

Ansys Rocky Tutorial

Bulk Material Characteristic Testing IV: Transfer Chute Analysis

Developed and curated by the Ansys Academic Development Team

Angus Foley, University of Newcastle

education@ansys.com

Anslys Software Used

This case study uses Ansys Rocky™, the particle dynamics simulation software.

Summary

This tutorial will cover the setup and simulation of an operational transfer chute by means of a feeding and receiving conveyor belts.

Prerequisites

It is recommended to complete the first three Bulk Material Characteristic Testing tutorials before this exercise. This can be found on the [Ansys Education Resources site](#).

Table of Contents

1. Introduction.....	3
2. Exercise: Simulating Transfer Chute.....	4
2.1 Physics	4
2.2 Modules.....	4
2.3 Geometry	4
2.4 Inlet Creation	5
2.5 Particle Creation	6
2.6 Particle Material Definition	6
2.7 Material Interaction Definition	6
2.8 Motion Frames	7
2.9 Inlet Definition.....	8
2.10 Solve Simulation	8
3. Analysis.....	8
3.1 Steady State Analysis.....	8
3.2 Wear Analysis	10
4. Extension	11
4.1 Investigate the various display and analysis options.	11

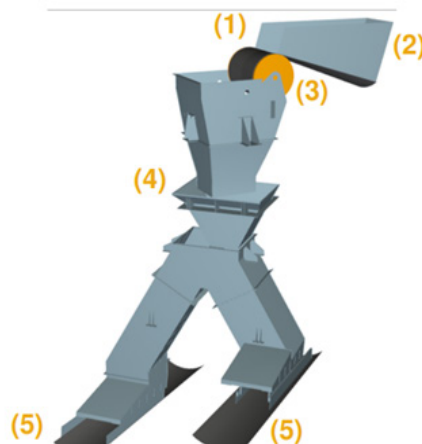
1. Introduction

Previous tutorials have explored the calibration side of DEM; however, this tutorial will focus more on where these calibrated materials are used. Before we begin the tutorial, it is important to understand how the calibration tests from the previous tutorials are used to calibrate such materials. Technicians will physically conduct numerous tests, some of which will include at least one of the three tests explored, which will produce results for an angle of repose, shear angle and others. The use of images and videos of the material behaviors can also be helpful in the calibration.

Using this data, Ansys Rocky experts will conduct multiple simulations (based off what we saw in the previous tutorials) while tweaking certain parameters to imitate the materials behaviors in that certain test. This is why having more tests can allow for much more accurate results, by having a material that behaves the same as the real material in several tests.

Typically, it is common for experts to increase the size of particles in the simulation compared to that of the real material, as for larger simulations, this will drastically decrease the computational time. However, this is done in the calibration stage, for example if a material tested has a size range of -10mm, meaning a top size of 10mm and all others less than, a distribution of 20mm - 40mm particles may be used for simulation (dependent of the size of the simulation and number of particles required). But it will be calibrated so that the large particles will behave similarly to that of the smaller particles.

This *Transfer Chute* tutorial will explore the operation of a feeding conveyor, transferring material through a chute on to a receiving conveyor belt, shown below. This simulation assumes the use of calibrated particles.



- The geometry in this tutorial is composed of:
 - (1) Feed Conveyor
 - (2) Skirt
 - (3) Pulley
 - (4) Chute
 - (5) Receiving Conveyors
- The complete geometry is subdivided into several parts in order to apply different movements to each one. In the tutorial directory, each .stl file can be found.

If a material has been calibrated correctly, the simulation should closely match real world results. The importance of this will be seen through this tutorial as it explores wear metrics and loading scenarios, which can be used to determine the efficiency and effectiveness of the transfer chute design.

2. Exercise: Simulating Transfer Chute

Note: This tutorial assumes the completion of Bulk Material Characteristic Testing Tutorials I, II and III. The simulation setup steps will be much briefer than previous, please refer to earlier tutorials for guidance if needed.

Download the five .stl files uploaded with this lab document: *chute*, *feed conveyor*, *pulley*, *receiving conveyor 1*, and *receiving conveyor 2*.

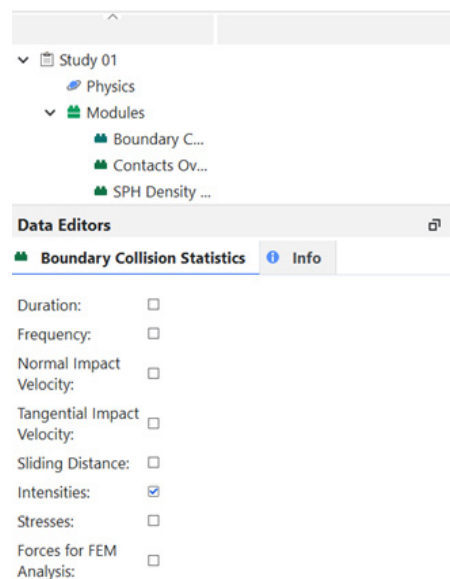
Open the Ansys Rocky software¹ and create a new project using the **New Project** icon, and save to an appropriate location.

2.1 Physics

Similarly, to the previous tutorials, set the **Rolling Resistance Model** under the **Momentum** tab in **Physics** to the **Type C: Linear Spring Rolling Limit**. For this tutorial we will also set the **Gravity** definition to act in the negative z-direction.

2.2 Modules

In the **Modules** tab, select the check box for **Boundary Collision Statistics**. You will then notice the **Boundary Collision Statistics** appears in the drop-down box shown below. Here you will select the check box for **Intensities**:

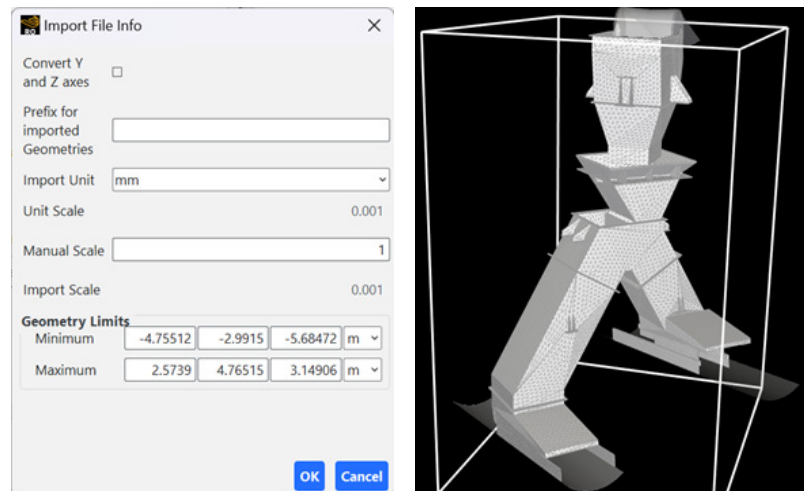


2.3 Geometry

Import the five geometry files supplied with this document and set the **Import Unit** to mm, leaving the rest as default. It is important to check the **Minimum** and **Maximum Geometry Limits** to ensure it is in the correct units.

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

Once you have imported the geometries, the mesh sizing for each needs to be refined, as this will provide a more accurate wear metric. To do this navigate to the **Geometry** tab and under the **Wall** sub-tab, change the **Triangle Size** to **100 mm** for each geometry. To then view this generated mesh, under a geometry of your choice, navigate to the **Coloring** tab and select the check box for **Edges**, you should see the below image (if you selected the chute geometry). However, we will leave this off for now.

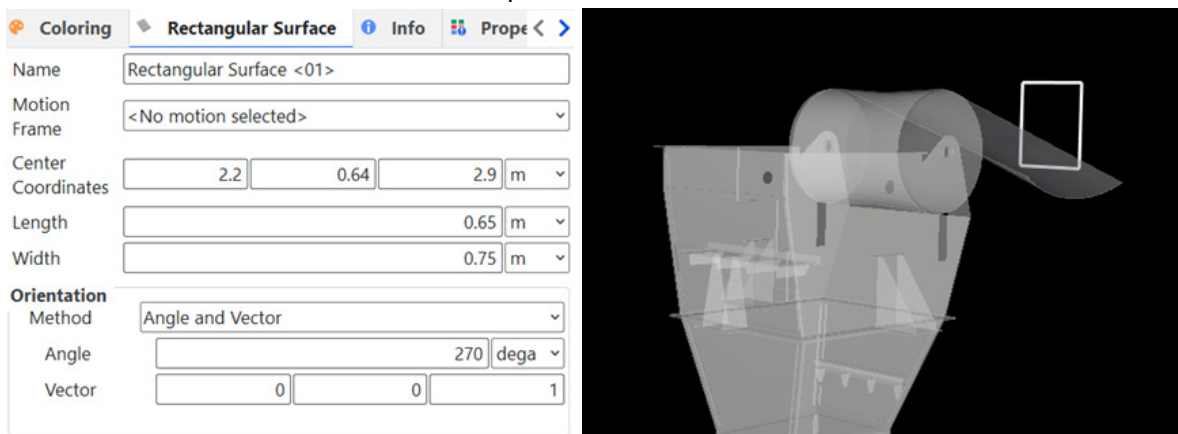


Open a new 3D workspace to view the imported geometry and set all the geometries **Transparency** to 70. This will allow a visual of the particles flowing through the chute.

2.4 Inlet Creation

Create a surface by right clicking on the **Geometry** tab and select **Create Rectangular Inlet**. A surface will now have been created under the **Geometry** tab labeled **Rectangular 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.

Define the **Name**, **Centre Coordinates**, **Length** and **Width** of this new surface as the following. This surface will be defined as an inlet in a later step.

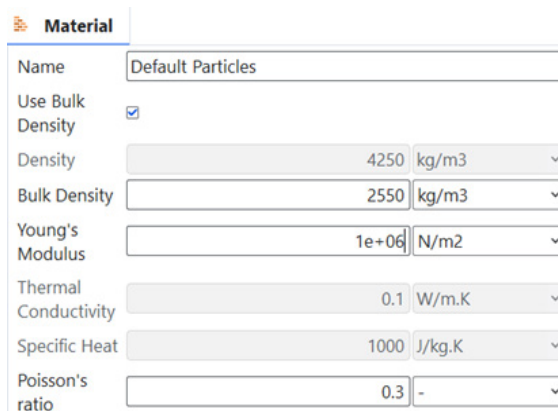


2.5 Particle Creation

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>**. Similar to the surface creation, define the **Name**, and under **Size**, define the Size as **50 mm** and under the **Movement** tab define the Rolling Resistance to **0.28**.

2.6 Particle Material Definition

There 3 types of materials in this simulation: mild steel chute walls, rubber conveyor belts and the particle material. Rocky generates a default material for each of these components. The walls and belts may be characterized by the default values for **Default Boundary** and **Default Belt** materials. The particles may be represented by the **Default Particles** material, however, the values for **Bulk Density** and **Young's Modulus** should be changed to the following.



The screenshot shows the 'Material' definition window for 'Default Particles'. The 'Name' field is 'Default Particles'. The 'Use Bulk Density' checkbox is checked. The 'Density' field is set to 4250 kg/m3. The 'Bulk Density' field is set to 2550 kg/m3. The 'Young's Modulus' field is set to 1e+06 N/m2. The 'Thermal Conductivity' field is set to 0.1 W/m.K. The 'Specific Heat' field is set to 1000 J/kg.K. The 'Poisson's ratio' field is set to 0.3.

2.7 Material Interaction Definition

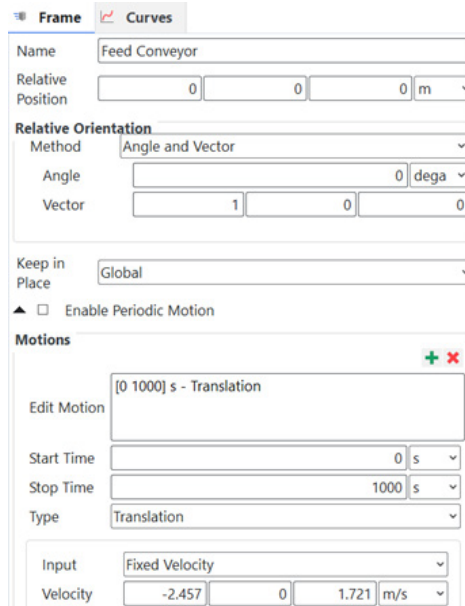
For this simulation we will be changing the material interaction definition between **Default Particle** with **Default Belt**, **Default Boundary** and **Default Particle**. To do this, select the **Material Interaction** tab and select **Default Particles** for the first drop down box and filter through each one for the second box defining the **Static Friction**, **Dynamic Friction** and **Restitution Coefficient** as follows, while leaving all other parameters as default.

	Default Belt			Default Boundary			Default Particle		
Default Particle	Static Friction	Dynamic Friction	Restitution Coefficient	Static Friction	Dynamic Friction	Restitution Coefficient	Static Friction	Dynamic Friction	Restitution Coefficient
	0.7	0.7	0.1	0.5	0.5	0.3	0.55	0.55	0.3

2.8 Motion Frames

To simulate the movement of the *feed conveyor* to allow the particles to enter the transfer chute, we will use a motion frame.

To do this navigate to the **Motion Frame** tab and then select **Create Motion Frame**. Select the newly created **Frame<01>** and adjust the parameters to the following. Ensure the **Keep in Place** option is set to **Global** as this will not allow the geometry to move out of place while the simulation is running, however, any particle in contact will act as if the boundary is moving.



Frame **Curves**

Name: Feed Conveyor

Relative Position: 0 0 0 m

Relative Orientation

Method: Angle and Vector

Angle: 0 deg

Vector: 1 0 0

Keep in Place: Global

☐ Enable Periodic Motion

Motions

Edit Motion: [0 1000] s - Translation

Start Time: 0 s

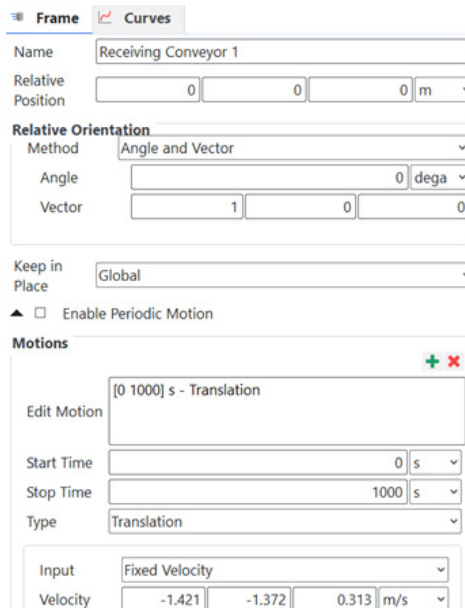
Stop Time: 1000 s

Type: Translation

Input: Fixed Velocity

Velocity: -2.457 0 1.721 m/s

Now, we need to create a motion frame for the *receiving conveyor 1* (*receiving conveyor 2* is not used in this simulation). Follow the same steps from the previous motion frame and set the parameters to the following.



Frame **Curves**

Name: Receiving Conveyor 1

Relative Position: 0 0 0 m

Relative Orientation

Method: Angle and Vector

Angle: 0 deg

Vector: 1 0 0

Keep in Place: Global

☐ Enable Periodic Motion

Motions

Edit Motion: [0 1000] s - Translation

Start Time: 0 s

Stop Time: 1000 s

Type: Translation

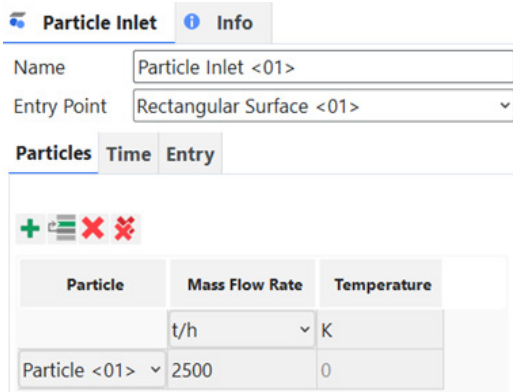
Input: Fixed Velocity

Velocity: -1.421 -1.372 0.313 m/s

To apply these motion frames and material definitions to the desired geometry, navigate to the **Geometry** tab. Under the **Wall** sub-tab for all three of the conveyor geometries, set the **Material** to **Default Belt** and leave all other geometries as the **Default Boundary**. Now set the corresponding motion frames defined earlier for the **feed conveyor** and **receiving conveyor 1** at the **Motion Frame** option.

2.9 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. We also need to define the inlet's **Name**, **Entry Point** and **Mass Flow Rate** to the following, leaving all other values as default.

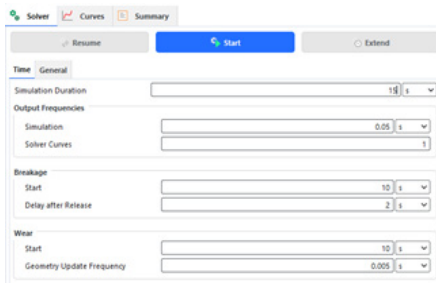


The screenshot shows the 'Particle Inlet' configuration window. The 'Name' field is set to 'Particle Inlet <01>' and the 'Entry Point' is set to 'Rectangular Surface <01>'. Below these are tabs for 'Particles', 'Time', and 'Entry'. The 'Particles' tab is active, showing a table with columns 'Particle', 'Mass Flow Rate', and 'Temperature'. The 'Particle' column has a dropdown menu with 'Particle <01>' selected. The 'Mass Flow Rate' column has a value of '2500' and a unit dropdown set to 't/h'. The 'Temperature' column has a value of '0' and a unit dropdown set to 'K'.

Particle	Mass Flow Rate	Temperature
Particle <01>	2500 t/h	0 K

2.10 Solve Simulation

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



The screenshot shows the 'Solver' configuration window. The 'Simulation Duration' is set to '18 s'. The 'Output Frequencies' section has 'Simulation' set to '0.05 s' and 'Solver Curves' set to '1'. The 'Breakage' section has 'Start' set to '10 s' and 'Delay after Release' set to '2 s'. The 'Wear' section has 'Start' set to '10 s' and 'Geometry Update Frequency' set to '0.005 s'.

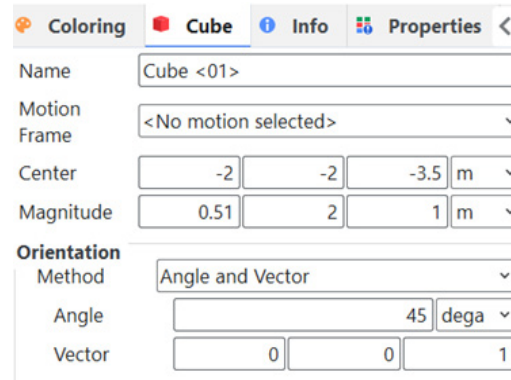
3. Analysis

3.1 Steady State Analysis

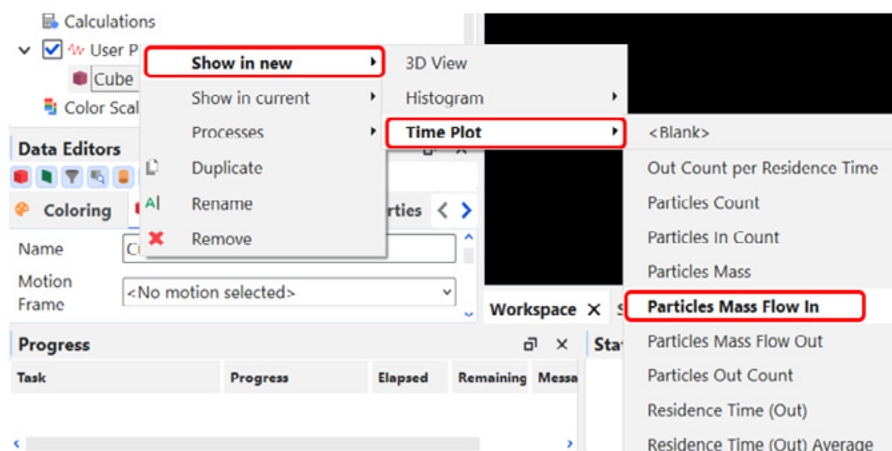
Before any analysis is undertaken, it is important to determine whether the system has reached a steady state. To do this, we will monitor the throughput at the end of the receiving conveyor belt.

Navigate to the **Particles** tab and select the small red **Cube** button, alternatively, you can right click on the Particles tab and select Processes and select Cube.

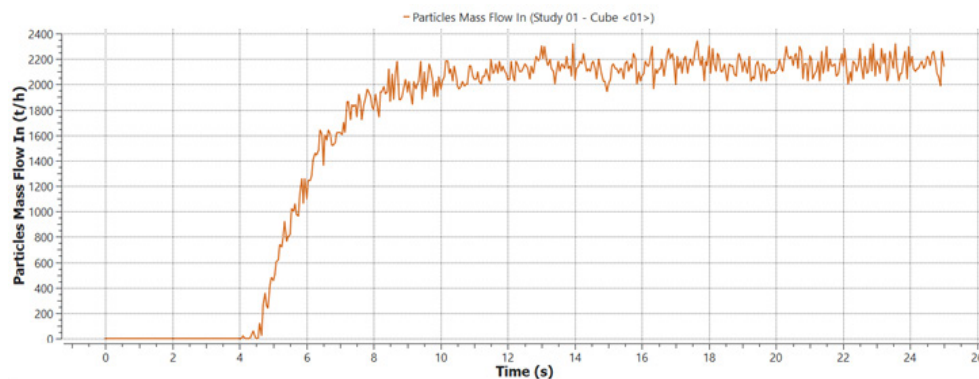
A **Cube<01>** entity will appear below the **User Processors** tab. Give the generated cube feature the following values.



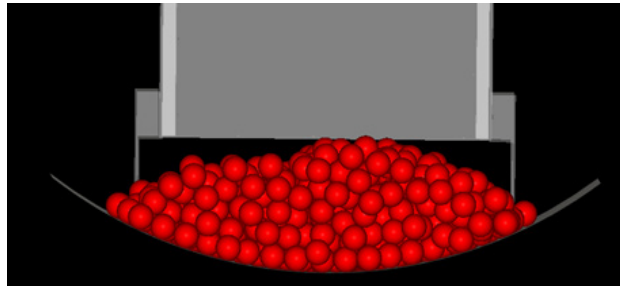
Right click on this entity and follow the below order to produce a plot displaying the mass of the particles entering the analysis box over time.



If the steps were followed correctly the plot should appear as the following. It can be determined from the plot that the simulation reaches steady state at approximately 12 seconds.



By hiding all other particles outside of the generated cube, we can analyze the cross section of the material and determine whether it is being loaded symmetrically. To do this de-select the eye icon to the right of the **Particles** tab. Ensure you are at the end of the simulation (or any time after 12 seconds) so there are particles to hide and have returned to the 3D viewer. Cubes established on each side of the belt may be used to evaluate central loading.

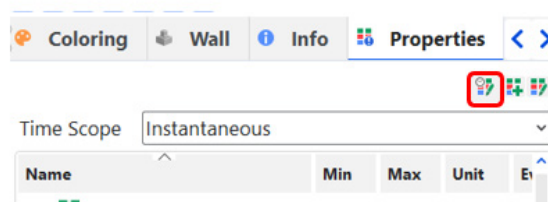


The particles are notably higher on the right side of the belt. Here we would expect to see a higher wear intensity at the right side of the belt where the chute feeds the particles. We will determine if this is the case in the following section.

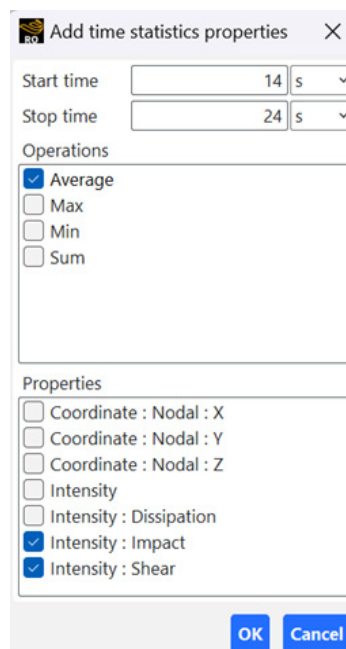
3.2 Wear Analysis

To assess the wear on the outgoing belt under steady state conditions, hide all geometries and particles excluding the *receiving conveyor 1*. To do so, de-select all the eye icons to the right of the geometries and the *Cube<01>* entity.

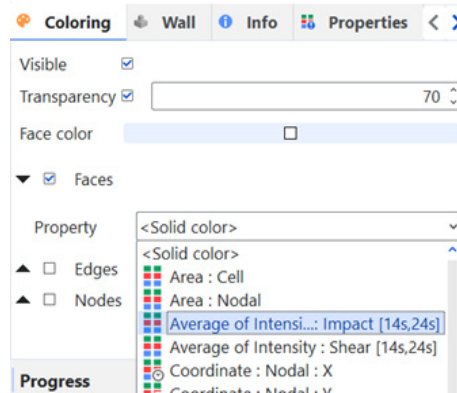
Navigate to the *receiving conveyor 1* tab and select the *Properties* sub-tab. Under this tab, select the small icon shown below and then click the green *Add* icon.



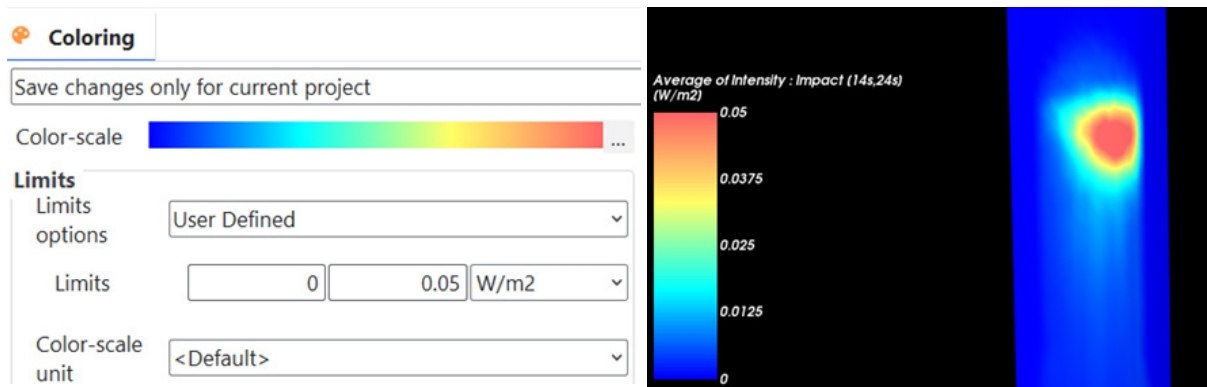
You will then be prompted with the following screen. Enter in the below values and select *OK*.



In the *receiving conveyor 1* **Coloring** sub-tab, select the Property you wish to view. The displays appears better if the transparency is switched off on the desired geometry.



The average impact and shear intensities are indicative of wear experienced by the surface of the belt. To achieve a better contrast of the intensity contour, adjust the **Limit options** under the **Color Scale** tab to *User Defined* and set appropriate upper and lower Limits.



Upon investigation, it is apparent our assumption of uneven loading was correct. It can be observed that due to this loading scenario we experience unevenly distributed wear patterns on the outgoing conveyor belt. You can do the same for the *Average of Intensity: Shear [14s,24s]* and should experience similar results.

4. Extension

4.1 Investigate the various display and analysis options.

As a further exercise, an investigation into the central loading on the belt, by mass. This can be achieved by creating cubes on either side of the receiving belt 1 and exporting the time plot data for each of the cubes.

Further analysis into the flow and wear profile of the chute geometry can be beneficial for future transfer chute design and optimization.

For extra help with any of these tasks, please refer to the Offline or Online User Manual located in the **Help** icon.

© 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.