Ansys Rocky Tutorial

Bulk Material Characteristic Testing I: Lifting Cylinder

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 lifting cylinder test with Ansys Rocky used in bulk material calibration. This test is performed to determine the static angle of repose of a bulk material.

Prerequisites

No previous experience in Ansys Rocky is required. All steps will be clearly presented for ease of following. Knowledge on Bulk Materials is preferred but not required.

Table of Contents

1. Introduction

Ansys Rocky

Ansys' Rocky is a specialized Discrete Element Method simulation software used to solve many complex engineering problems. It is designed primarily for the analysis of particle-based systems, such as granular materials, powders and other bulk materials. Rocky is capable of modeling real particle shapes including solids, 2D shells, and rigid and flexible fibers.

The core operational concept of DEM software revolves around identifying particle collisions between other particles and/or boundaries and computing the contact forces. This process typically employs the soft-sphere approach, where particles are rigid, and any deformation at contact is modeled as an overlap as seen in figure 1. DEM systematically monitors the movement of individual particles throughout the simulated domain, allowing for the prediction of each particle's position in the subsequent time step. For more information into the soft-sphere method and other variations of normal contact force models, please refer to the help menu found in Rocky.

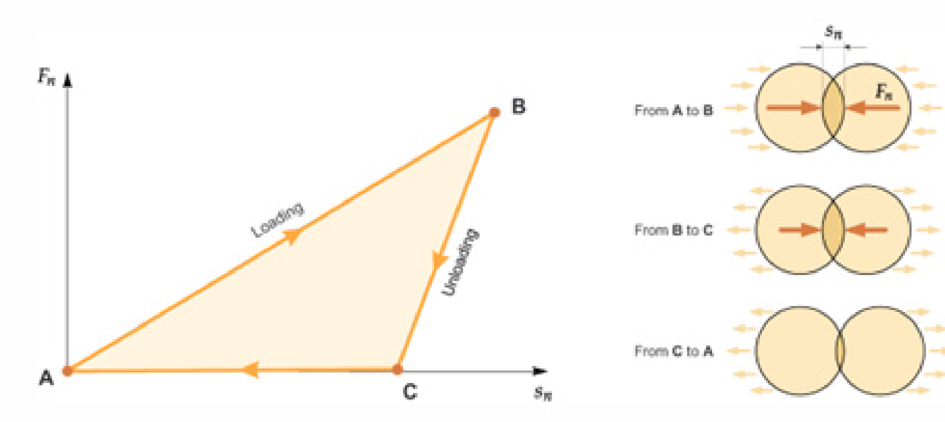


Figure 1 Typical normal-force overlap response for Ansys Rocky's default hysteretic linear spring model

Ansys Rocky is capable of many complex calculations, many of which will be explored in other tutorials on the Ansys Education Resources site.

Bulk Materials

The term *Bulk Materials* refers to large quantities of substances that are generally stored, transported, and handled in loose, granular, or particulate form. Bulk materials can include grains, ores, minerals, coal and other similar substances. These materials are often measured and processed in large quantities due to their nature, and they play a crucial role in various industries, including agriculture, mining, and manufacturing.

Many of these bulk materials are transported and stored in large infrastructure that require high efficiency, safety, and longevity. These infrastructures include conveyor systems, transfer chutes used as a transfer point in a conveyor system, hoppers, bins, and stockpiles used as a temporary storage of bulk material, and many other complex systems.

To effectively and efficiently handle, store and transport these materials, Discrete Element Method (DEM) simulations are utilized for optimizing systems. DEM simulations allow users to optimize the operation of a bulk material system by closely simulating material behaviors by computing individual particles as a discrete element. These DEM simulations require calibration to produce accurate results by closely representing the behavior of the desired material. This calibration process is often done by numerous tests and comparisons to ensure the most well represented material properties. Many factors, including particle type, size and moisture content can drastically change the behaviors of these bulk materials, turning a smooth flowing material into a problematic material resulting in major issues including blockages and spillage.

The most common bulk material characteristic is the Angle of Repose (AoR), which will be outlined in this tutorial by means of the *Lifting Cylinder Test*. The AoR is referred to as the steepest slope of the unconfined material, measured from the horizontal plane on which the material can be amassed without collapsing, which can be seen in figure 2. Generally, the AoR is related to the static friction coefficient and internal shear angle.

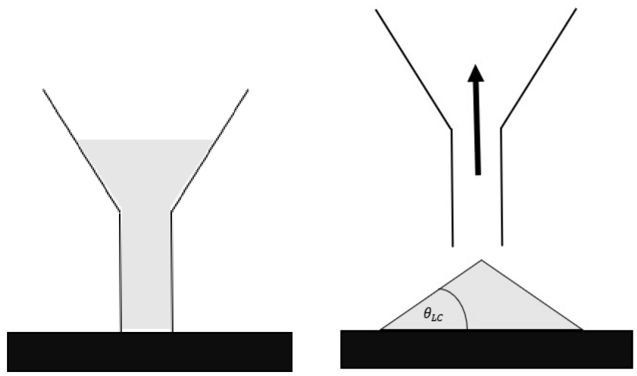


Figure 2 Lifting cylinder test for determining static angle of repose

Benefits of the AoR are its uses in indicating a materials cohesiveness, referring to the Carr classification of flowability for powders in table 1. However, testing for a materials AoR is more typically done for materials with low cohesiveness, as it is hard to obtain for cohesive materials, rather more indirect methods are used for these material types.

Table 1 Carr classification of flowability

Description	Angle of Repose
Very free flowing	<30°
Free flowing	30-38°
Fair to passable flow	38-45°
Cohesive	45-55°
Very cohesive (non-flowing)	≥55°

Understanding the flowability of a material is crucial for the design of transfer chutes, hopper, bins and other transportation and handling systems, hence the importance for accurate DEM calibrations. There are many factors that are to be considered in these designs that can the determinant of a successful system, most of which are recognized in the calibration phase.

The angle of repose is strongly influenced by the particle's characteristics, including the shape and surface, and material properties such as sliding and rolling friction and particle stiffness; however, it should be noted that particle stiffness has an influence on the AoR between 1e6 to 1e10 Pa, values above this have very minimal effect. The AoR typically sees an increase with an increase of friction coefficients, decrease of the particle size and a deviation from spherical particles. However, for DEM simulations, some of these characteristics are not typically modeled as they can drastically increase the solve time, hence, some simplifications need to be utilized. Some of these simplifications include the use of spherical particles and a lower particle stiffness.

This tutorial will explore various parameters that can affect the AoR of a material, replicating those material properties listed above, including interparticle friction, resulting from differing moisture contents and particle rolling resistance as a practical way to represent the non-sphericity of particles. The coefficient of restitution (CoR) is another important parameter that can be used to alter a particles behavior. The CoR is used to describe the ratio of the final velocity to the initial velocity between two objects after a collision. In other words, it describes the objects 'bounciness'.

For further readings into the Angle of Repose of materials and additional testing methods and the influential properties, the articles referenced have been attached.

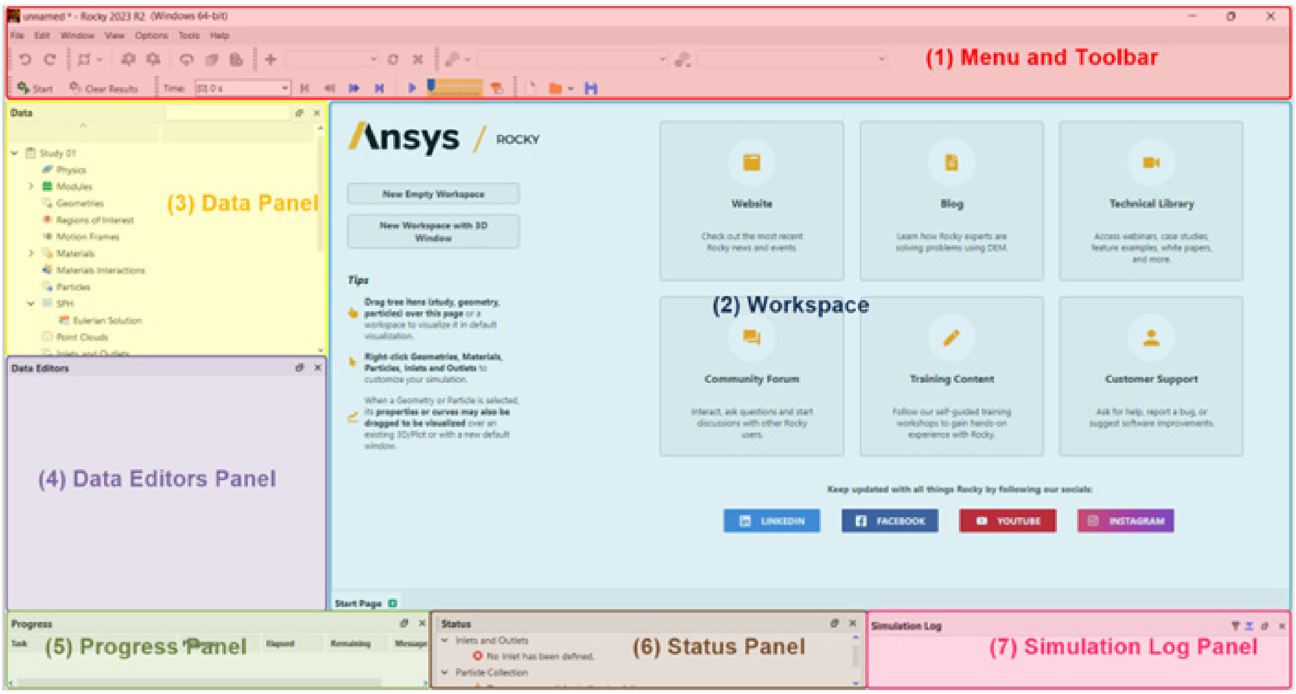
Further Readings

Al-Hashemi, H. M., & Al-Amoudi, O. S. (2018, February). A review on the angle of repose of granular materials. Powder Technology, pp. 397-417.

Ye, F., Wheeler, C., Chen, B., Hu, J., Chen, K., & Chen, W. (2019, February). Calibration and verification of DEM parameters for dynamic particle flow conditions using a backpropagation neural network. Advanced Powder Technology, pp. 292-301.

2. Introduction to Ansys Rocky Interface (UI)

Once a new project has been created in Ansys Rocky[1], which is demonstrated in the following section, the following screen will appear. This screen has been highlighted fort his tutorial to outline the different sections within the UI.



The Rocky user interface (UI) is customizable, you can add/remove/reposition any window or panel available. To change back to the default, select View from the main Toolbar, and then click Reset layout.

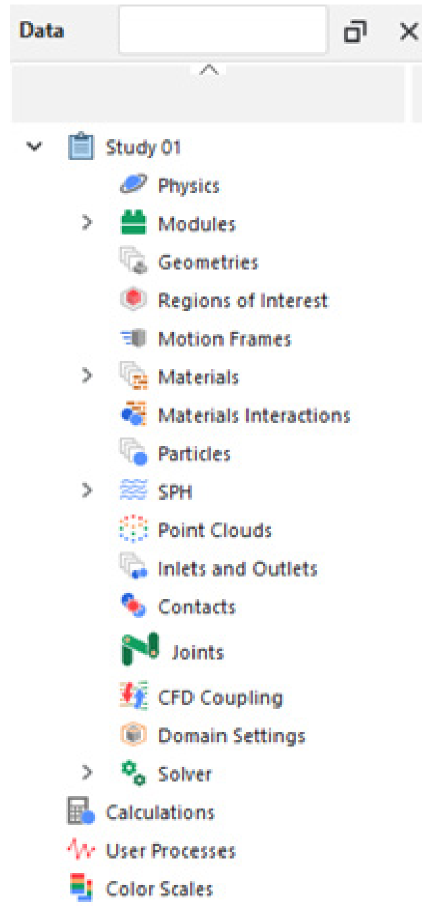
The default layout contains the following components:

(1) ***Menu and Toolbar:*** Contains the main program menus, shortcuts, camera options, Timestep controls, and display tools.
(2) ***Workspace:*** Displays the available windows that have been opened for the project (3D Views, Motion & Particle Previews, and Plots & Histograms).
(3) ***Data Panel:*** Displays the project tree through which the setup parameters are defined.
(4) ***Data Editors Panel:*** Displays the details of the item that is selected in the Data panel.
(5) ***Progress Panel:*** Shows the processing tasks currently being performed.
(6) ***Status Panel:*** Shows any warnings or errors regarding the current project.
(7) ***Simulation Log Panel:*** Lists any Solver warnings or errors.

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

Data Panel Items

To set up any Rocky simulation, from the ***Data*** panel under ***Study 01***, follow the items listed top-down and one-by-one:

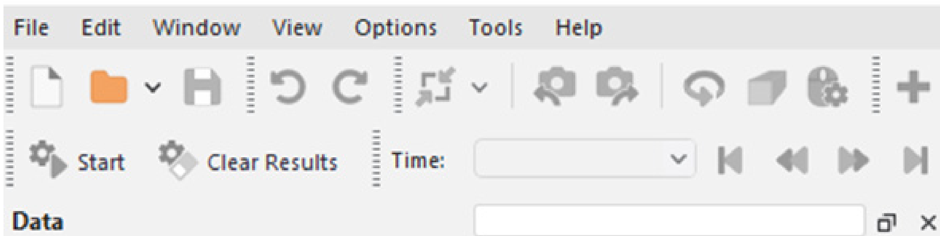


- *Study 01:* Change the study name from the default (Study 01) and add a Description.
- *Physics:* Set physical conditions (Gravity, Momentum, Coarse-Graining, and Thermal models)
- *Modules:* Enable additional models and data collection options.
- *Geometries:* Import, add, and edit geometry components.
- *Regions of Interest:* (For certain external Modules only.) Create a Cube or Cylinder region where custom calculations can be performed.
- *Motion Frames*: Add and preview movement to the simulation components (Geometries).
- *Materials:* Define materials and set densities and other properties.
- *Materials Interactions:* Define adhesion and other properties for materials interactions.
- *Particles:* Create particles, set size distributions, and preview particle shapes.
- *SPH:* Define fluid (SPH elements) parameters and boundary conditions.
- *Point Clouds:* (For certain external Modules only.) Import field data that is defined in a text file.
- *Inlets and Outlets:* Define particle and SPH inlets and/or outlets rates and release locations.
- *Contacts:* Enable the collection of contact data.
- *Joints:* Enable the collection of joints data.
- *CFD Coupling:* Set up LBM air flow or define one- or two-way coupling with Ansys Fluent fluid dynamics solver.
- *Domain Settings:* Define the domain behavior and periodic boundaries.
- *Solver:* Define how the DEM solver processes the simulation and collects data.
- *Particles Calculations:* Displays user-defined particle properties, such as particle tagging.
- *User Processes:* Displays user-defined processes, such as analysis cubes and planes.
- *Color Scales:* Shows display details of all plotted properties.
- *Note:* These items will be covered in more detail in later tutorials.

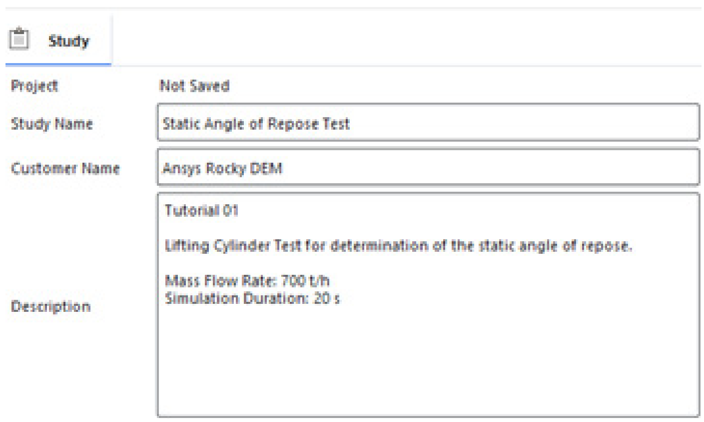
3. Exercise: Simulating Lifting Cylinder Test

Download the two .stl files supplied with this document, *AOR_Base* and *AOR_Funnel*.

Open Ansys Rocky and create a new project using the ***New Project*** icon.

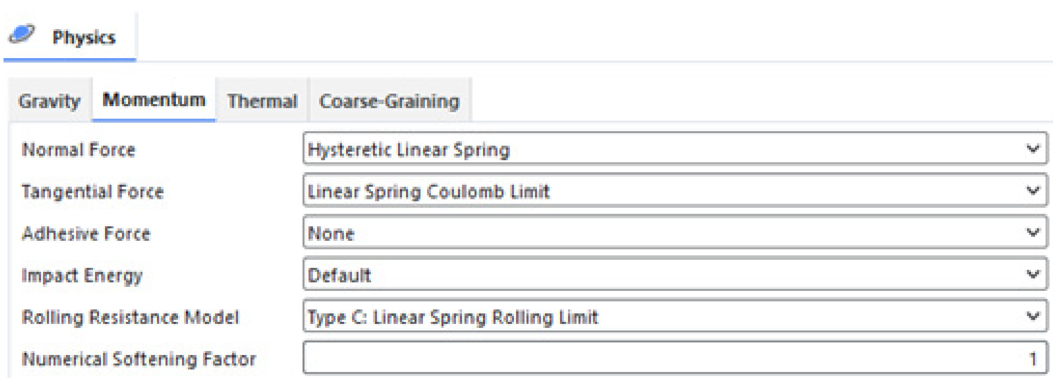


Before you begin setting up the project, it is good practice to first save the file. Using the ***Save as*** option in the File icon, save the project in an appropriate location.



Physics

Now, it is important to set the correct physics definitions before any simulation. This can be done through the ***Physics*** tab. Under the ***Momentum*** tab, set the parameter for ***Rolling resistance Model*** as shown below, while leaving all others as default.



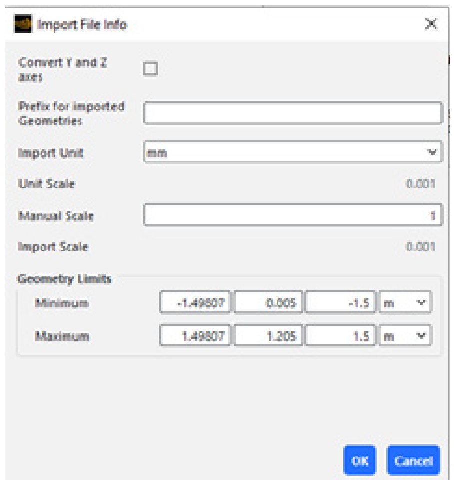
The Rolling Resistance Model selected as Type C: Linear Spring Rolling Limit is the recommended option for simulations where the effects of particle rolling resistance are of importance. This allows the inclusion of a damping value to suppress oscillations present in particles from interactions between other particles and boundaries.

> Note: It is important to observe which direction Gravity will be acting in your simulation. In this simulation it will be acting in the default negative Y direction, but can be changed in the Gravity tab.

Geometry

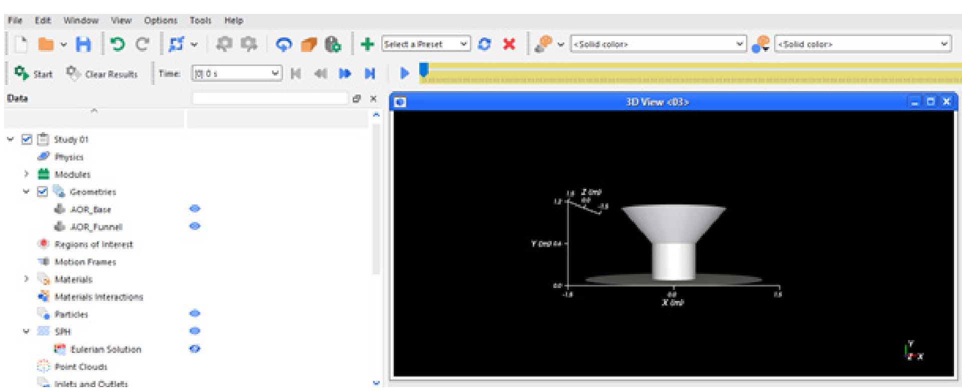
To import the previously saved geometry, right click on the ***Geometry*** tab and select ***Import Wall***. You will then be prompted by a window, find the two .stl files (AOR_Base and AOR_Funnel) and import them both.

You will then be prompted with another window. Change the ***Import Unit*** to *mm* and leave the rest as default. If done correctly it should appear like the following. It is important to check the ***Minimum*** and ***Maximum Geometry Units*** to ensure it is in the correct units.



> Note: .stl files are not saved with embedded units, so it is important the correct units are selected when importing geometries.

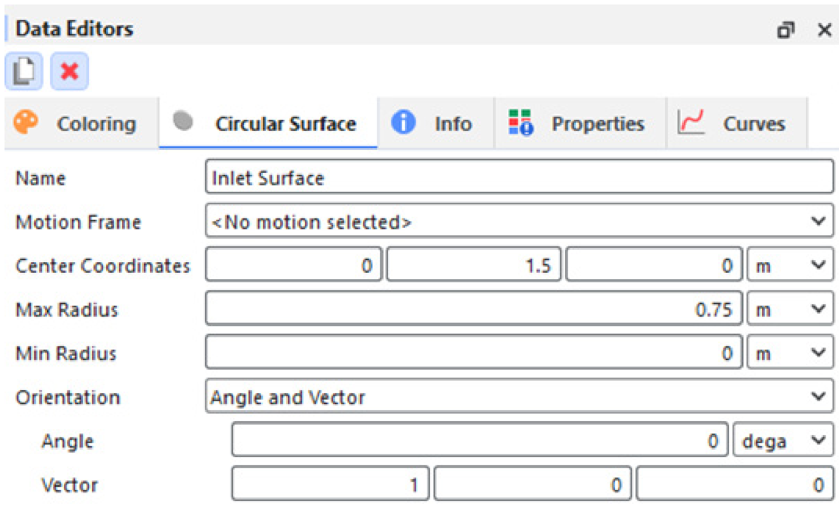
To then view this imported geometry, in the ***Workspace***, select the ***New Workspace with 3D View***. Alternatively, you can select and drag the ***Geometry*** entity into the workspace. If done correctly, it should appear as below.



Inlet Creation

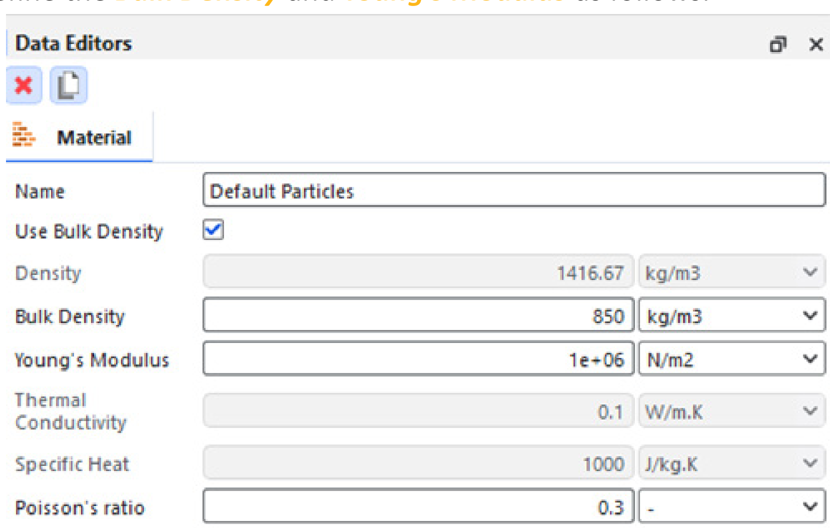
Once the geometries are imported, a surface is to be created to define an inlet for particles. To create a surface, right click on ***Geometry*** and select ***Create Circular Inlet***. A surface will now have been created under the Geometry tab labeled ***Circular Surface <01>***. It can be beneficial to rename this surface something you will remember, as you will need to define an inlet at this location.

To adjust the parameters of this surface you have just made, click on the Circular Surface <01> entity and define the ***Name***, ***Centre Coordinates***, ***Max Radius*** and ***Min Radius*** to the following. This surface will be defined as an inlet in a later step.



Particle Material Definition

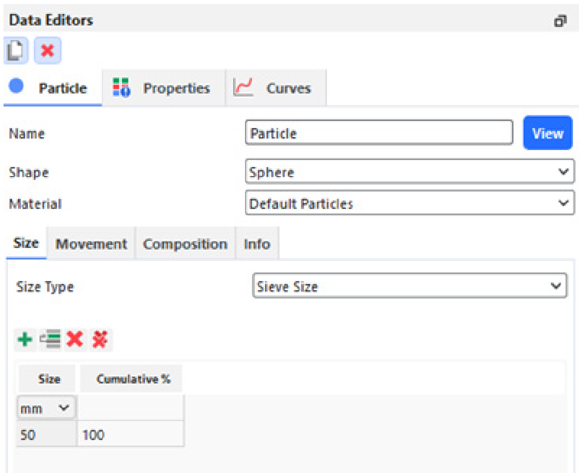
To define parameters for the particles just created, select ***Default Particles*** under the ***Materials*** tab. You will need to define the ***Bulk Density*** and ***Young's Modulus*** as follows.

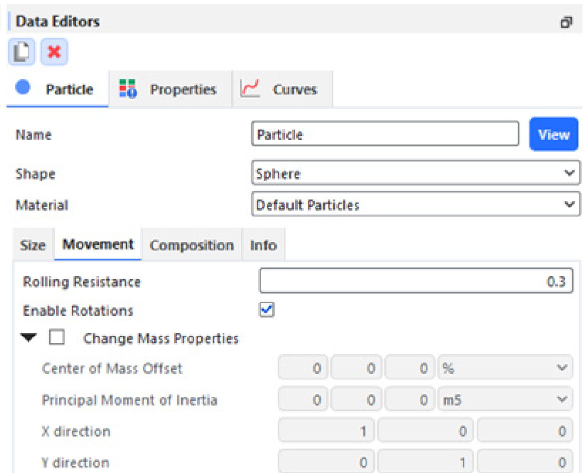


The ***Young's Modulus*** is set to a smaller value to decrease solve time by reducing the loading and unloading stiffness of the particles.

Particle Creation

To create a new particle, right click on the ***Particle*** tab and select ***Create Particle***. This particle will appear under the Particle tab as ***Particle <01>***. Similar to the surface creation, define the ***Name***, and under ***Size***, define the ***Size*** and under the ***Movement*** tab define the ***Rolling Resistance*** to the following and leave all else default.

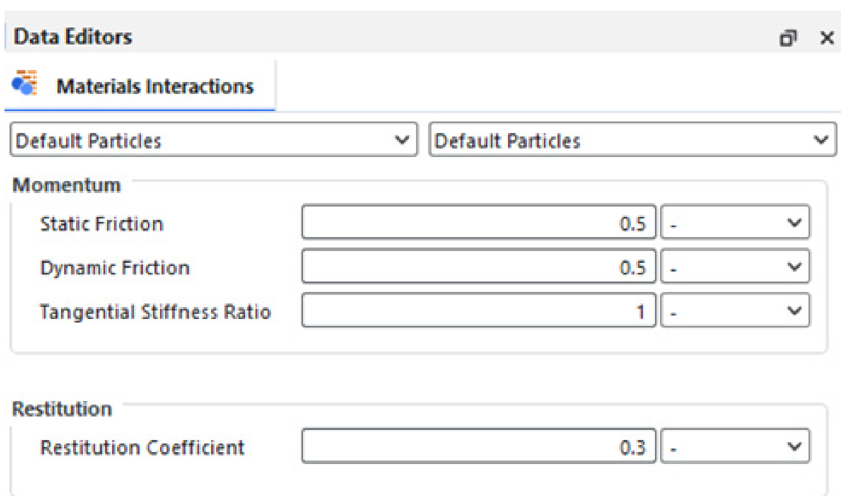




To view this particle in a separate 3D view, select the ***View*** icon in the above image. This is generally more common for irregular shape particles.

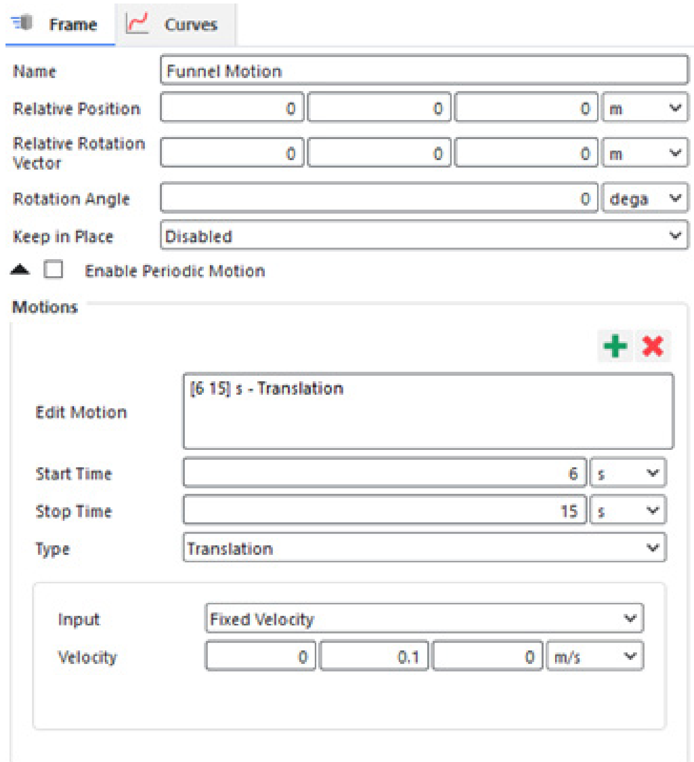
Material Interaction Definition

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 Fiction*** as follows, while leaving all other parameters as default.

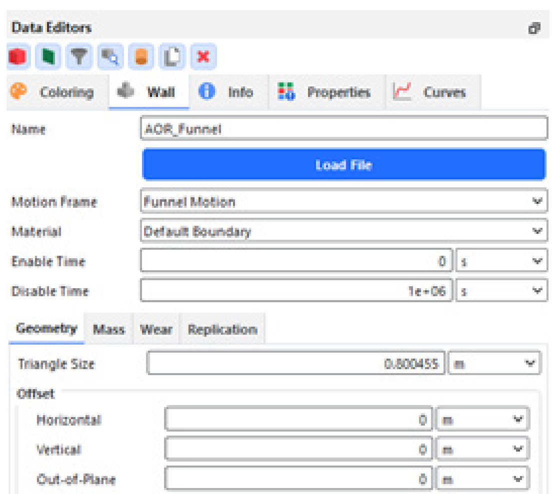


Create Motion Frame

To simulate the funnel geometry moving to allow the particle to fall through we have to set a motion frame. To do this right click on ***Motion Frames*** and select ***Create Motion Frame***. To add a motion, select the green plus button (***Add Motion***) and define the ***Name***, ***Start Time***, ***Stop Time*** and ***Velocity*** as follows, and leave all other parameters as default.

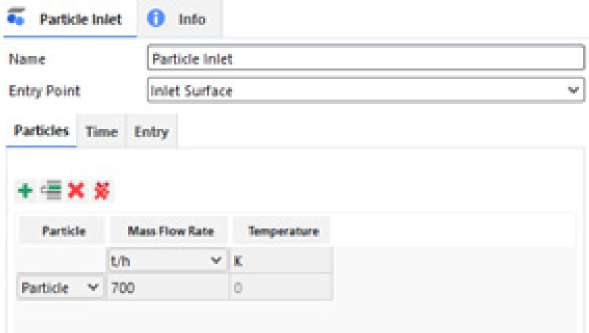


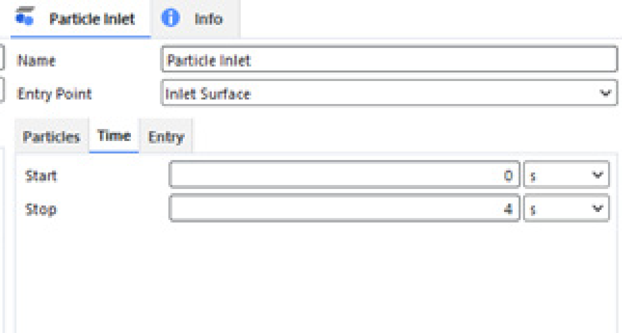
To then apply this motion frame to our geometry, we need to click on the ***AOR_Funnel*** geometry and set the ***Motion Frame*** to our ***Funnel Motion*** frame.



Inlet Definition

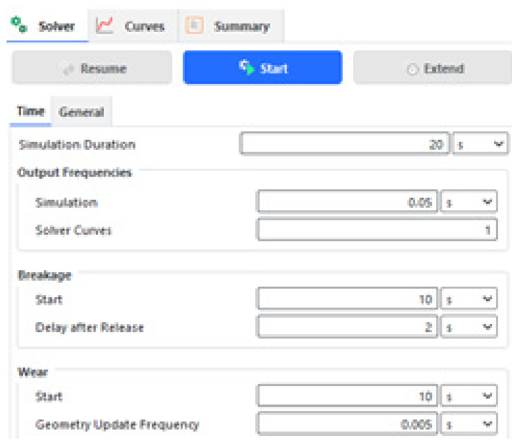
To define our previously made surface as a particle inlet, we need to right click on ***Inlets and Outlets*** and then select ***Create Particle Inlet***. This Inlet will appear under the ***Inlets and Outlets*** tab, as ***Particle Inlet <01>***, select this. In here we will add a particle by selecting the green plus button (***Add***) and select ***Particle*** (the previously created particle) under the ***Particle*** tab and set ***Mass Flow Rate*** to 700 (*t/h*). We also need to define the inlets ***Name***, ***Entry Point*** and ***Start*** and ***Stop*** under the ***Time*** tab to the following, leaving all other values as default.



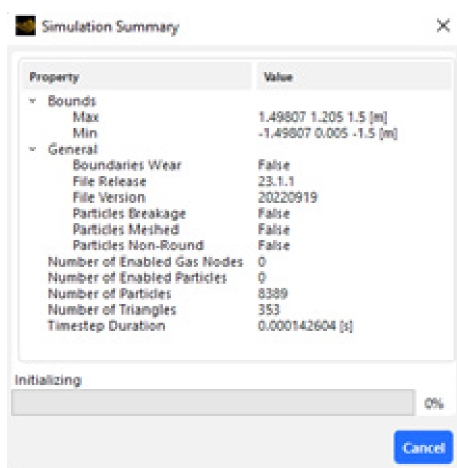


Solve Simulation

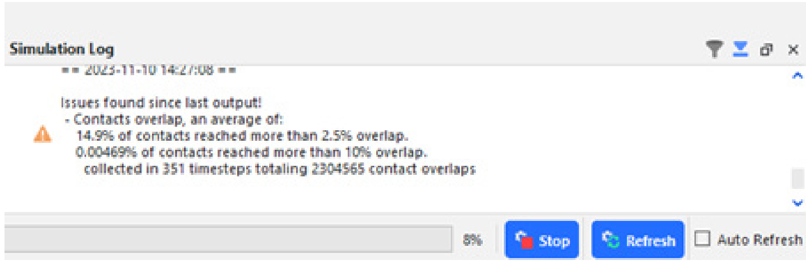
To then run this simulation, head to the ***Solver*** tab and select the ***Start*** button. We will leave all of these values as default for this simulation.



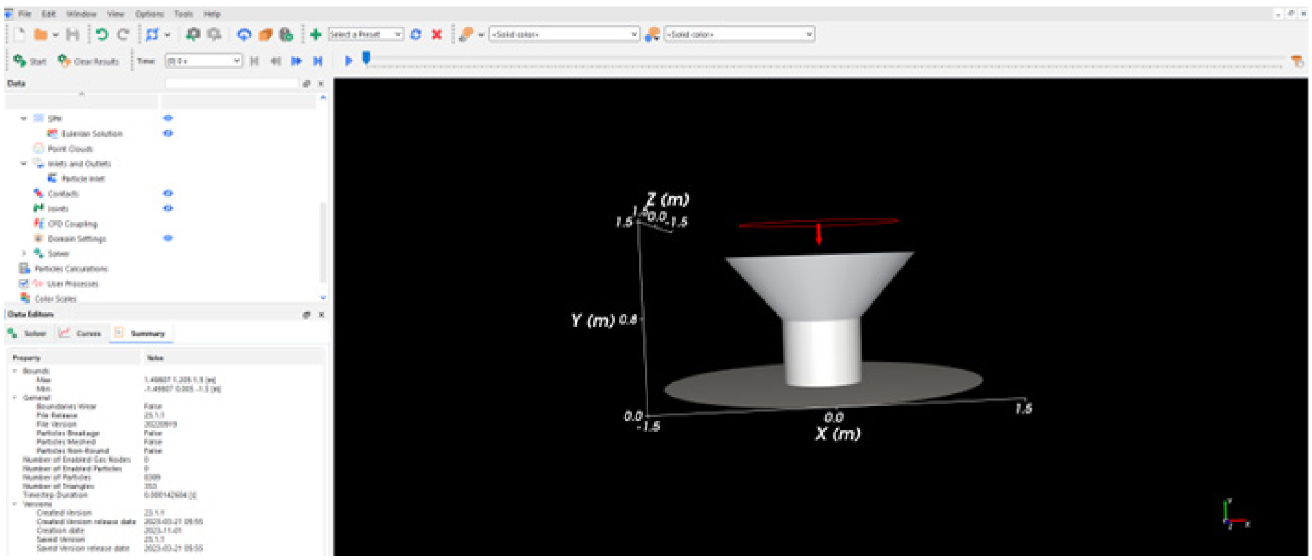
Once you have started the simulation, a ***Simulation Summary*** window will appear. This gives you a brief summary on some simulation properties, these can also be found in the ***Summary*** tab under the ***Solver*** section.



When running you can also manually ***Refresh*** the simulation to show the solution up to that time step, or set it up to ***Auto Refresh***, however, this will require more processing time. These can be found in the bottom right corner along with the ***Stop*** icon.



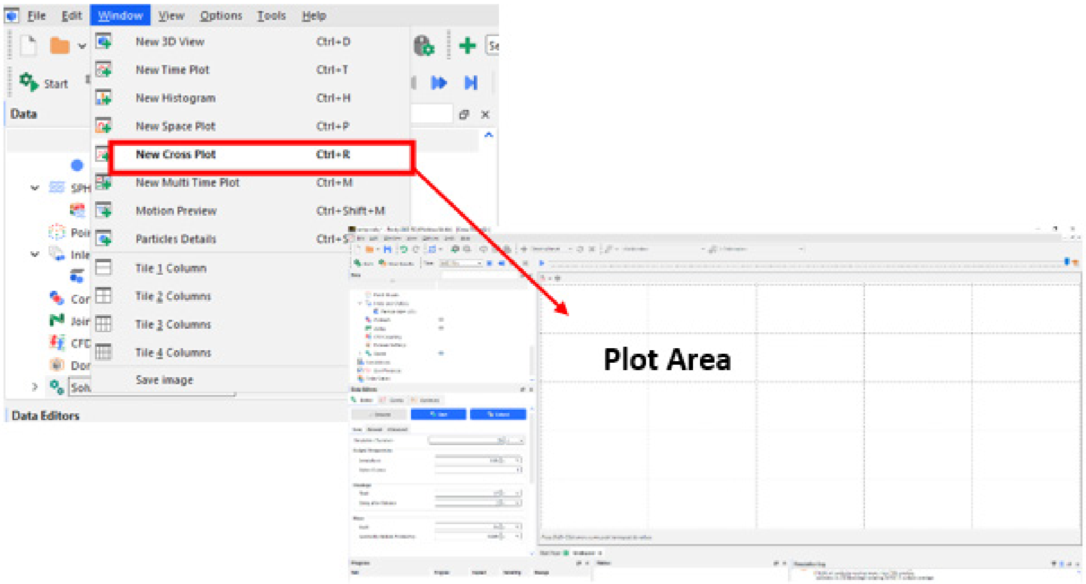
Once the simulation is complete, to play it, press the blue play button (***Play Simulation***) at the top right of the 3D Viewer. You can also zoom/rotate to visually inspect the simulation.



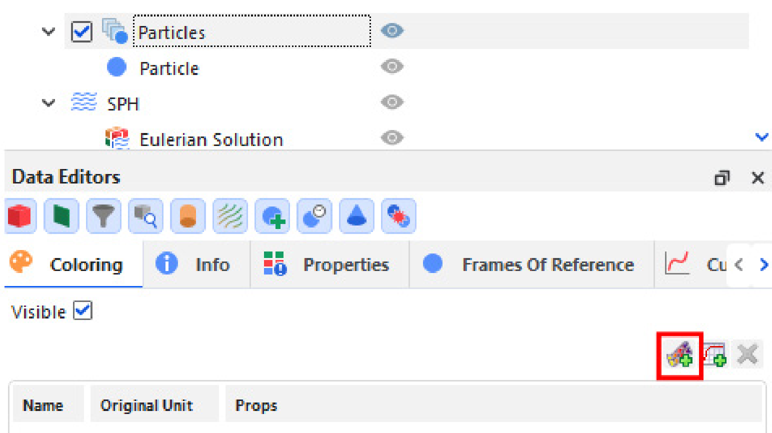
4. Analysis: Estimation of Angle of Repose

The purpose of this tutorial was to run a simulation of a real test conducted to determine the angle of repose of a material. Now, we are going to investigate how to estimate that angle. You can determine the angle of repose by using online tools or a CAD package, alternatively you can use the following procedure.

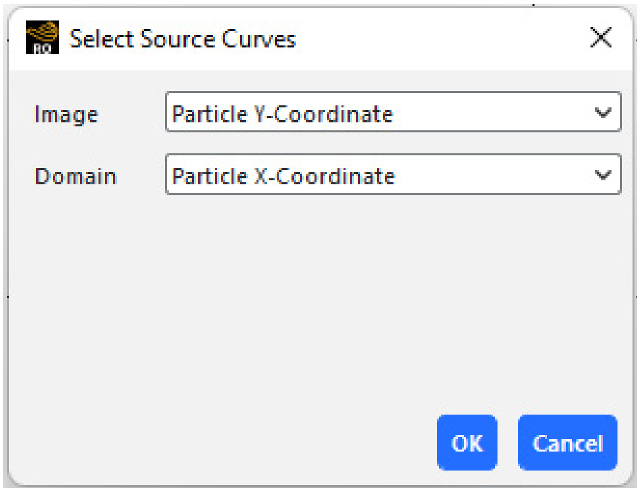
Using the ***Window tab***, select ***New Cross Plot***.



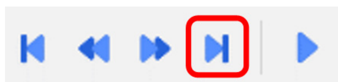
Now to select the variable to plot, select the ***Particles*** tab and under ***Coloring***, click the small icon shown below.



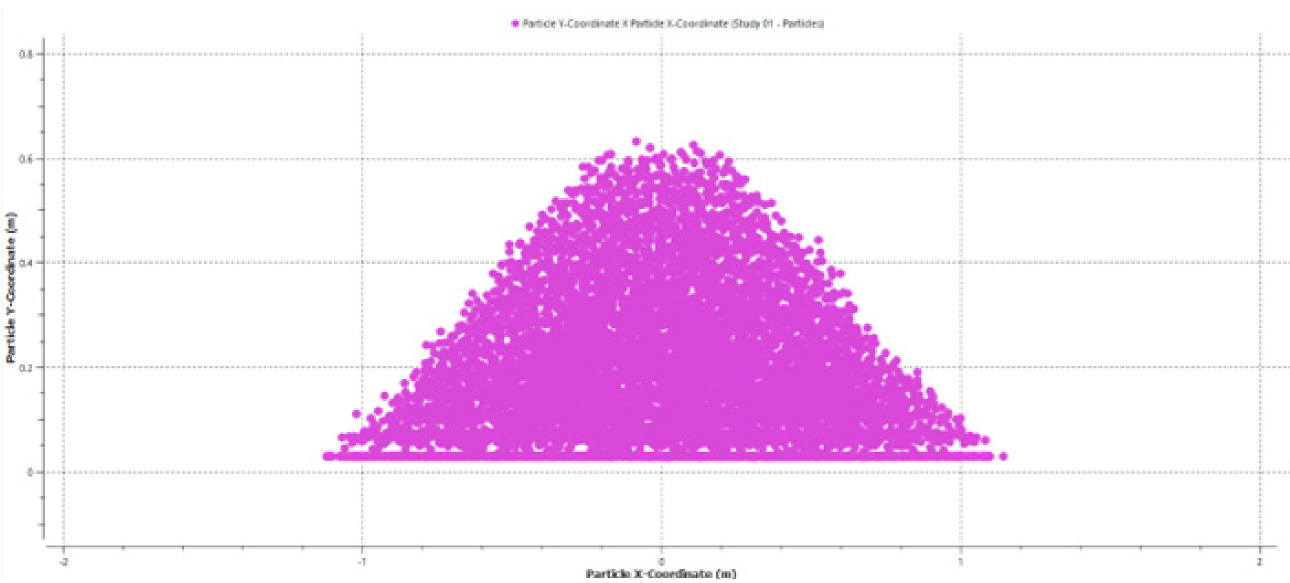
You will then be prompted with a ***Select Source Curves*** window. Define using the drop-down box the ***Image*** and ***Domain*** as follows and select OK.



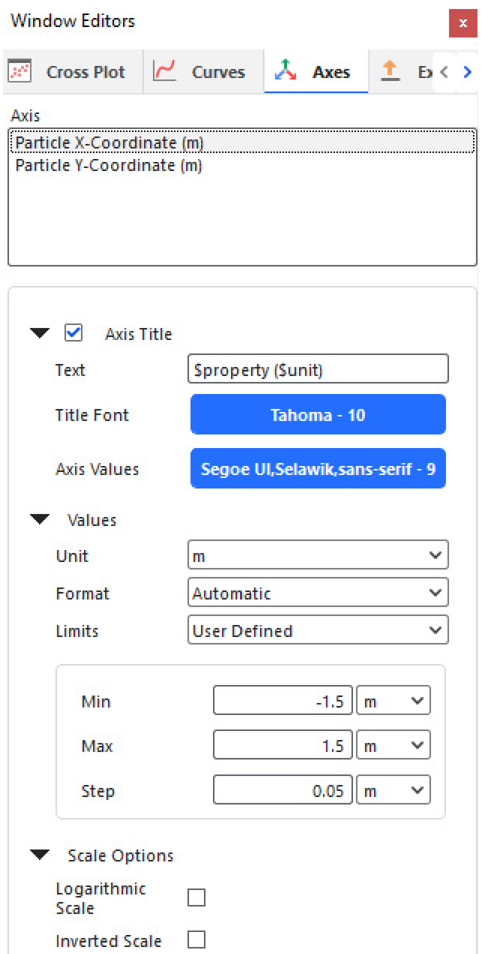
Note: If the current time step you are on isn't at the end of the simulation, the plot may appear empty or an irregular shape. To fix this, play the simulation to the end or using the Last Time Step button shown below skip to the end.



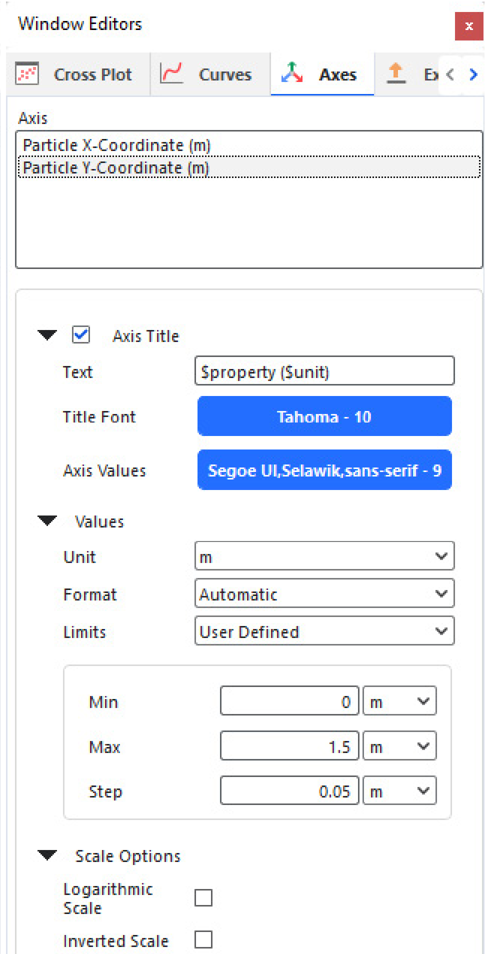
If done correctly the cross plot should appear like the following.



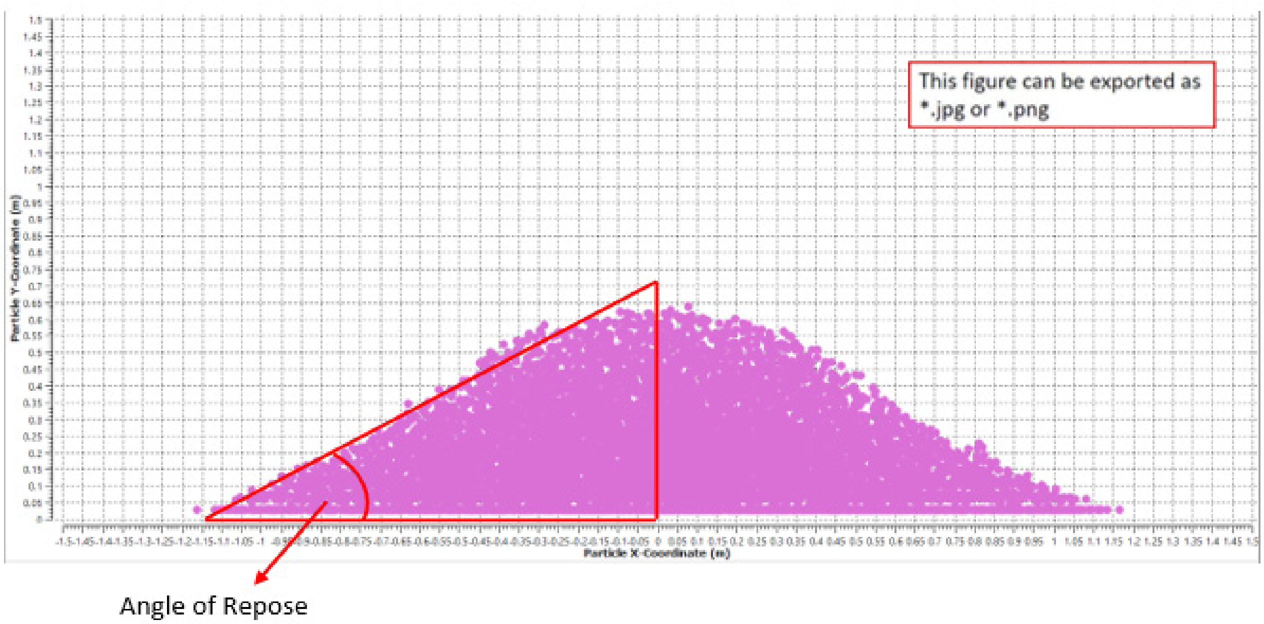
To modify and re-scale the cross-plot, right click in the cross-plot view and select ***Settings***. You will be prompted by a ***Windows Editor*** window. Navigate to the ***Axes*** tab and define ***Particle X – coordinate (m)*** and ***Particle Y-Coordinate (m)*** under ***Axis*** as the following.



Window Editors
Cross Plot
Curves
Axes
Axis
Particle X-Coordinate (m)
Particle Y-Coordinate (m)
Axis Title
Text
$property ($unit)
Title Font
Tahoma - 10
Axis Values
Segoe UI,Selawik,sans-serif - 9
Values
Unit
m
Format
Automatic
Limits
User Defined
Min
-1.5
Max
1.5
Step
0.05
Scale Options
Logarithmic Scale
Inverted Scale



Window Editors
Cross Plot
Curves
Axes
Axis
Particle X-Coordinate (m)
Particle Y-Coordinate (m)
Axis Title
Text
$property ($unit)
Title Font
Tahoma - 10
Axis Values
Segoe UI,Selawik,sans-serif - 9
Values
Unit
m
Format
Automatic
Limits
User Defined
Min
0
Max
1.5
Step
0.05
Scale Options
Logarithmic Scale
Inverted Scale



This figure can be exported as *.jpg or *.png
Particle Y-Coordinate (m)
Particle X-Coordinate (m)
Angle of Repose

By exporting this as an image file, we can open with an application that allows sketching (Microsoft Office Word works well).

To calculate the angle of repose in this case we have the x-coordinate as 1.14m and the y-coordinate as 0.72m. Using Trigonometry, we have:

$$AOR = \tan^{-1}\left(\frac{0.72}{1.14}\right) \cong 32°$$

> Note: The values you end up with in your simulation 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.

AoR		**Particle Rolling Resistance**		
		0.4	0.5	0.6
Particle Static/Dynamic Friction	0.5			
	0.6			
	0.7			

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.

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