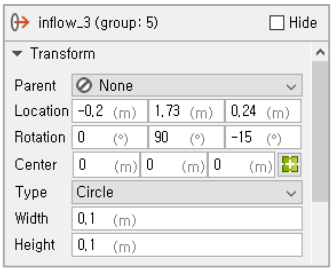
2. Double-click every **inflow** created in the Projects dialog box one by one to set the option values in the **Object** dialog box as shown below.

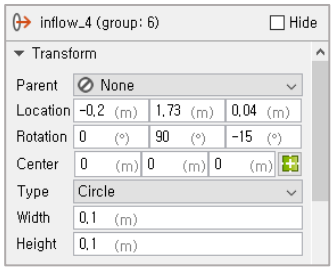
- **Width** and **Height** are set to the following values for all inflows.
  - **Width: 0.1**
  - **Height: 0.1**
- inflow
  - **Location: 0.2, 1.73, 0.16**
  - **Rotation: 0, -90, 15**
- Inflow_1
  - **Location: 0.2, 1.73, -0.04**
  - **Rotation: 0, -90, 15**
- Inflow_2
  - **Location: 0.2, 1.73, -0.24**
  - **Rotation: 0, -90, 15**
- Inflow_3
  - **Location: -0.2, 1.73, 0.24**
  - **Rotation: 0, 90, -15**
- Inflow_4
  - **Location: -0.2, 1.73, 0.04**
  - **Rotation: 0, 90, -15**
- Inflow_5
  - **Location: -0.2, 1.73, -0.16**
  - **Rotation: 0, 90, -15**

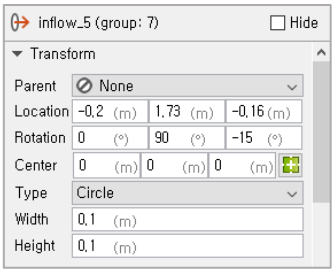




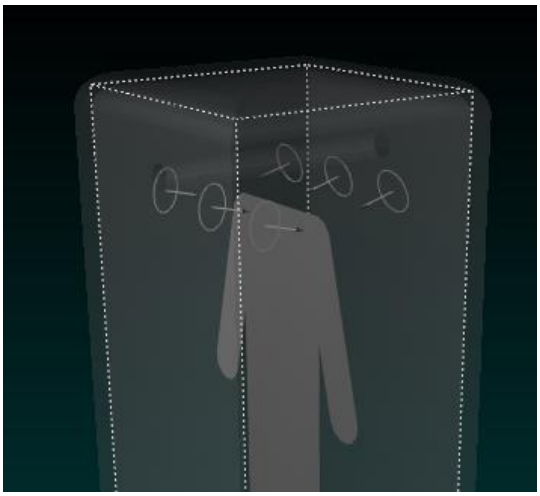






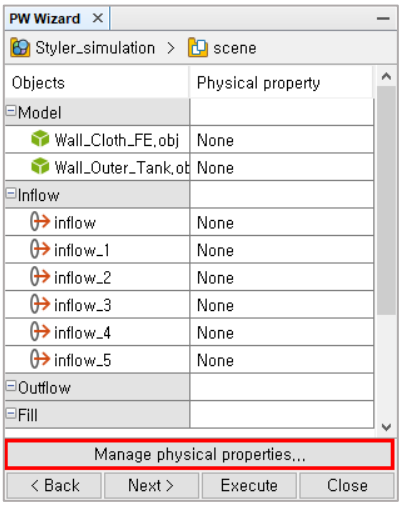


3. After confirming that the **inflows** are well defined as shown in the figure on the right, click the **Next** button in the **PW Wizard** dialog box to proceed to the next page.



Create and configure physical properties

1. At the bottom of the **PW Wizard** dialog box, click **Manage physical properties...**.

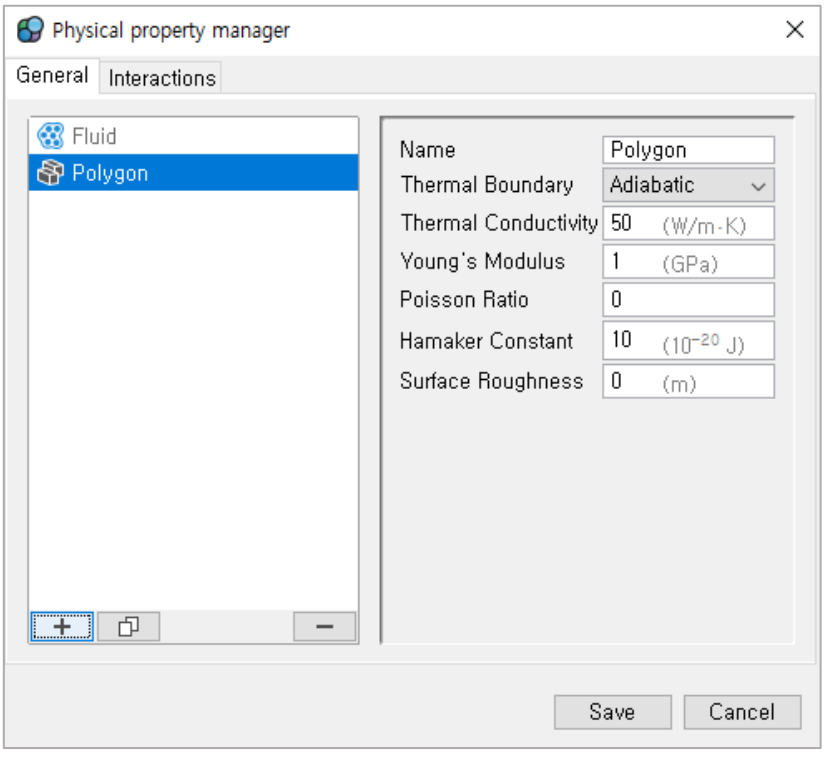


2. When the **Physical property manager** dialog box appears, click the **+** button to create a **fluid**.

   The basic physical property of the generated fluid is water. So you can use it as is.

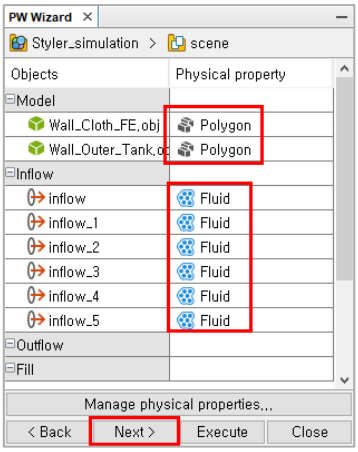
3. Click the **+** button again to create a **polygon**.

   Polygon is a physical property that specifies the rigid or flexible body defined as a wall by RecurDyn.

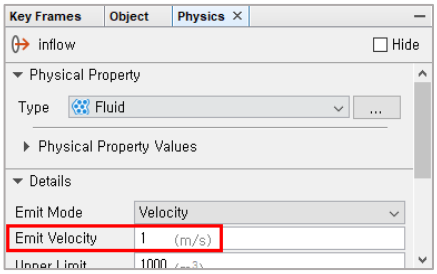


4. Click the **Save** button to close the dialog box.

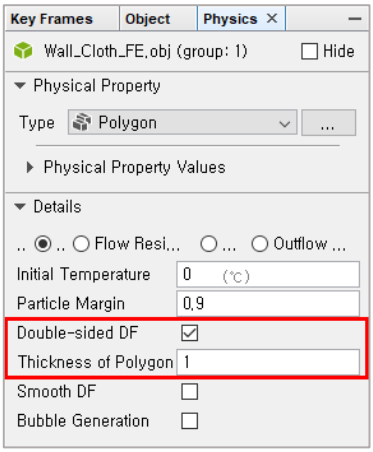
5. In the **PW Wizard** dialog box, change the **physical properties** from **None** to the values specified below.
   - **Wall_Cloth_FE.obj: Polygon**
   - **Wall_Outer_Tank.obj: Polygon**
   - **All inflows: Fluid**



6. In the **Physics** tab of every **inflow**, set the value for **Emit Velocity** as shown below.
   - **Emit Velocity: 1**



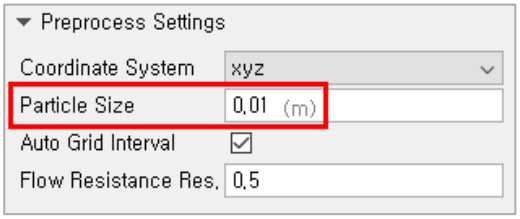
7. In the **Physics** tab of **Wall_Cloth_FE.obj**, check **Double-sided DF** and set the Thickness of Polygon as follows.
   - **Thickness of Polygon:1**



8. Click the **Next** button at the bottom of the **PW Wizard** dialog box to proceed to the next page.

Configure particles and preferences

In the PW Wizard dialog box, change the preferences.

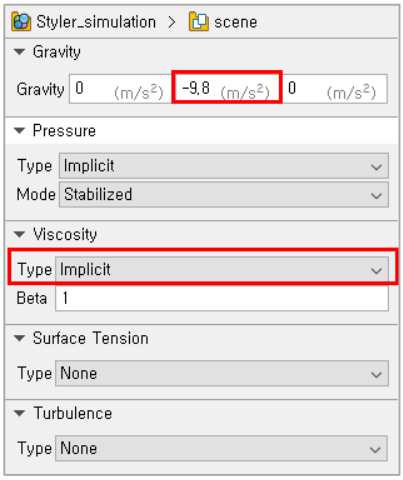


1. In the **Particle Size** field, enter **0.01**.
2. Click **Next** to proceed to the next page.

Note: In the case of **SPI**, the **unit** in **Particleworks** must be set to **meter** regardless of the **unit** in **RecurDyn**. If the unit in **RecurDyn** is set to **meter**, you don't need to convert the **particle size** or **domain size** and you just need to use the same values in Particleworks. If the unit in **RecurDyn** is set to **millimeter**, you need to convert the millimeter values to the meter values and enter the meter values in **Particleworks**.

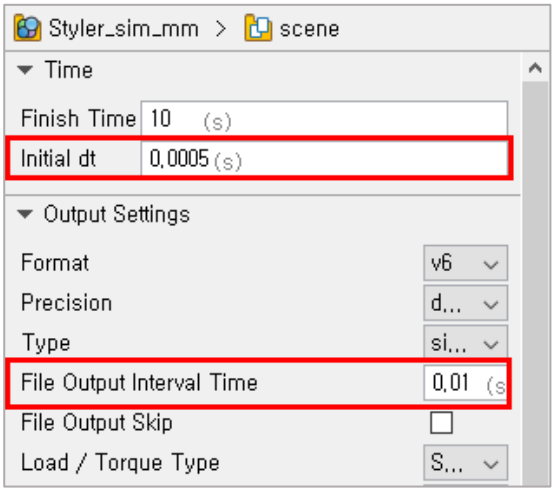
---

3. In the **Y** field of the **Gravity** pane, enter **-9.8**, just like in **RecurDyn**.
4. Set the **Viscosity** option as shown below.
   - Type: Implicit
5. Leave the others at the **default** settings and click **Next**.
6. On the **Thermal** page, press **Next** without changing any of the default values.



Configure analysis conditions

Set the analysis options in the PW Wizard dialog box.



1. In the **Time** pane, enter **0.0005** in the **Initial dt[s]** field.
2. Enter 0.01 in the File Output Interval Time[s] field.

Create particles

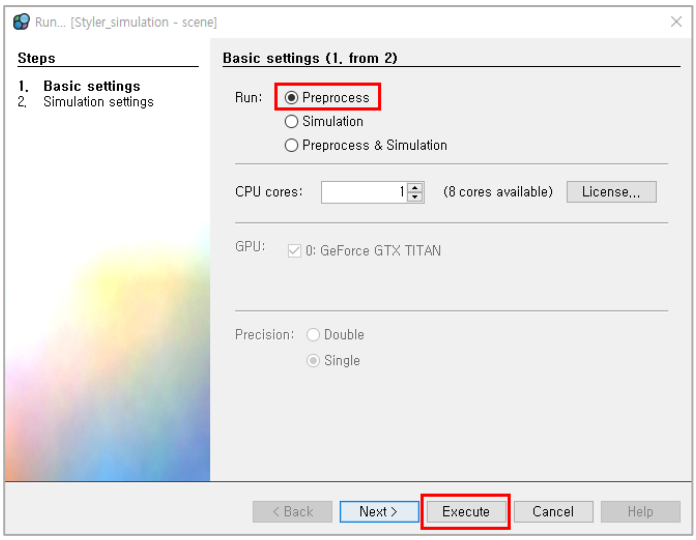
1. In the **PW Wizard** dialog box, click the **Execute** button when you have finished all the settings.

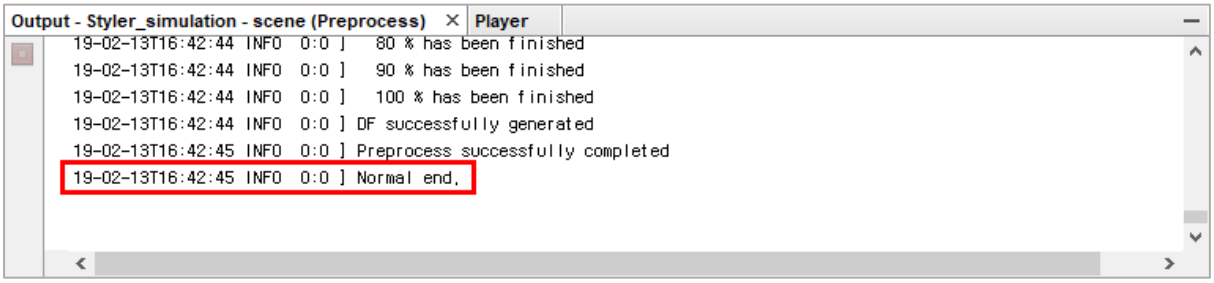
   The **Run** dialog box appears.

2. Set the **Run** option to **Preprocess**.

3. Click **Execute**.

   If the message saying that the particle creation is completed appears in the **Output** dialog box as shown below, the particles have been created.





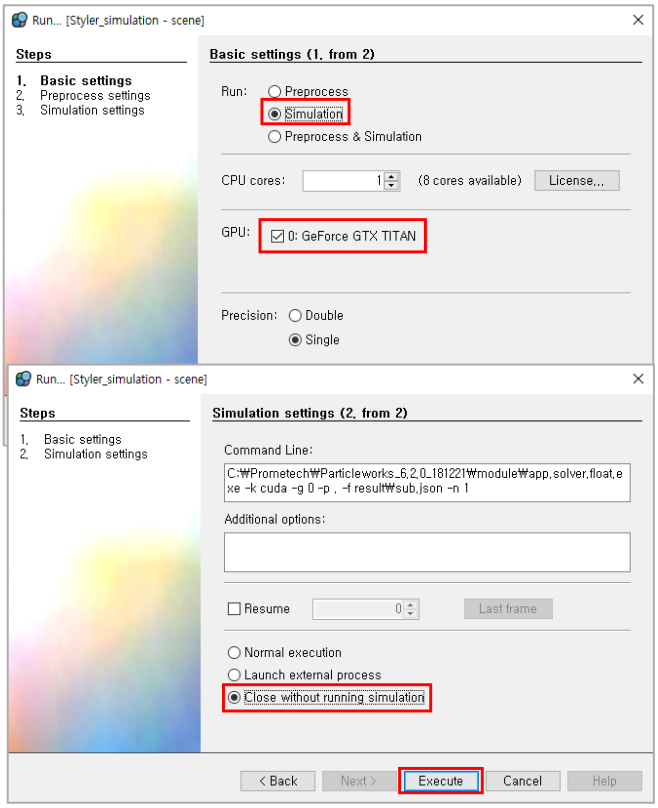
Preparing for Co-simulation

To do co-simulation in RecurDyn, you need to create some files in Particleworks. The relevant files are automatically generated by Particleworks as you go through the standalone analysis.

Perform standalone analysis in Particleworks.

1. On the **Simulation** tab, click **Run**.
2. Set the **Run** option to **Simulation**.
3. Enter the number of **CPU cores** to be used for the analysis.

   If there is a GPU, select the GPU to be used for the analysis.
4. Click **Next**.
5. Check **Close** without running simulation.
6. Click **Execute**.



7. Save the project.

The **Particleworks** program may be turned off in Chapter 5. However, in this tutorial, we will proceed to the next chapter with the program turned on to check the analysis results in real time during analysis.

Chapter

5

Co-simulation

Task Objectives

In this chapter, we will perform co-simulation using RecurDyn and Particleworks to analyze the interaction between flexible bodies and particles.

Estimated Time to Complete This Task

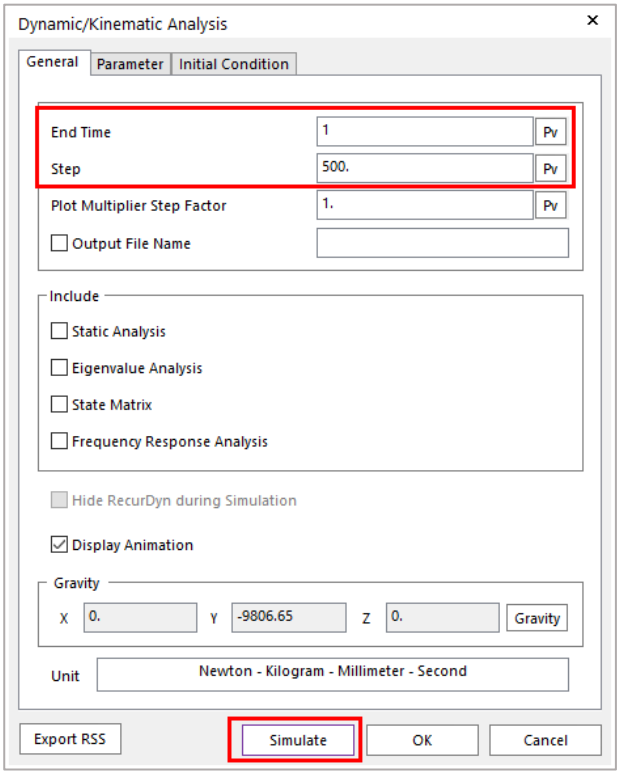
10 minutes

Co-simulation

Perform co-simulation in RecurDyn

1. Run **RecurDyn** and open the **Styler_flex_model_cloth_start.rdyn** file you copied in Chapter 4.
   (File path: <Project Location>/ Styler_simulation/scene/Styler_flex_model_cloth_start.rdyn)

2. In the **Simulation Type** group of the **Analysis** tab, click **Dyn/Kin** to open the **Dynamic/Kinematic Analysis** dialog box.
3. Check the simulation end time and step.
   - **End Time**: 1
   - **Step**: 500



4. Click **Simulate** to perform the analysis.

Tip: Some of the **options** in the **Dynamic/Kinematic Analysis** dialog box are as follows.

- **End Time**: Defines the simulation duration.
- **Step**: Defines the number of frames that are saved during the entire simulation duration.
- **Plot Multiplier Step Factor**: Defines the number of saved data points for plotting. The number of plot data points is defined by multiplying Step by Plot Multiplier Step Factor.

Chapter

6

Checking the Analysis Results

Task Objectives

In this chapter, we will check how particles affect a flexible body using Contour.

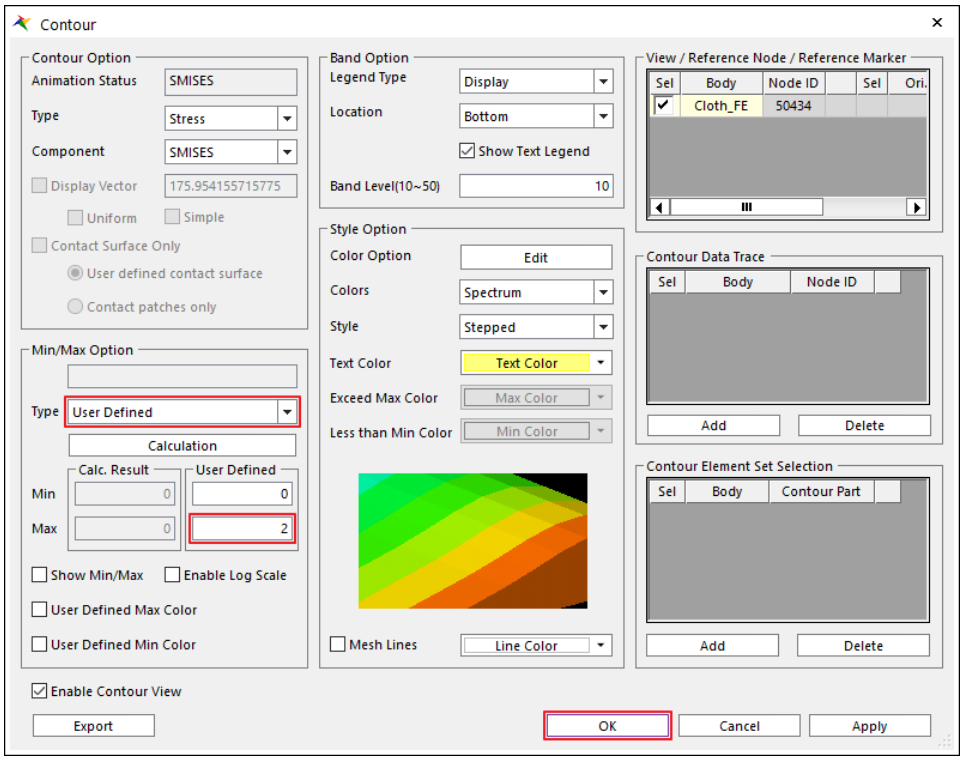
Estimated Time to Complete This Task

5 minutes

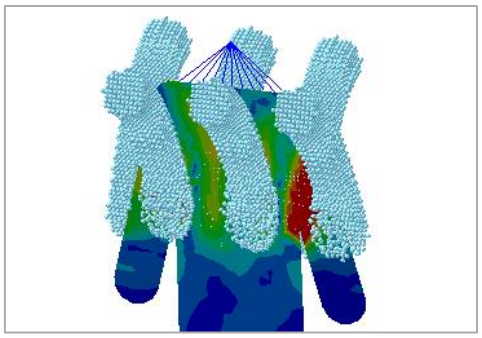
Checking Contour

After the analysis is completed, you can check the result using Contour to see whether the particles affect the flexible body. And since we do not actually model the exact physical properties and interactions of the fabric, the values from the stress or reaction force are not exact experimental values. So please check only whether the flexible body is affected by the particles.

1. In the **FFlex** group of the **Flex** tab, click **Contour**.
2. In the **Contour** dialog box, set the **Type** to **User Defined** and click the **Calculation** button.



3. Change the **Max** value of User Defined to **2** and click **OK**.
4. In the **Animation Control** group of the **Analysis** tab, click the Play button to see **Contour**.



Thanks for participating in this tutorial