

Ansys Rocky Tutorial

Bulk Material Characteristic Testing II: Shear Box

Developed and curated by the Ansys Academic Development Team

Angus Foley, University of Newcastle

education@ansys.com

Summary

This tutorial will cover the setup and execution of a simulated shear box test used in bulk material calibration. This test is performed to determine the angle of repose and shear angle of a bulk material.

Prerequisites

It is recommended to complete Bulk Material Characteristic Testing I: Lifting Cylinder tutorial before this exercise. This can be found on the [Ansys Education Resources site](#).

Table of Contents

1. Introduction.....	3
2. Exercise: Simulating Shear Box Test.....	4
2.1 Physics	4
2.2 Geometry	4
2.3 Gate Movement	6
2.4 Particle Material Definition	6
2.5 Particle Creation	7
2.6 Material Interaction Definition.....	7
2.7 Inlet Definition.....	7
2.8 Solve Simulation	8
2.9 Analysis: Estimation of Shear Angle	9

1. Introduction

To understand the importance of calibrating bulk material for handling and processing refer to the [Angle of Repose Test Ansys Innovation Course](#) here.

The test we will focus on for this tutorial is the *Shear Box test*, which aims to determine the internal strength of a bulk material when no consolidation loads are applied via the shear angle (internal friction angle) of the material. The shear angle refers to the angle formed between the horizontal plane and the direction of the maximum shear stress in the material. It is a characteristic property that describes the material's resistance to deformation under shear loading conditions.

In granular materials or bulk solids, the shear angle is an important parameter that influences their behavior during handling, processing, and storage. It indicates the internal friction and cohesion within the material. When shear forces are applied to a bulk material, particles within the material tend to rearrange themselves to accommodate the stress. The shear angle represents the maximum angle at which the material remains stable and does not undergo further deformation or collapse.

The shear angle, similarly, to the angle of repose can be used for a number of applications when designing systems containing bulk materials. These include designing mass flow geometry, which is the optimal flow regime for silos, hopper, etc., stability analysis, which allows engineers to assess the stability of slopes and stockpiles and container wall loads, which is useful for analyzing the structural integrity of walls. As expected, the shear angle increases with an increase moisture content and other factors similar to the AoR.

This test is conducted in a box of known dimensions and is fitted with one wall that can be removed quickly in both the simulation and real testing, shown in figure 1 as the right bold wall. The shear stress from the material in the box forms the shear angle denoted below as θ_{SB} . For the purpose of this simulation, we will include a second lower-lying box to catch the materials falling. In real testing, this fallen material can be used as another calibration parameter, known as residual mass, which refers to the mass (or percentage) of material that has fallen, or even used to determine the angle of repose.

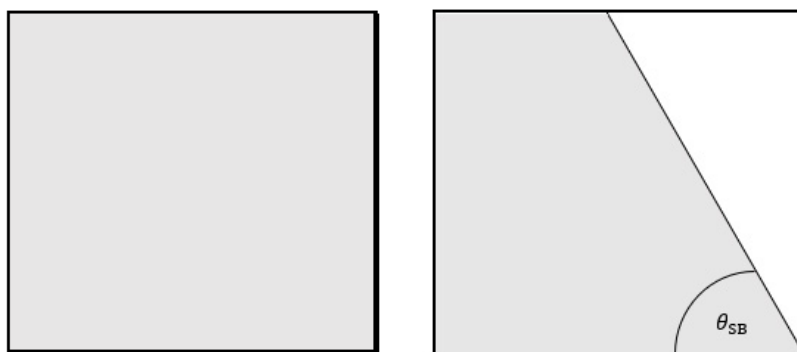


Figure 1: Shear box test before and after wall is removed

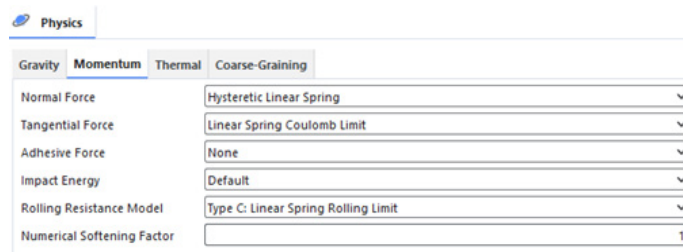
2. Exercise: Simulating Shear Box Test

Download the three .stl files uploaded with this lab document, *shearbox*, *shearbox_gate* and *shearbox_tray*.

Open Ansys Rocky¹ and create a new project using the New Project icon and save to an appropriate location.

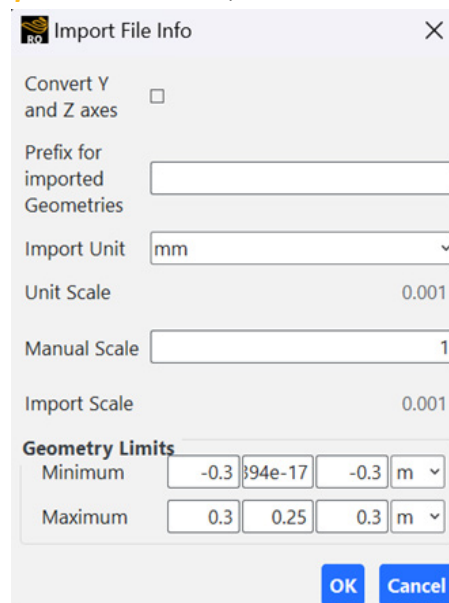
2.1 Physics

Now, it is important to set the correct physics definitions before any simulation. This can be done through the **Physics** tab. Set the **Rolling Resistance Model** under the **Momentum** tab to the following, while leaving all others as default.



2.2 Geometry

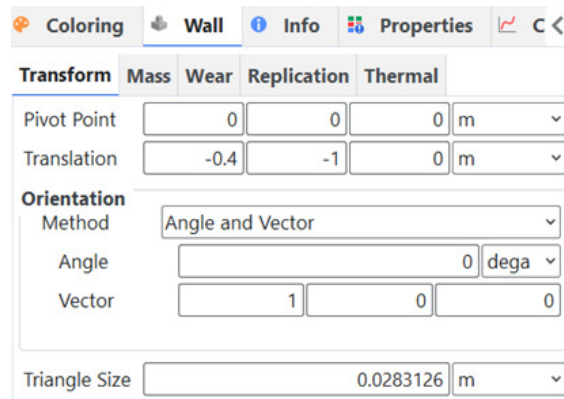
Import the two downloaded geometries: *shearbox* and *shearbox_gate*, with an **Import Unit** of *mm*. Now, the final geometry, *shearbox_tray* needs to be imported. Once you have been prompted by the **Import File info tab**, select the **Import Unit** as *mm*, as shown below.



Open a new 3D workspace to view the imported geometry by selecting the **New Workspace with 3D View**. Alternatively, you can select and drag the **Geometry** entity into the workspace.

¹ This resource was made with Ansys Rocky 23R2. Interface may look different, depending on which release you are using.

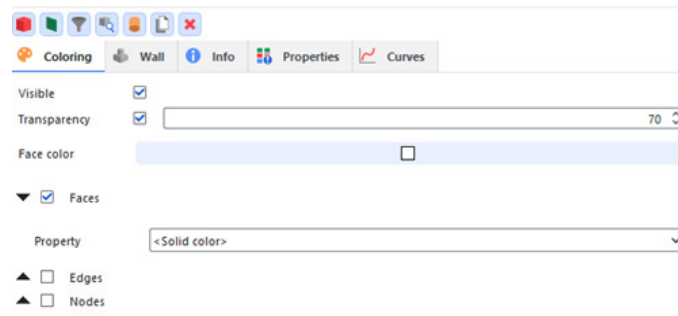
You will notice the newly imported geometry is not in the correct position, so we need to move it. This can be done either using CAD tools such as Ansys Discovery, or in Ansys Rocky, for this tutorial we will use the latter. To do this navigate to the *shearbox_tray* entity under the **Geometry** tab and select **Transform**. Set the **Pivot Point** and **Translation** to the following.



Transform	Mass	Wear	Replication	Thermal
Pivot Point	0	0	0	m
Translation	-0.4	-1	0	m
Orientation				
Method	Angle and Vector			
Angle	0 dega			
Vector	1	0	0	
Triangle Size	0.0283126 m			

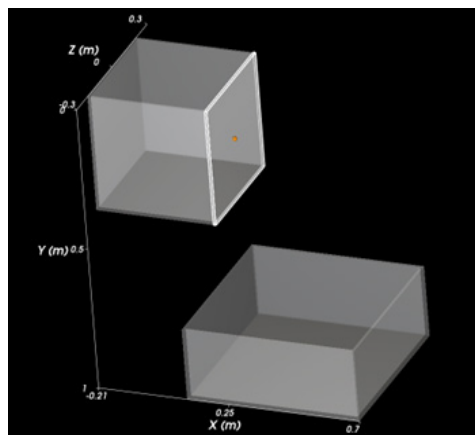
You will notice if you hide the *shearbox_gate*, the *shearbox_tray* is positioned so that it is directly underneath the gate, where the material will fall once the gate is removed.

Now, we need to make the geometries transparent. To do this select one of the geometries under the **Geometry** tab, and choose the **Coloring** option and enable **Transparency**. This can be adjusted to be more or less transparent, but we will leave it as 70 for this tutorial.



Coloring	Wall	Info	Properties	Curves
Visible	<input checked="" type="checkbox"/>			
Transparency	<input checked="" type="checkbox"/>			
Face color	[Light Blue]			
Faces	<input checked="" type="checkbox"/>			
Property	<Solid color>			
Edges	<input type="checkbox"/>			
Nodes	<input type="checkbox"/>			

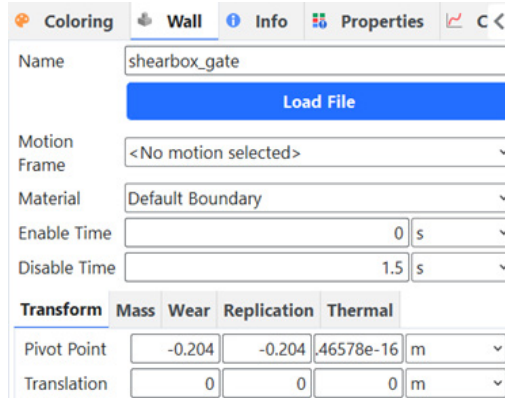
If done correctly the final geometry should appear like the following with the *shearbox_gate* selected, ensuring it is on the correct side of the box.



2.3 Gate Movement

To simulate the *Gate* opening to allow the particles to fall through it is most effective to disable the geometry at a specific time.

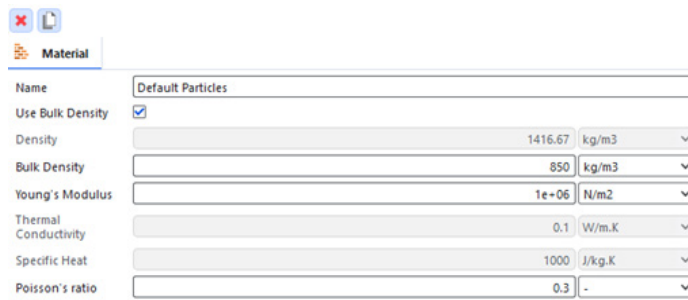
To do this navigate to the **Geometry** tab and select the *shearbox_gate* and then the **Wall** option. Now change the **Disable Time** to the following and leave all the rest as default. This will allow the *shearbox* to fill and hold particles for 1.5 seconds then the *shearbox_gate* geometry will be removed and allow the particles to fall through.



The screenshot shows the 'Wall' tab in the ANSYS software interface. The 'Name' field is set to 'shearbox_gate'. Below it is a 'Load File' button. The 'Motion' dropdown is set to '<No motion selected>'. The 'Material' dropdown is set to 'Default Boundary'. The 'Enable Time' is set to 0 s, and the 'Disable Time' is set to 1.5 s. Below these fields are tabs for 'Transform', 'Mass', 'Wear', 'Replication', and 'Thermal'. The 'Transform' tab is active, showing 'Pivot Point' as (-0.204, -0.204, 46578e-16) m and 'Translation' as (0, 0, 0) m.

2.4 Particle Material Definition

To define parameters for the particles just created, select **Default Particles** under the **Materials** tab. You will need to define the **Bulk Density** and **Young's Modulus** as follows.



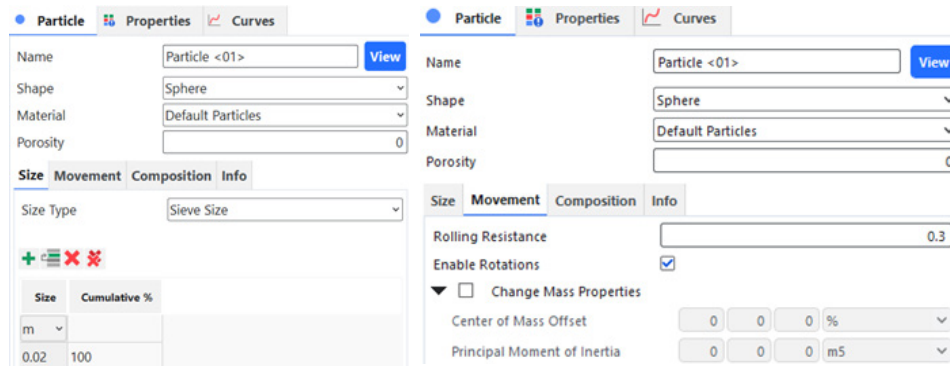
The screenshot shows the 'Material' tab in the ANSYS software interface. The 'Name' field is set to 'Default Particles'. The 'Use Bulk Density' checkbox is checked. The 'Density' is set to 1416.67 kg/m3. The 'Bulk Density' is set to 850 kg/m3. The 'Young's Modulus' is set to 1e+06 N/m2. The 'Thermal Conductivity' is set to 0.1 W/m.K. The 'Specific Heat' is set to 1000 J/kg.K. The 'Poisson's ratio' is set to 0.3.

The **Young's Modulus** is set to a smaller value to decrease solve time by reducing the loading and unloading stiffness of the particles².

² It should be noted, values for the Young's Modulus would typically be higher, however due to limitations while creating this tutorial, this value was chosen.

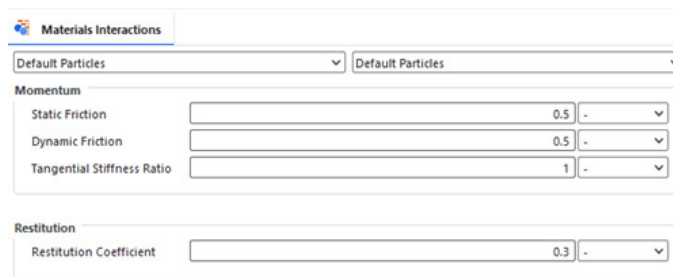
2.5 Particle Creation

Create a new particle, by right clicking on the **Particle** tab and select **Create Particle**. This particle will appear under the Particle tab as **Particle <01>**. Define **Size** and **Rolling Resistance** of the particle as the following and leave all else default.



2.6 Material Interaction Definition

For this simulation we will only be changing the **Default Particle** to **Default Particle interactions** as we are only interested in the interparticle interactions. To do this select the **Material Interaction** tab and select **Default Particles** for **both** drop down boxes and define the **Static Friction** and **Dynamic Friction** as follows, while leaving all other parameters as default.



2.7 Inlet Definition

To fill the shear box, we will use a **Volumetric Inlet**, as this provides a much faster fill than a standard particle inlet, like that of the one in the *Static Angle of Repose Test*. The Volumetric Inlet method allows you to inject a region of closely packed particles into a simulation all at one time, whereas a **Particle Inlet** releases particles in a continuous stream. The former is more applicable for simulations that require all particles in the simulation before analysis, the latter is better for on-going simulations that require a constant feed of particle, for example transfer chutes and conveyor belts.

To do this, navigate to the **Inlets and Outlets** tab and right click and then select **Create Volumetric Inlet**. This inlet will appear under the **Inlets and Outlets** tab, as **Volumetric Inlet <01>**, select this. In here we will add a particle by selecting the green plus button (**Add**) and select **Particle <01>** (the previously created particle) under the **Particle** tab and set **Mass** to 54 (kg). We also need to define the inlets **Sed Coordinates**, **Center Coordinates** and **Dimensions** under the **Region** tab to the following, leaving all other values as default.

Volumetric Inlet **Info**

Name: Volumetric Inlet <01>

Particles **SPH** **Region**

Seed Coordinates: 0 -0.2 0 m

Geometries

☒ ☐

☐ shear_box

☐ shearbox_gate

☐ shear_box_tray

Use Geometries to Compute ☐

Box bounds

Center Coordinates: 0 -0.2 0 m

Dimensions: 0.4 0.4 0.4 m

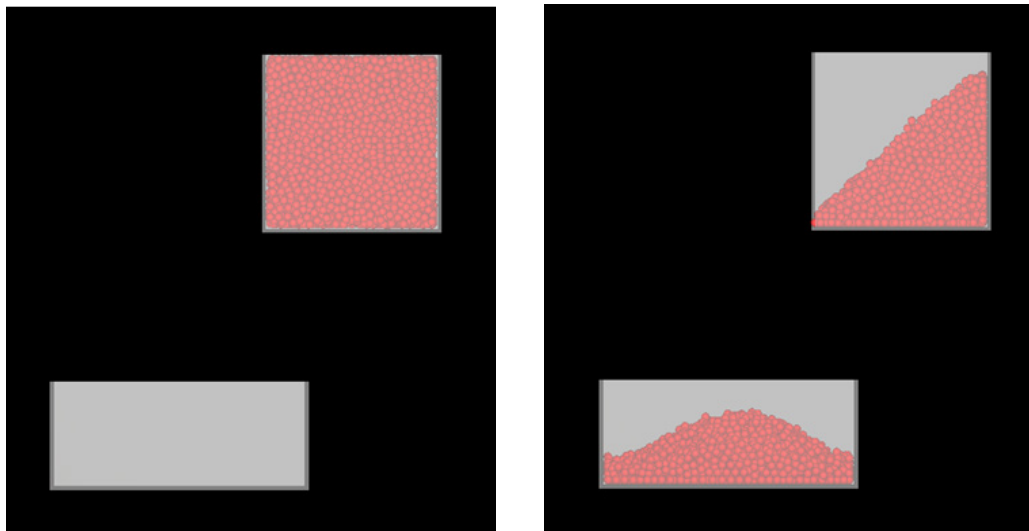
2.8 Solve Simulation

To then run this simulation, head to the **Solver** tab and adjust the Simulation Duration to 8 (s).

Before you solve the simulation, it is good to know the capabilities of the device you are using. Rocky has both *CPU* and *GPU* solver capabilities. For CPU processing, you are required to choose a Number of Processors. For GPU or Multi-GPU processing, you are required to select one or more compatible graphics card you want to use.

Once you have set the **Simulation Target** in the **General** tab, select the **Start** button to execute the simulation. We will leave all other values as default for this simulation.

The following is the solved simulation at the beginning and end of the test.

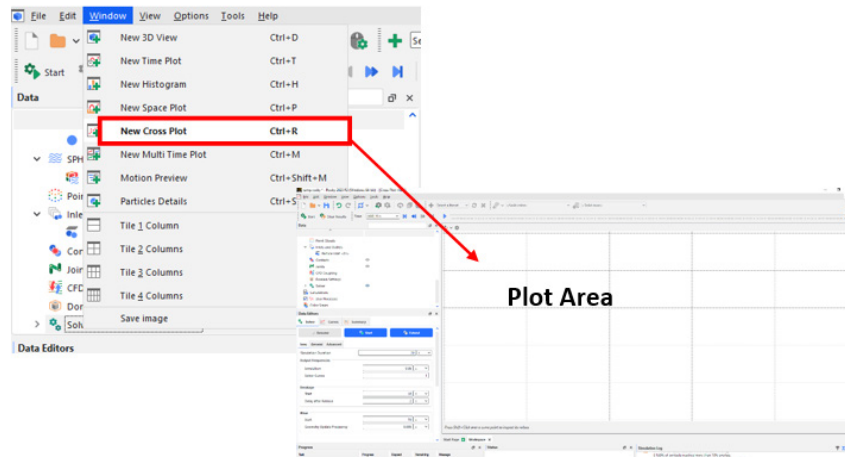


If you wish to remove the axes right click on the **3D window** and deselect **Bounding Box**. The particle inlet can also be hidden by hiding it in the **Geometry** tab.

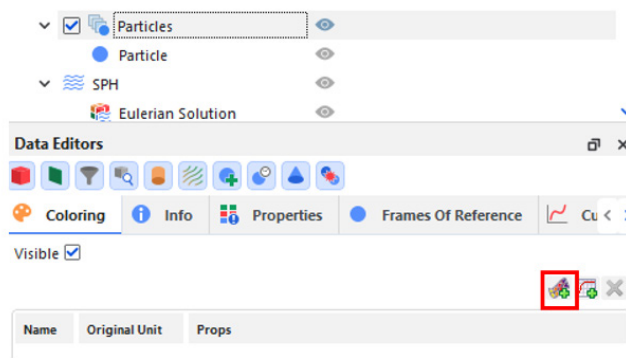
2.9 Analysis: Estimation of Shear Angle

The purpose of this tutorial was to run a simulation of a real test conducted to determine the angle of repose and shear angle of a material. Similar to the *Lifting Cylinder Tutorial*, we will export the shapes as *Cross Plots* to an image.

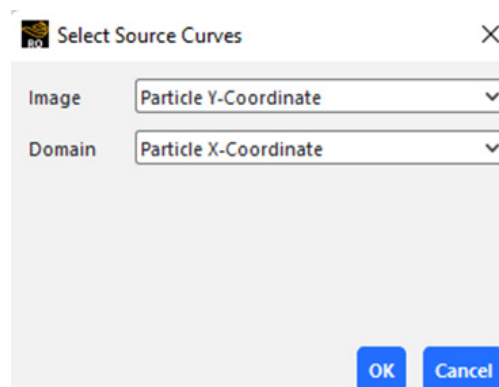
Using the **Window** tab, select **New Cross Plot**.



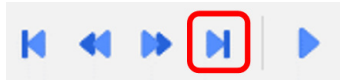
Now to select the variable to plot, select the **Particles** tab and under **Coloring**, click the small icon shown below.



You will then be prompted with a **Select Source Curves** window. Define using the drop-down box the **Image** and **Domain** as follows and select OK.



Note: If the current time step you are on isn't at the end of the simulation, the plot may appear empty or an irregular shape. To fix this, play the simulation to the end or using the Last Time Step button shown below skip to the end.



To modify and re-scale the cross-plot, right click in the cross-plot view and select **Settings**. You will be prompted by a **Windows Editor** window. Navigate to the **Axes** tab and define **Particle X – coordinate (m)** and **Particle Y-Coordinate (m)** under **Axis** as the following.

Axis

Particle X-Coordinate (m)

Particle Y-Coordinate (m)

▼ ☒ Axis Title

Text

Title Font

Axis Values

▼ Values

Unit

Format

Limits

Min

Max

Step

Axis

Particle X-Coordinate (m)

Particle Y-Coordinate (m)

▼ ☒ Axis Title

Text

Title Font

Axis Values

▼ Values

Unit

Format

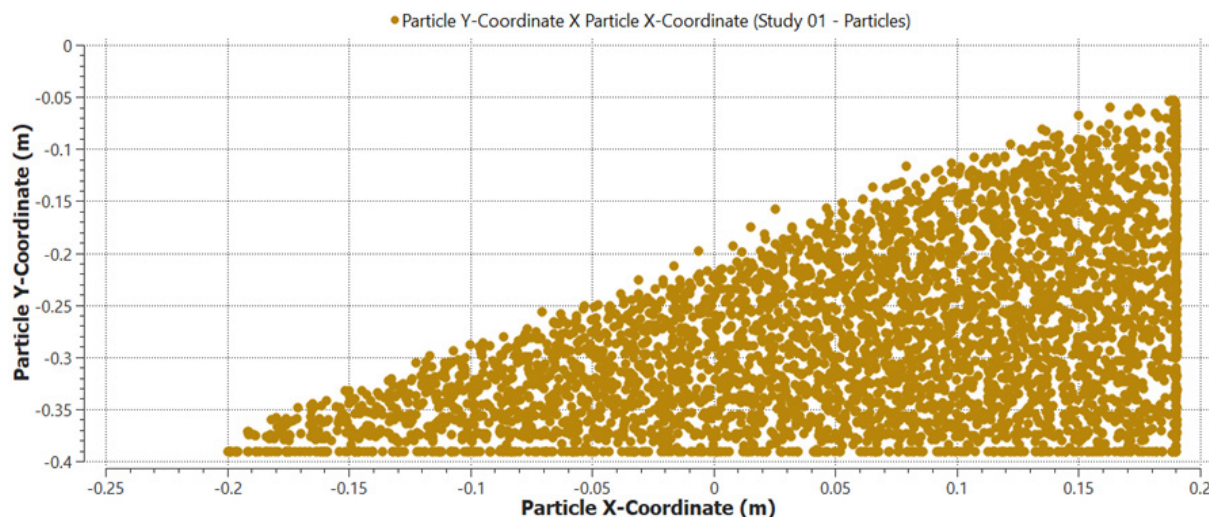
Limits

Min

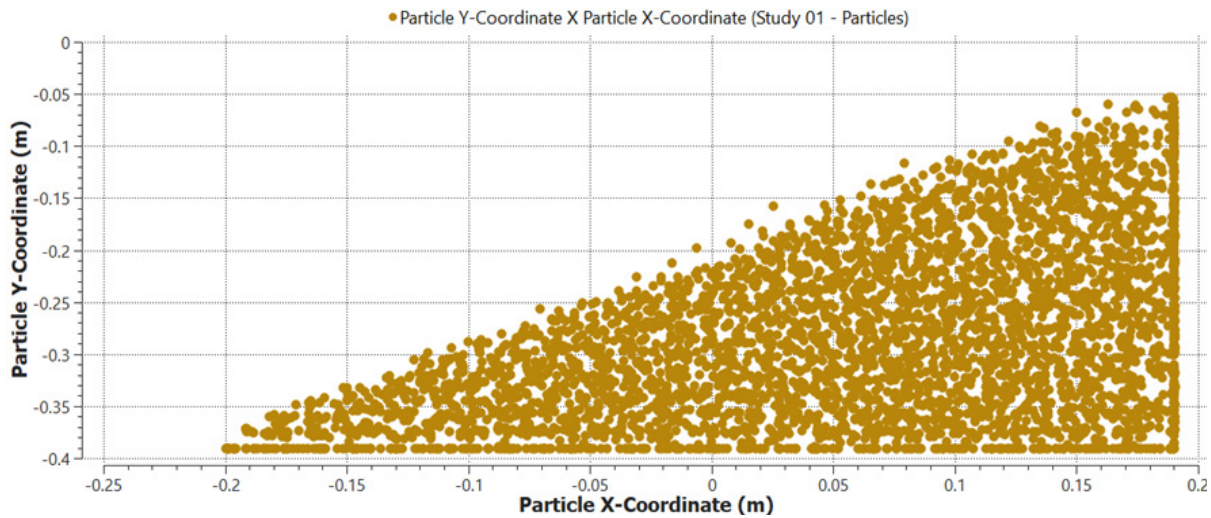
Max

Step

If done correctly the cross plot should appear like the following. Since we are only interested in the shear angle, we have narrowed the view into the material left in the *shearbox*.



Now, to calculate the angle of repose and shear angle, define the following gradients, similarly to the *Lifting Cylinder Tutorial*.



Shear Angle:

$$\theta_{Shear} = \tan^{-1} \left(\frac{-0.4 - (-0.03)}{-0.21 - 0.19} \right) \cong 42.8^\circ$$

Note: The values you end up with in your project may vary slightly with the ones shown in this tutorial.

Now repeat this to complete the following table by changing the particle rolling resistance and particle static and dynamic friction.

Shear Angle		Particle Rolling Resistance		
		0.4	0.5	0.6
Particle Static/Dynamic Friction	0.5			
	0.6			
	0.7			

© 2024 ANSYS, Inc. All rights reserved.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics focused on particle methods and discrete elements.

Ansyes Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, MicroProjects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit www.ansys.com for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.