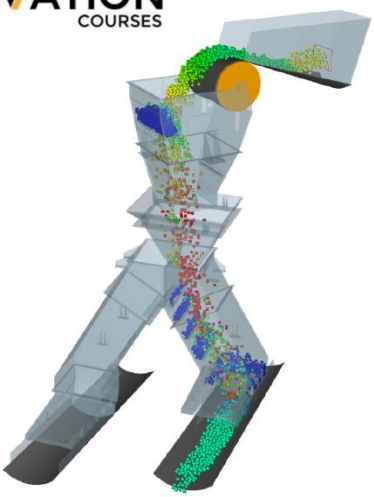
Ansys INNOVATION COURSES



Rocky Tutorial

Transfer Chute

Lesson 2 - Post-Processing

Ansys / ROCKY

Rocky Tutorial - Transfer Chute

1.1. Lesson 2 (Part B): Post-Processing

1.1.1. OBJECTIVES

The purpose of this tutorial is to introduce some basic methods for analyzing a simulation after youhave processed it. We will continue from where we left off in Part A.

You will learn how to:

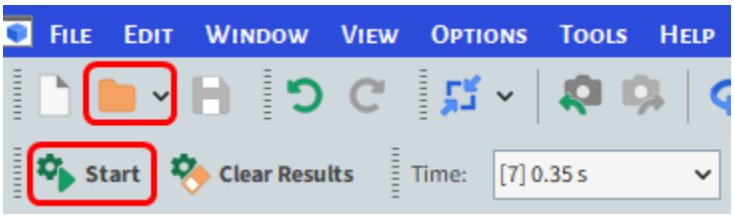
- Create an Animation
- Visualize Properties in a 3D View Window
- Create Graphs and Plots
- Filter Data with User Processes
- Export results

And you will use these features:

- Animation panel (videos)
- Time toolbar
- Multi Time plot
- Time plot
- User Process - Cube
- User Process - Plane

1.1.2. OPEN PROJECT

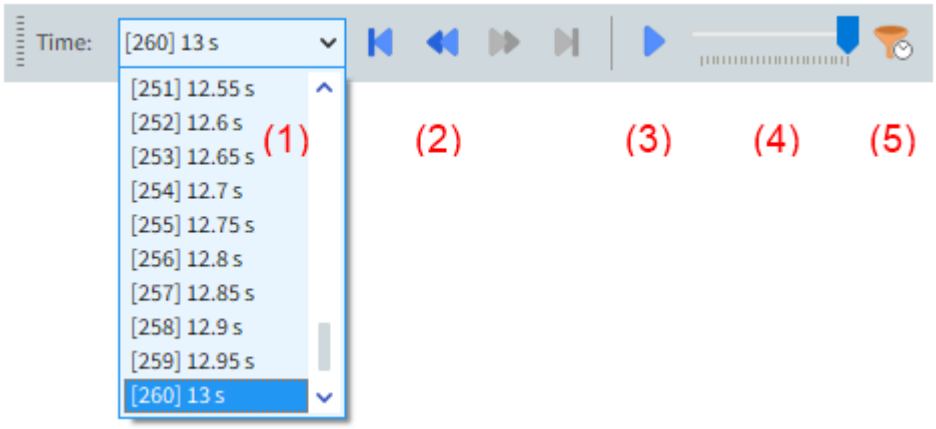
- If you completed **Part A** of this tutorial, ensure that Rocky project is open. (**Part B** will continue from where **Part A** left off.)
- Download the `dem_tut01_files.zip` file.
- Unzip `dem_tut01_files.zip` to your working directory.
- Open **Rocky 2024 R1**. (Look for Rocky 2024 R1 in the Program Menu or use the desktop shortcut.)
- From the Rocky program, click the **Open Project** button, find the **tutorial_01_input_files** folder, and then from the **tutorial_01_A_pre-processing** folder, open the **tutorial_01_pre-processing.rocky** file.



- Process the simulation. (From the **Simulation** toolbar, click the **Start** button.)

1.1.3. MOVING TROUGH OUTPUTS

Now that the project has completed processing, we can begin to analyze it. For example, you canuse the **Time** toolbar in the following ways:



(1) Select a specific output/time from the drop-down list.

(2) Use the arrow buttons (from left-to-right) to:

- Go to first output
- Step back one output
- Step forward one output

- Go to last output

(3) Play the animation.

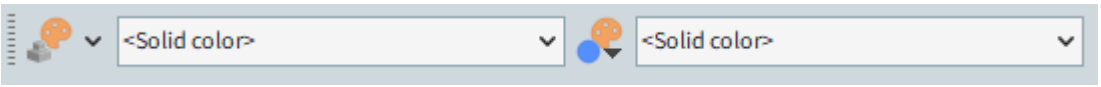
(4) Slide to the output you want using the slider bar.

(5) Use the Timeset Filter to display only a specified time range.

1.1.4. COLORING BOUNDARY/PARTICLES

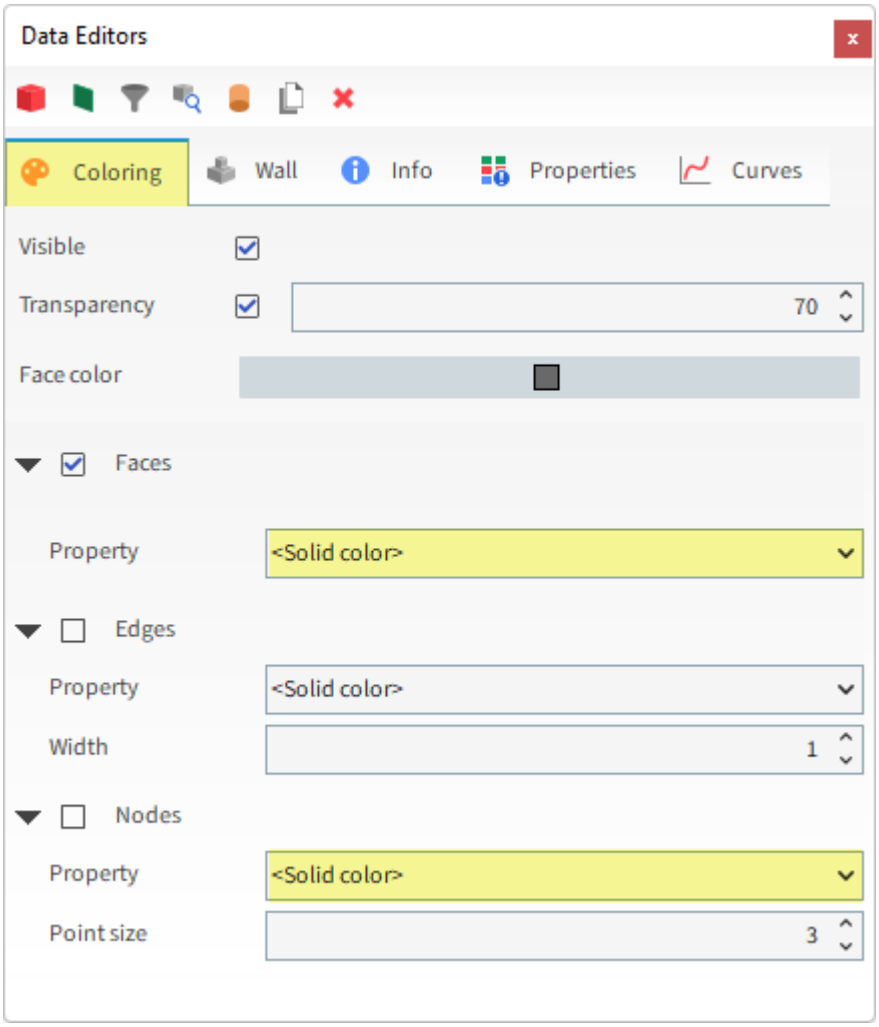
There are 3 different ways to color the boundaries and/or the particles:

Use the **Coloring service** toolbar to color all the boundaries/particles by a property:



Use the **Coloring** tab by doing the following:

- From the **Data** panel, select either a boundary under **Geometries** or the main **Particles** entity.
- From the **Data Editors** panel, select the **Coloring** tab, expand **Faces** (for boundaries) or **Nodes** (for particles) and then select the desired property to color. This way, only the selected item will be colored (not all of them as with the other options).

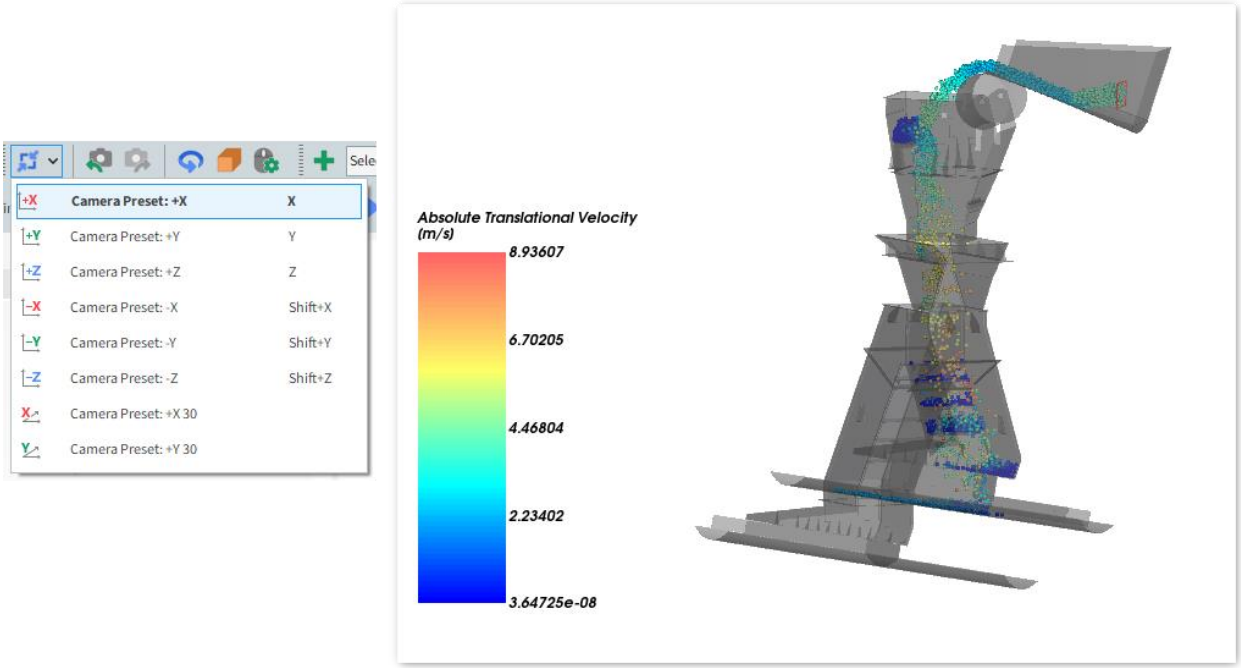


Use the **Properties** tab by dragging and dropping the desired property over a **3D View** window.For this tutorial, we want to color our particles by velocity.

1. From the **Data** panel**,** select **Particles** and then from the **Data Editors** panel, select the **Properties** tab.

2. Select **Absolute Translational Velocity** and then drag and drop it onto the **3D View** window.

3. You can then use your mouse to zoom and pan, and use your mouse or the options on the **Fit** toolbar (as shown) to change the orientation.

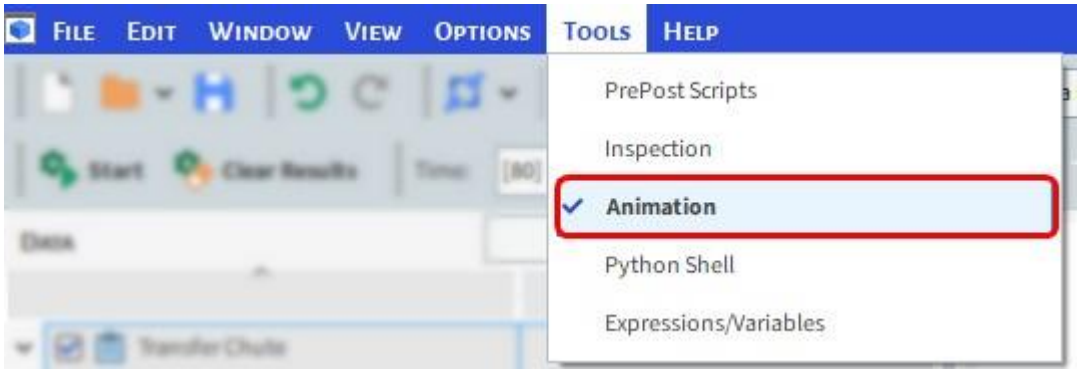


1.1.5. ANIMATION

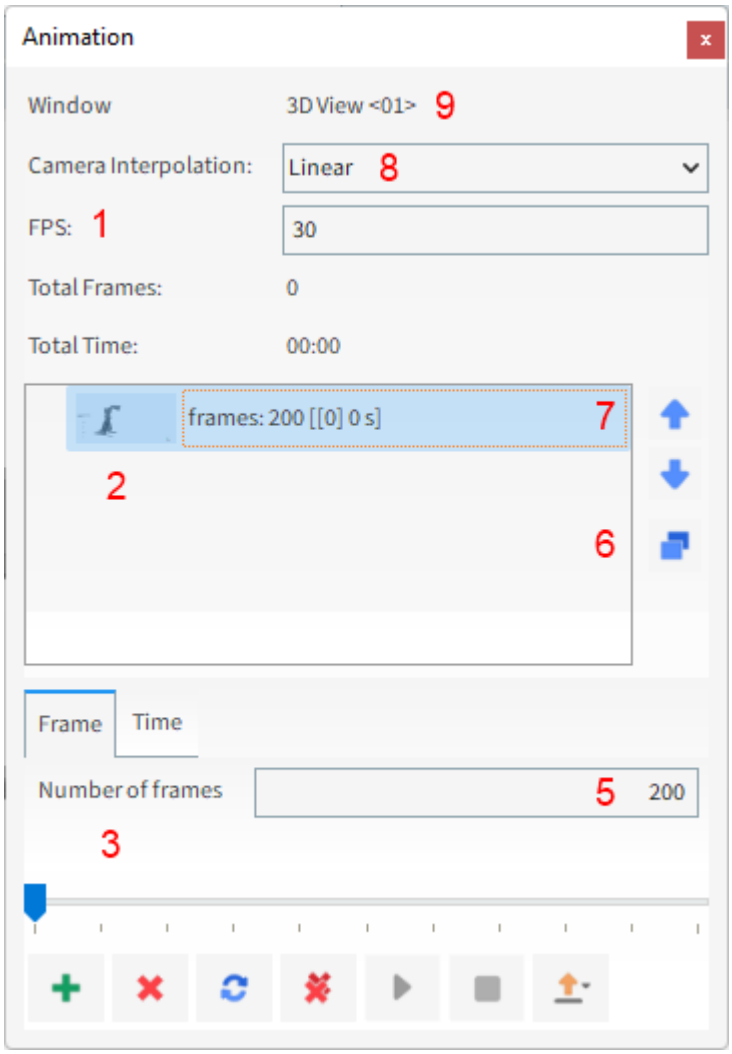
To create an animation (video) in Rocky, you set key frames of a particular **3D View** window at specifiedoutputs.

Rocky will interpolate between the created key frames using the available outputs saved during the simulation.

1. To show the **Animation** panel, from the **Tools** menu, select **Animation.**



1.1.6. ANIMATION PANEL



(1) Frames per Second **(FPS)** will change the playback speed of the animation. At least 30 **FPS** is recommended. To create a smooth animation, the O**utput Frequency** should not be greater than **1/FPS**.

(2) Key Frames list.

(3) Select a specific moment in the animation.

(4) Add Key Frame / Remove Key Frame / Update Current Key Frame / Remove All Key Frames / Play / Stop / Export (video or images).

(5) Number of frames between the selected Key Frame and the next one. The Number of Frames divided by the FPS gives the real animation time. This value can be changed to display the animation in real time.

(6) Duplicate the selected Key Frame.

(7) Move the selected Key Frame Up or Down to change the order.

(8) Camera Interpolation method.

(9) Name of 3D View window that is currently selected.

1.1.7. CREATING AN ANIMATION

For this tutorial, a simple animation using only 2 Key Frames in real time will be created (13s).

1. Since we use an **Output Frequency** of 0.05 s, we should use an **FPS** of 20 or less (FPS should be less or equal 1/Output Frequency). Use **FPS** equal to 20.

2. Select the 3D View you set up earlier. Then, using the **Time** toolbar, change the output to **0 s**.

3. Add the first Key Frame by clicking the **Add Key Frame** (green plus) button.

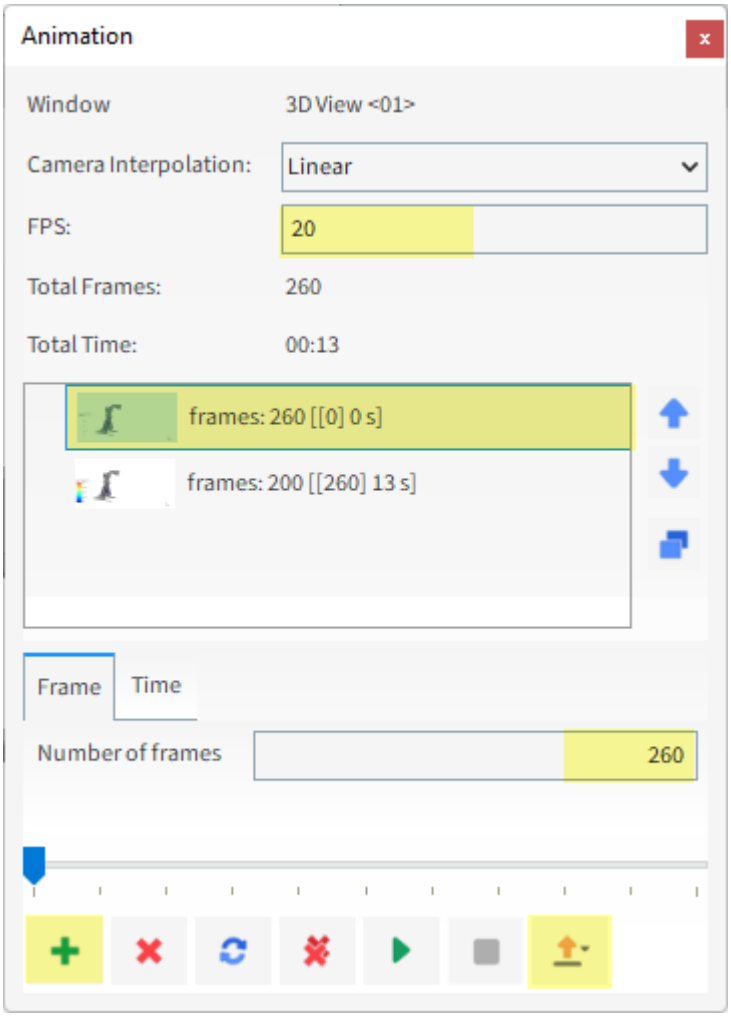
4. Select the new frame and then from the **Frame** tab, change the **Number of Frames** to 260 (as shown). Since there are 260 output files in this simulation, and our FPS is **20**, this will give us the full 13 seconds between our first and second frames. **(260 / 20 = 13)**

5. Use the **Time** toolbar to change the time to the last output, and add a second Key Frame.

6. Your **Total Time** should be 00:13 (real time).

7. Click **Play** to preview the movie in the 3D View window.

8. Click **Export Animation** to save the movie to an AVI file.



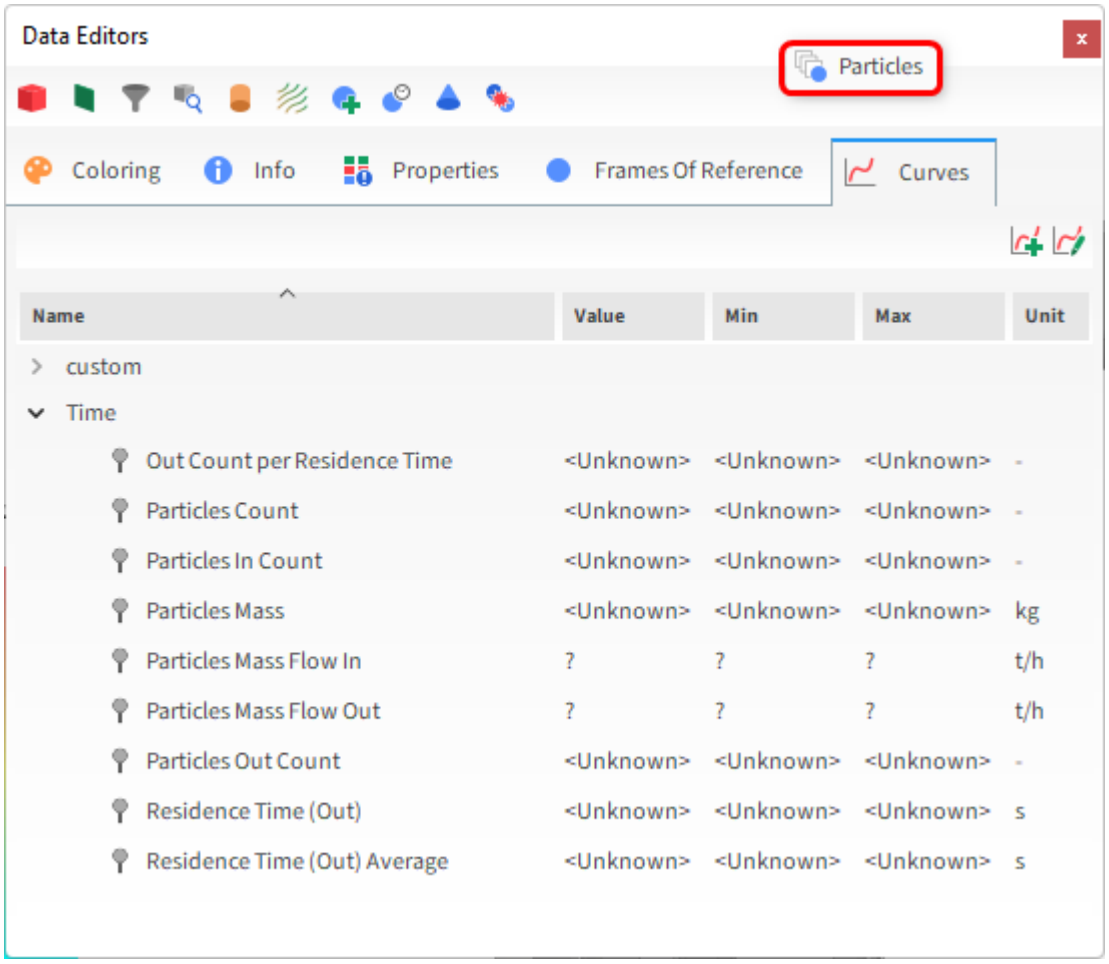
1.1.8. TIME-DEPENDENT PLOTS

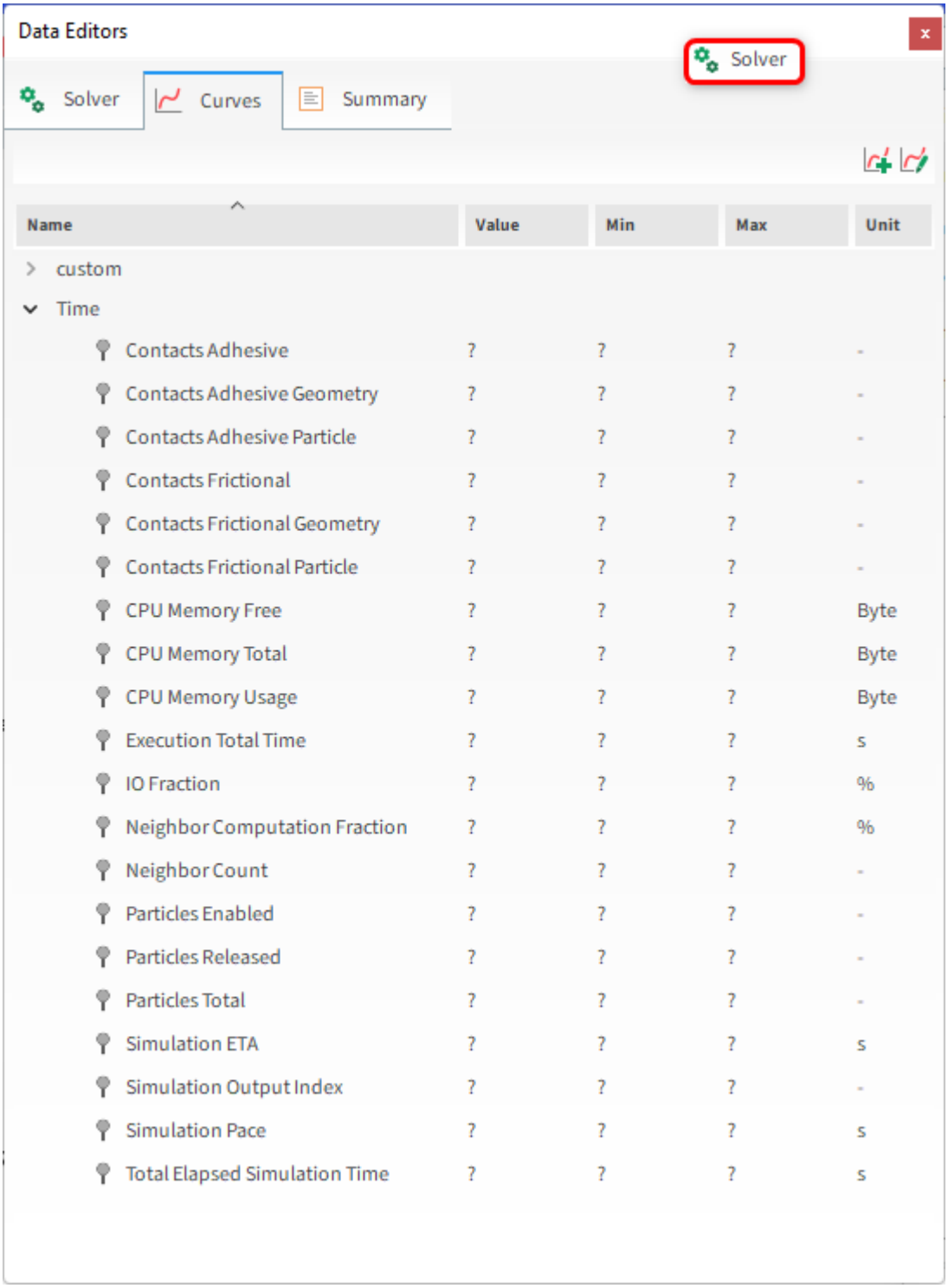
All the **Properties** are calculated for every timestep and every Triangle (boundary mesh) or Particle.

In order to create a **Time Plot** or a **Multi Time Plot**, you must select one of the following operationsto transform the **Properties** into a single time-dependent curve:

- **Minimum:** Lowest value among all particles/triangles
- **Maximum:** Highest value among all particles/triangles
- **Average:** Mean value among all particles/triangles
- **Sum:** Sum of all values among all particles/triangles
- **Sum Squared:** Sum of the squared values among all particles/triangles
- **Variance:** Squared deviation of a value from its mean
- **Standard Deviation:** Squared root of the variance

Particles and **Solver** each contain a **Curves** tab, which includes several pre-defined curves that canbe plotted without applying any additional operations.





1.1.9. MULTI TIME PLOT

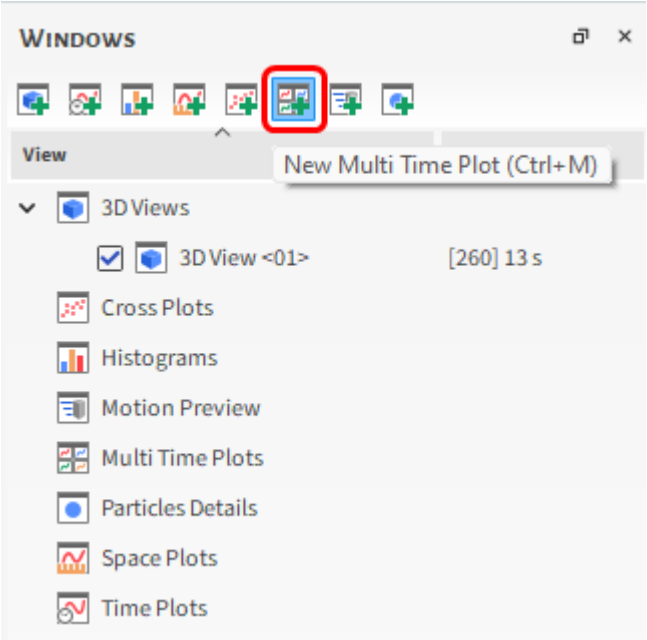
The **Multi Time Plot** is a useful tool to compare different curves at the same time, but are plottedeither on the same grid, or on a separate one (subplot).

In this tutorial we will compare the amount of particles that entered the domain **(Particles In Count)**,left the domain **(Particles Out Count)**, and the total amount of particles inside the domain **(Particle Count)**

at each output.

To create a **Multi Time Plot**, do the following:

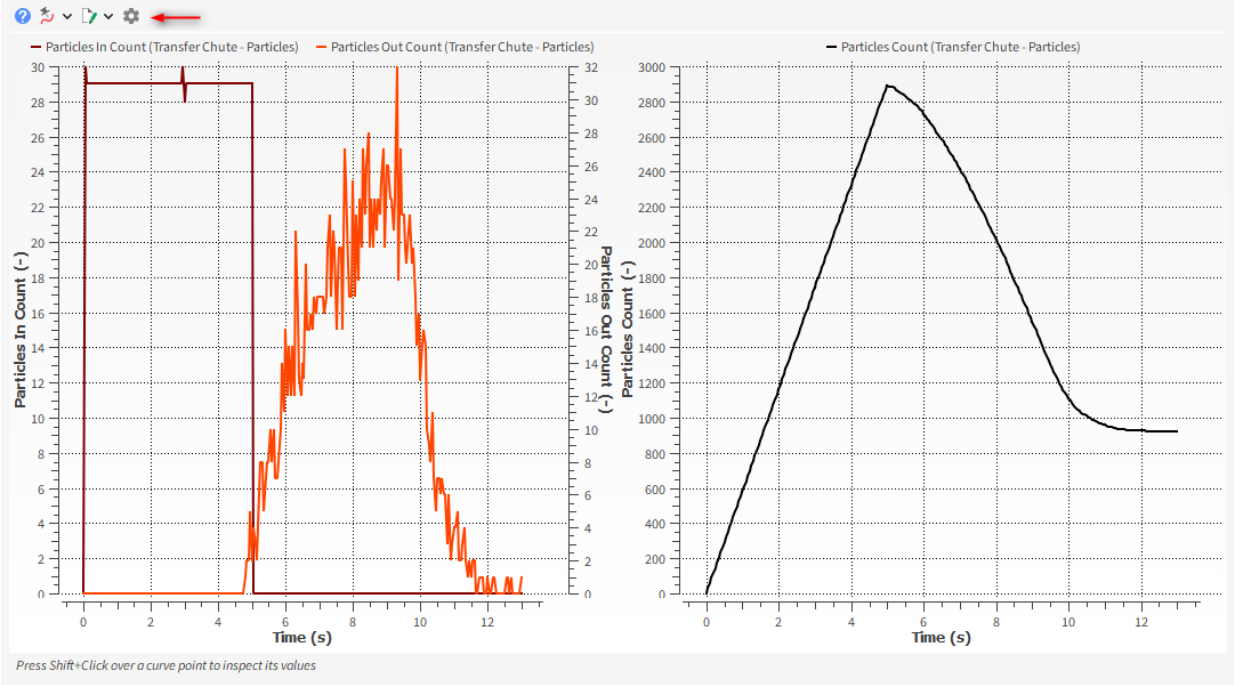
1. From the **Windows** panel (if not visible, point to **View**, and click on **Windows**), select **New Multi Time Plot**, or use the shortcut **Ctrl+M** (as shown).



2. From the **Data** panel, select **Particles** and then from the **Data Editors** panel, select the **Curves** tab.

3. From the **Curves** tab, drag and drop **Particles In Count** over the plot window. Repeat the same procedure for **Particles Out Count.**

4. To plot the total number of particles in a separate subplot, click and hold **Particles Count,** and then with the **Ctrl** key pressed, drag and drop the curve over the plot.



5. In the top left corner of the plot, you can select **Configure Window** to edit text display, colors, axes limits, units and other related options.

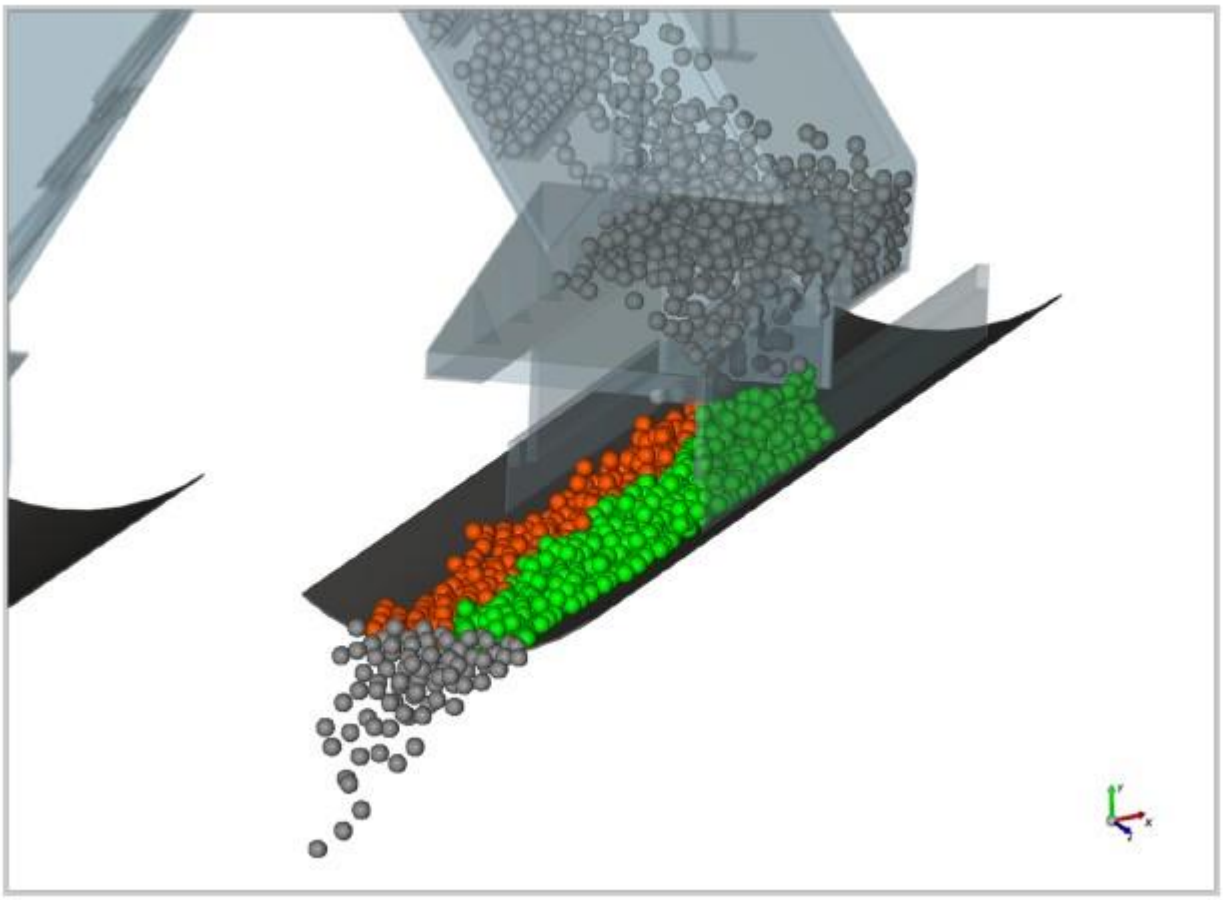
1.1.10. USER PROCESSES

For some DEM analyses the data must be restricted to a particular region, or a particular subset ofmaterial.

Rocky **User Processes** are used to divide and analyze **Particles, Geometries,** and **Fluids** and includethe following types:

- **Cube:** Create a subset of data based upon a box region.
- **Cylinder:** Create a subset of data based upon a cylinder region.
- **Plane:** Create a subset of data based upon a plane.
- **Polyhedron (Envelope):** Create a subset of data based upon a custom shape region that you import via .stl file.
- **Property:** Create a subset of particles/geometry based upon a particular property value or range.
- **Cell Inspector:** Select a single, individual particle or triangle (boundary).
- **Particles Trajectory:** Create the particles` path lines for a specified time range.
- **Particle Time Selection:** Create a subset of particles based upon a time filter.
- **Eulerian Statistics:** Transform the discrete properties into continuous values by averaging the values over discretized regions.

To illustrate the use of these tools, a **Cube** and a **Plane User Process** will be used to analyze themass unbalance on the receiving conveyor.

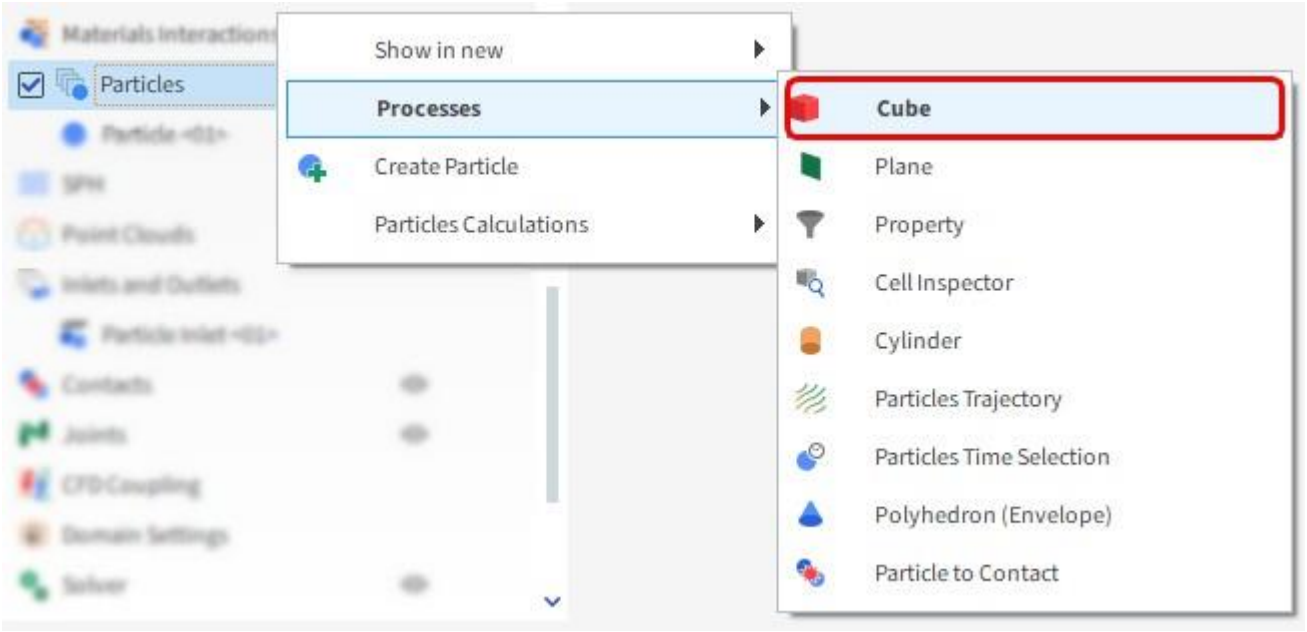


One **Cube** and two **Planes** will be used: the **Cube** to isolate the receiving conveyor and the **Planes** to divide those particles into two subsets: left (orange) and right (green).

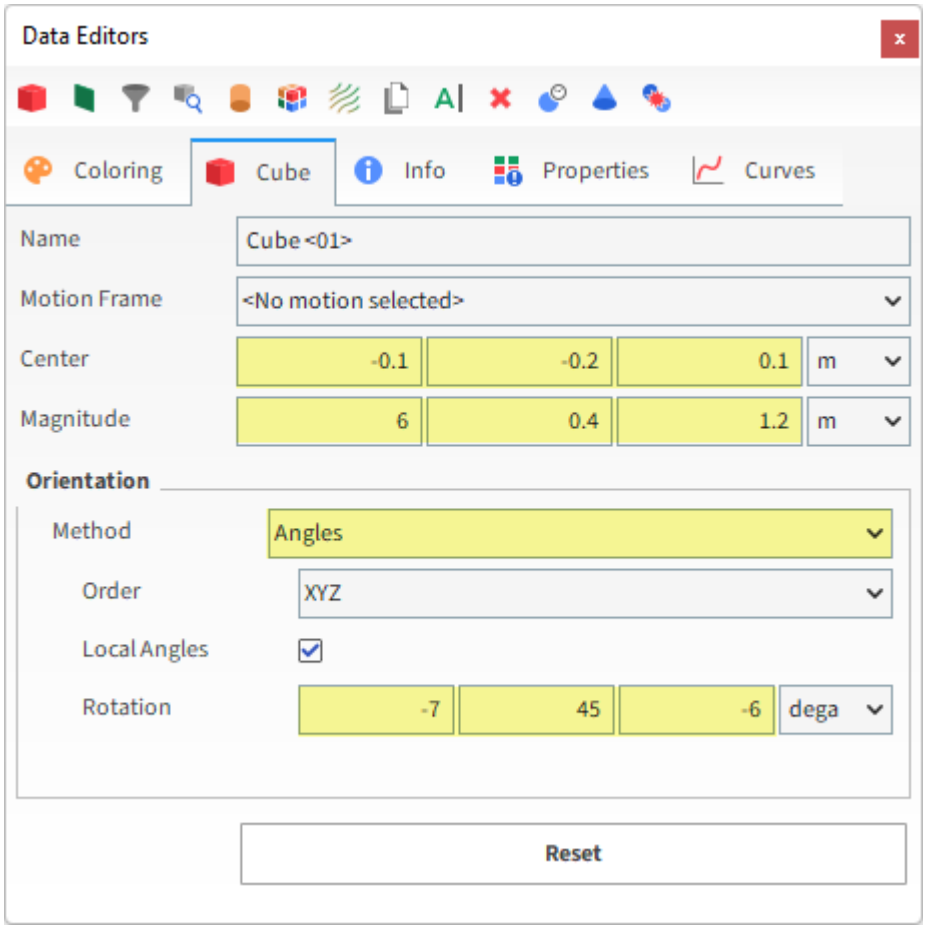
1.1.11. USER PROCESSES - CUBE

The first **User Process** will be the **Cube**. To create it, do the following:

1. From the **Data** panel, right-click **Particles**, point to **Processes**, and then select **Cube**.

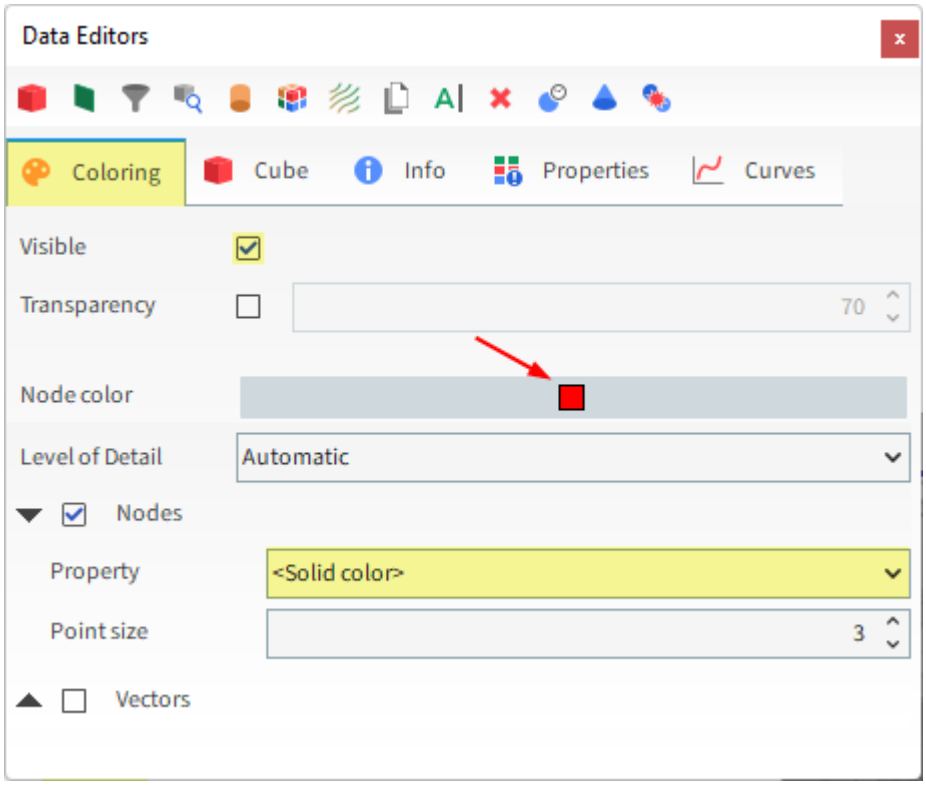


2. From the **Data Editors** panel, in the **Cube** tab, use the values shown in the image for **Center, Magnitude, Method,** and **Rotation**.

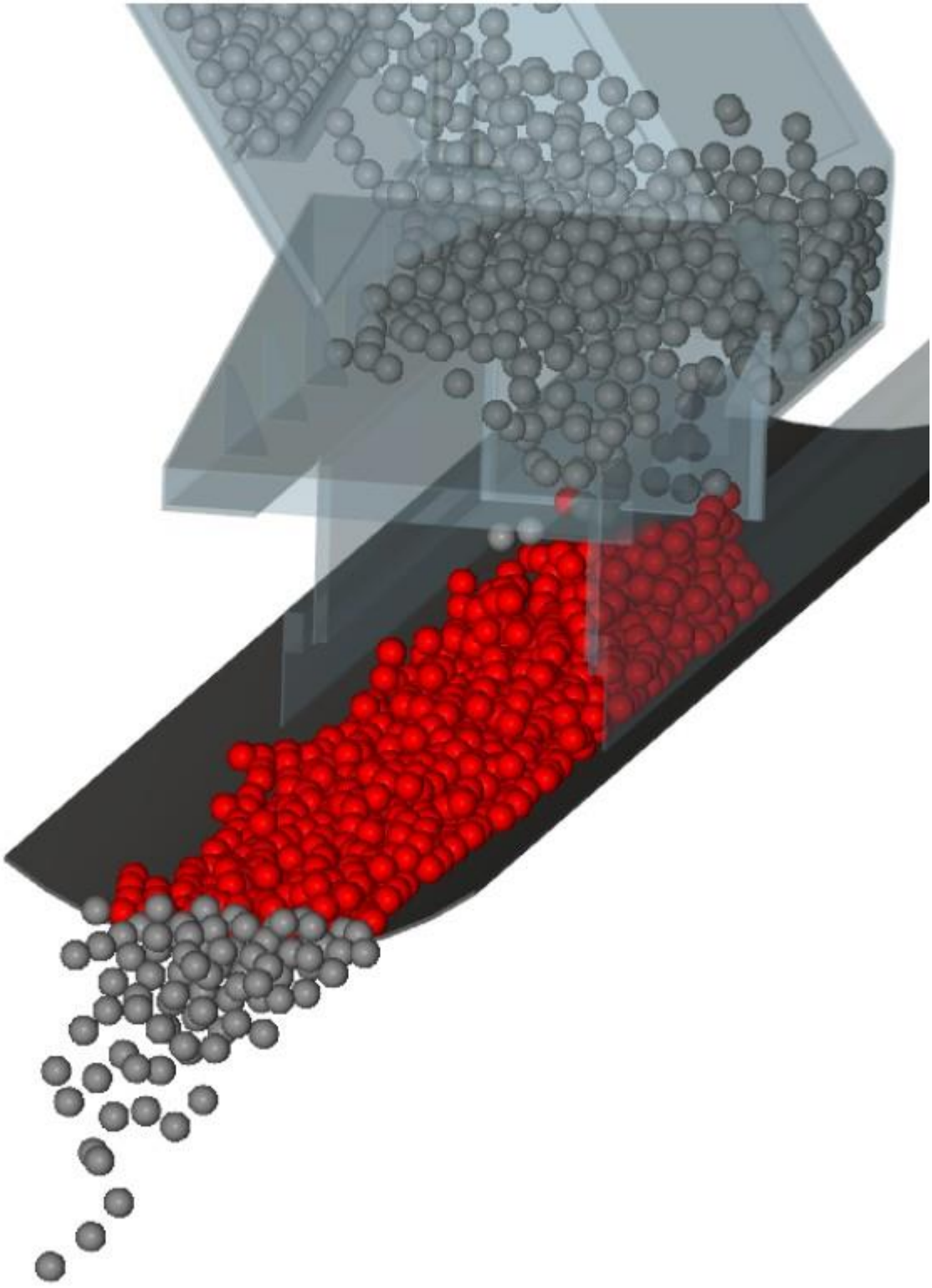


3. From the **Windows** panel, select the **3D View <01>** window.

4. From the **Coloring** tab, select **Solid Color** as **Nodes | Property**, and ensure the **Node color** is setto red and the **Visible** checkbox is selected.



Note: ***User Processes*** *can be manually changed using the* ***3D View****, or adjusted using the parameters displayed in the* ***Data Editors*** *panel. For this tutorial, we defined exact parameters in the* ***Data Editors*** *panel.*



Once the cube has been created, the subset of particles inside this region can be used to createspecific plots, new properties and also to create new **subsets** derived from it.

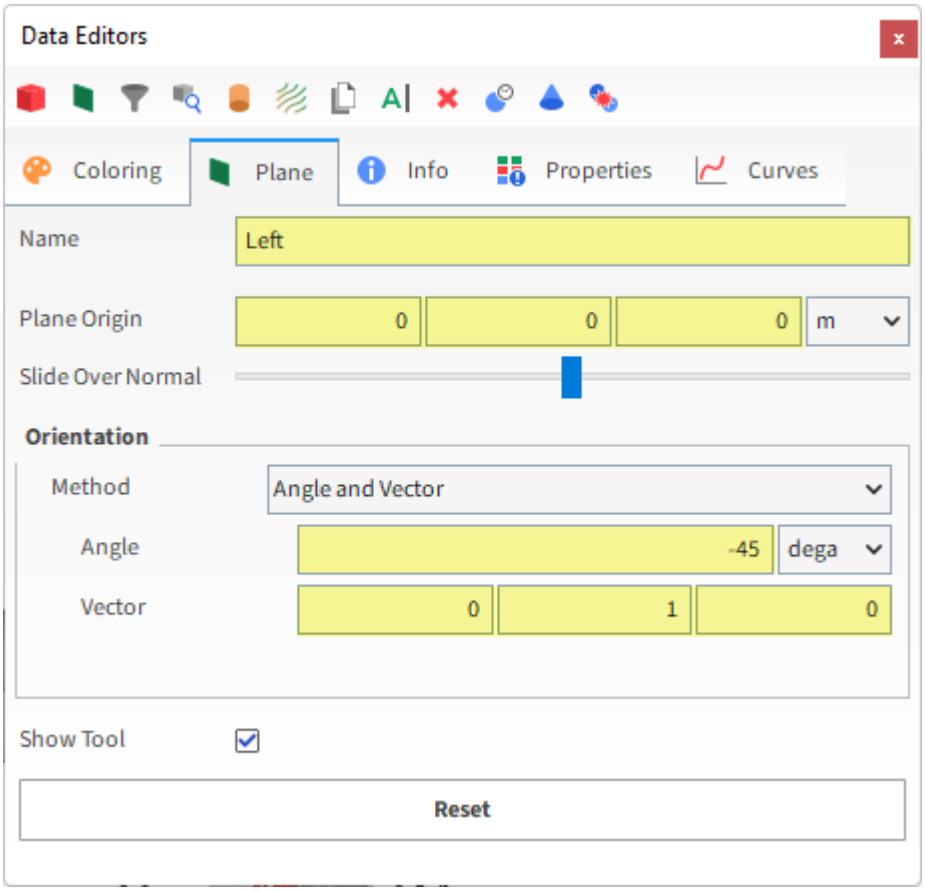
In this tutorial, we want to divide only the particles on the conveyor (envolved by Cube <01>), intotwo sets: left and right. In order to do that, two **Planes** will be created based upon the **Cube** sub- selection of **Particles**.

1.1.12. USER PROCESSES - PLANE

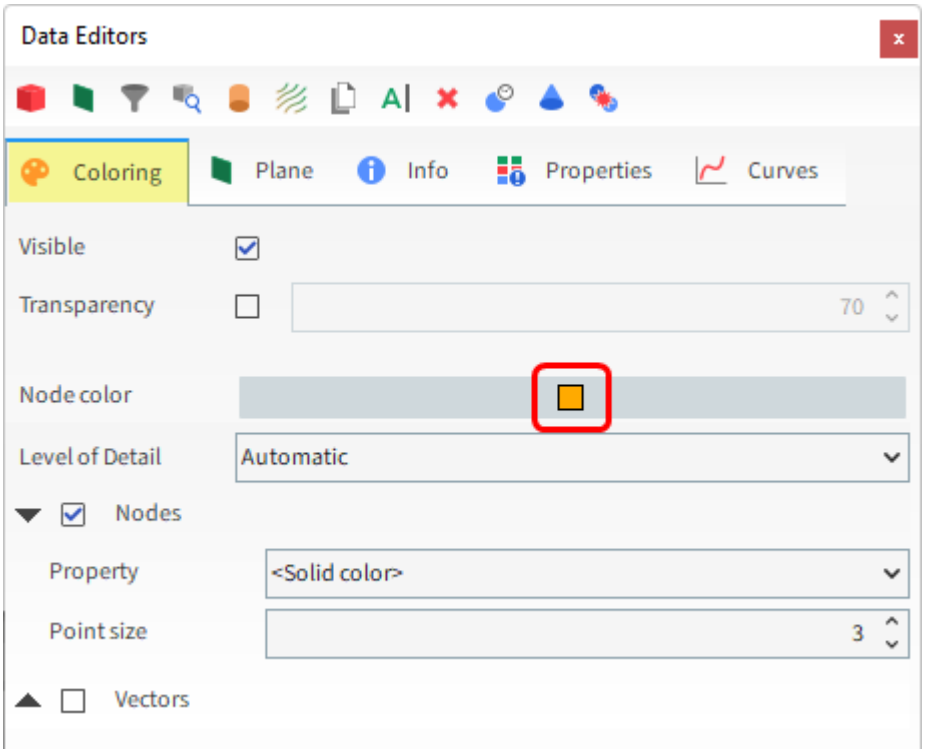
To create the first plane, do the following:

1. Right-click **Cube <01>**, point to **Processes**, and then select **Plane**.

2. From the **Data Editors** panel, select the **Plane** tab and then define the **Name, Plane Origin** and **Orientation** values (as shown).

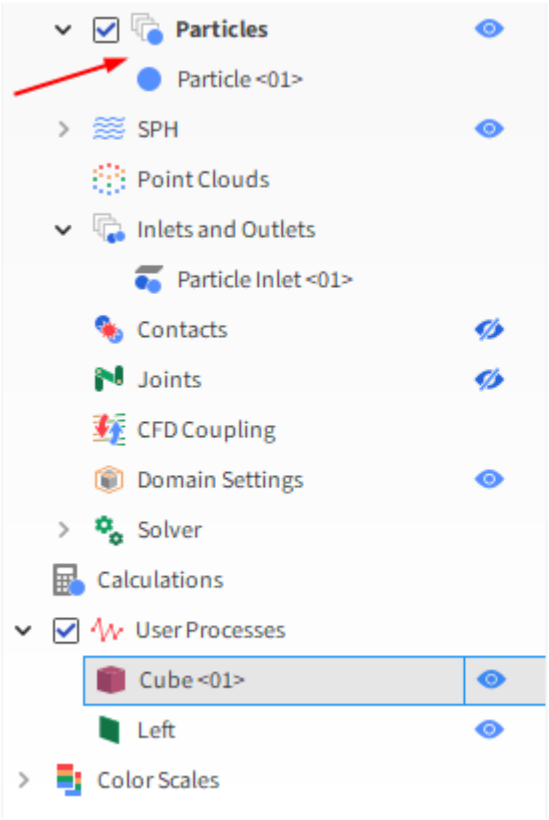


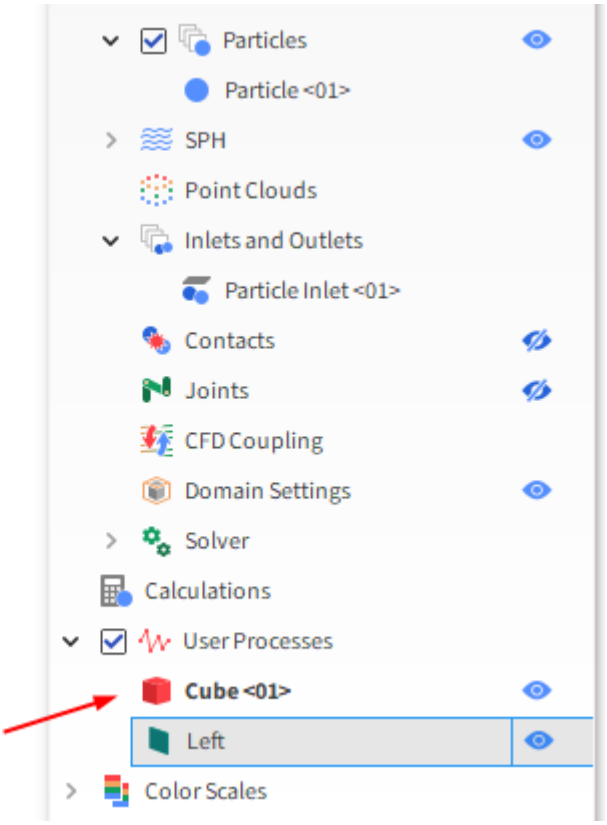
3. From the **Coloring** tab, set also the **Node color** to orange.



When a User Process is selected in the **Data** panel, Rocky highlights the association between it andother User Processes by displaying the parent User Process name in **Bold.**

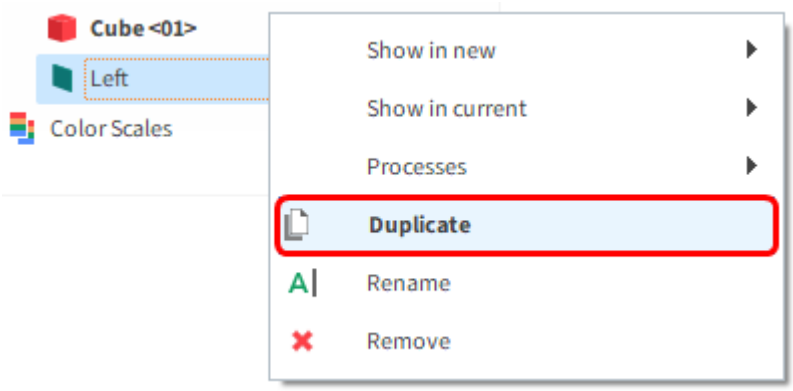
For example, when you select **Cube <01>**, **Particles** will be displayed in **bold** letters. And when youselect the **Left** plane, **Cube <01>** will be **bold.**



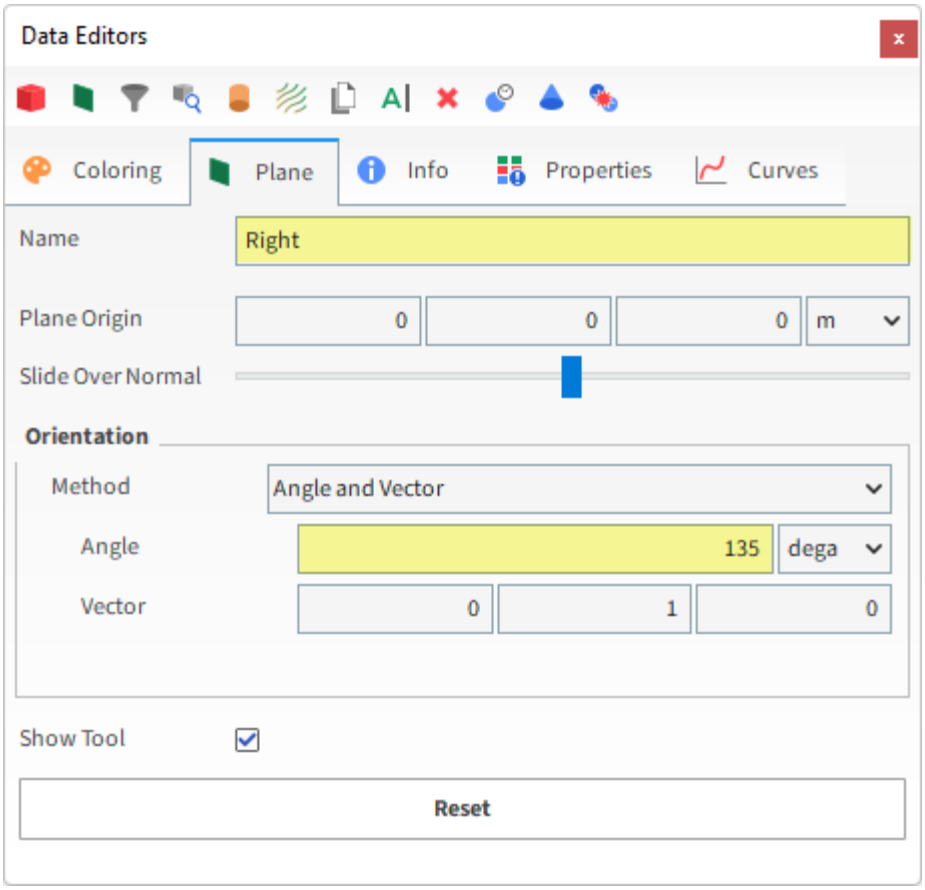


The second **Plane** is exactly the opposite of the previous, so we will create a copy of it:

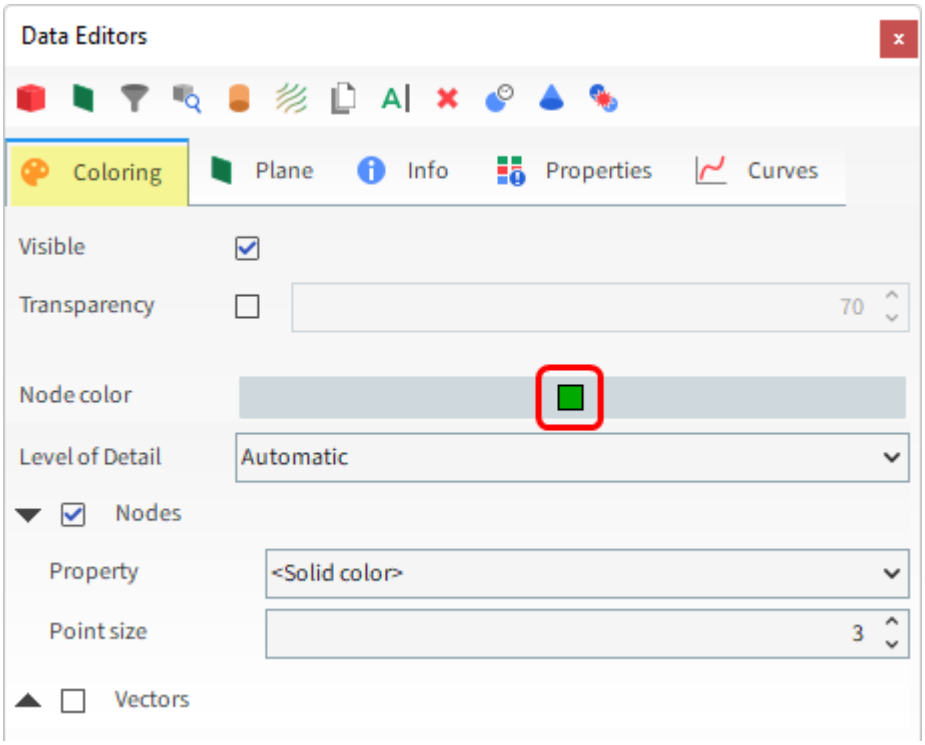
1. From the **Data panel**, right-click **Left** and then select **Duplicate.**



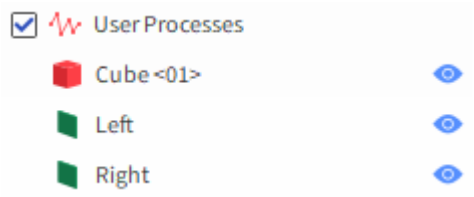
2. From the **Data Editors** panel, select the **Plane** tab and then modify the **Name** and **Plane Orientation | Angle** value.



3. From the **Coloring** tab, set also the **Node color** to green.

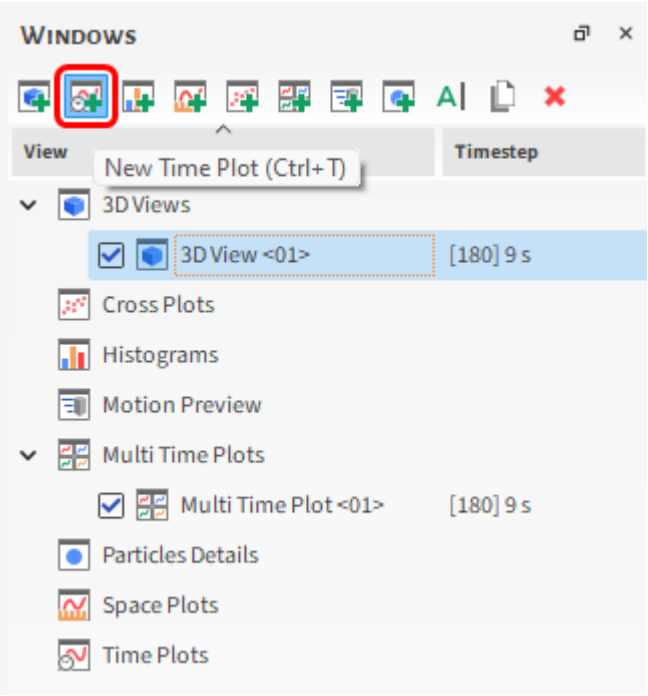


Tip: *You can visualize the planes in the 3D View window by ensuring that the eye icons for Left and Rightare turned on.*



The next step is to create a **Time Plot** comparing the unbalance between both sides of the conveyor.

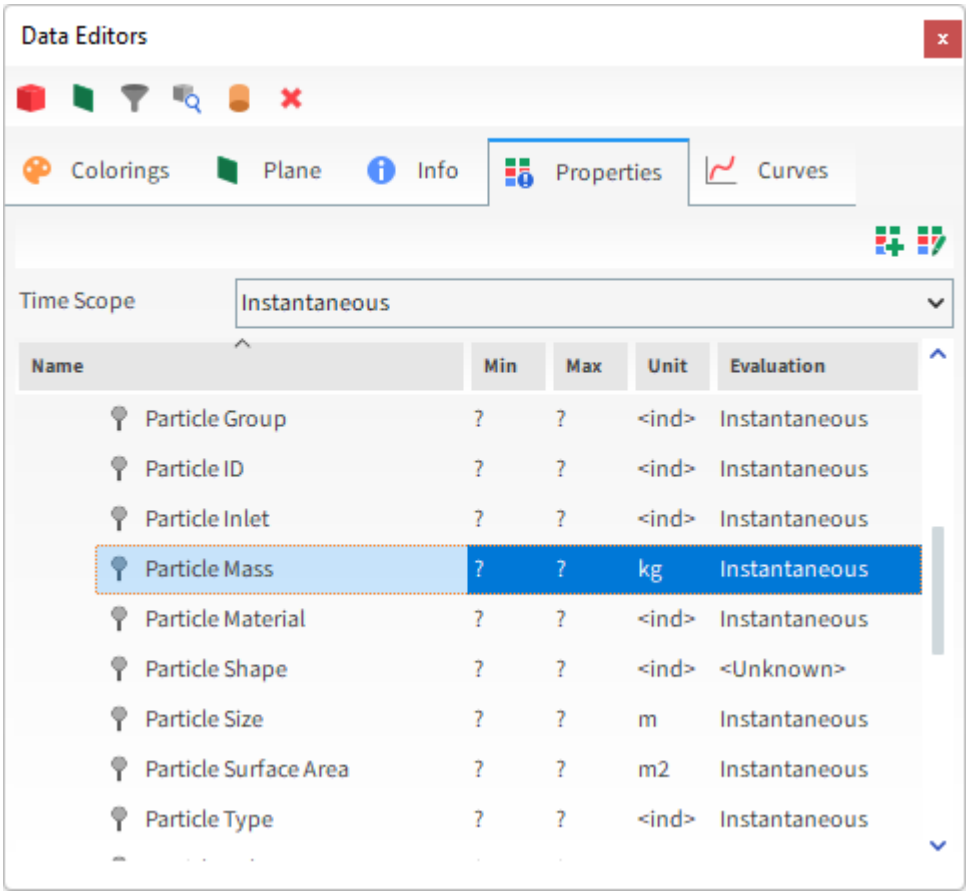
1. Similar to the **Multi Time Plot,** create a **Time Plot** by selecting **New Time Plot** from the **Windows** panel, or by using the shortcut **Ctrl+T.**



1.1.13. TIME PLOT

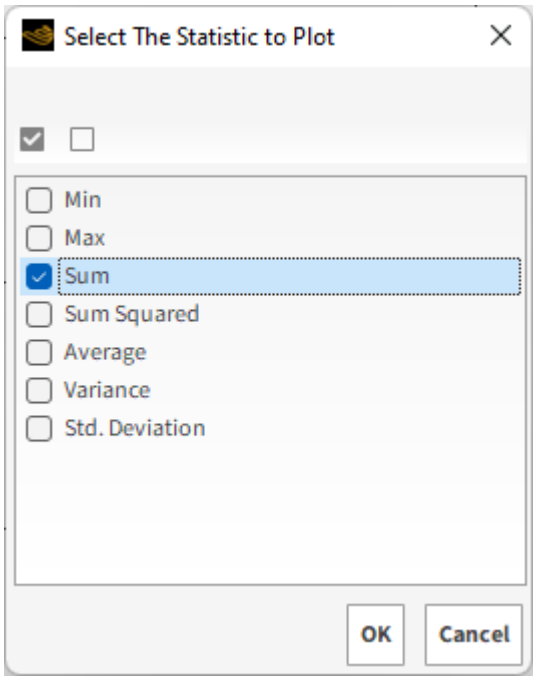
2. From the **Data** panel, under **User Processes**, multi-select both the **Left** and **Right** planes.

3. From the **Data Editors** panel, select the **Properties** tab, and then drag and drop **Particle Mass** over the plot.



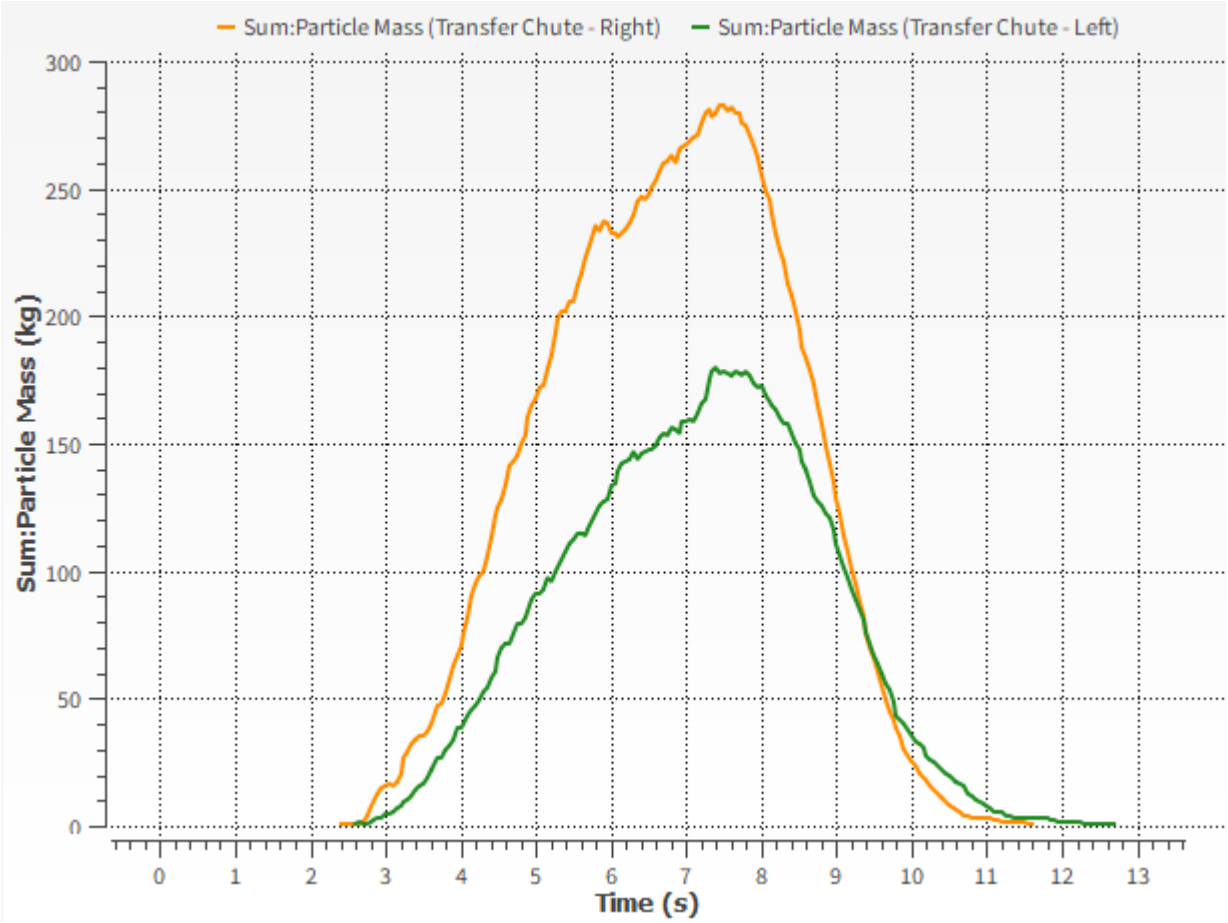
Note: *Properties will either be* ***Instantaneous*** *or will have resulted from a* ***Statistical*** *analysis. This categorization will be shown in the* ***Evaluation*** *column.*

4. A new dialog will be displayed asking which operation you want to apply to the properties to turnit into a curve. Select only **Sum**, and then click **OK**.


Select The Statistic to Plot
Min
Max
Sum
Sum Squared
Average
Variance
Std. Deviation
OK
Cancel

1.1.14. POST PROCESSING - USER PROCESSES

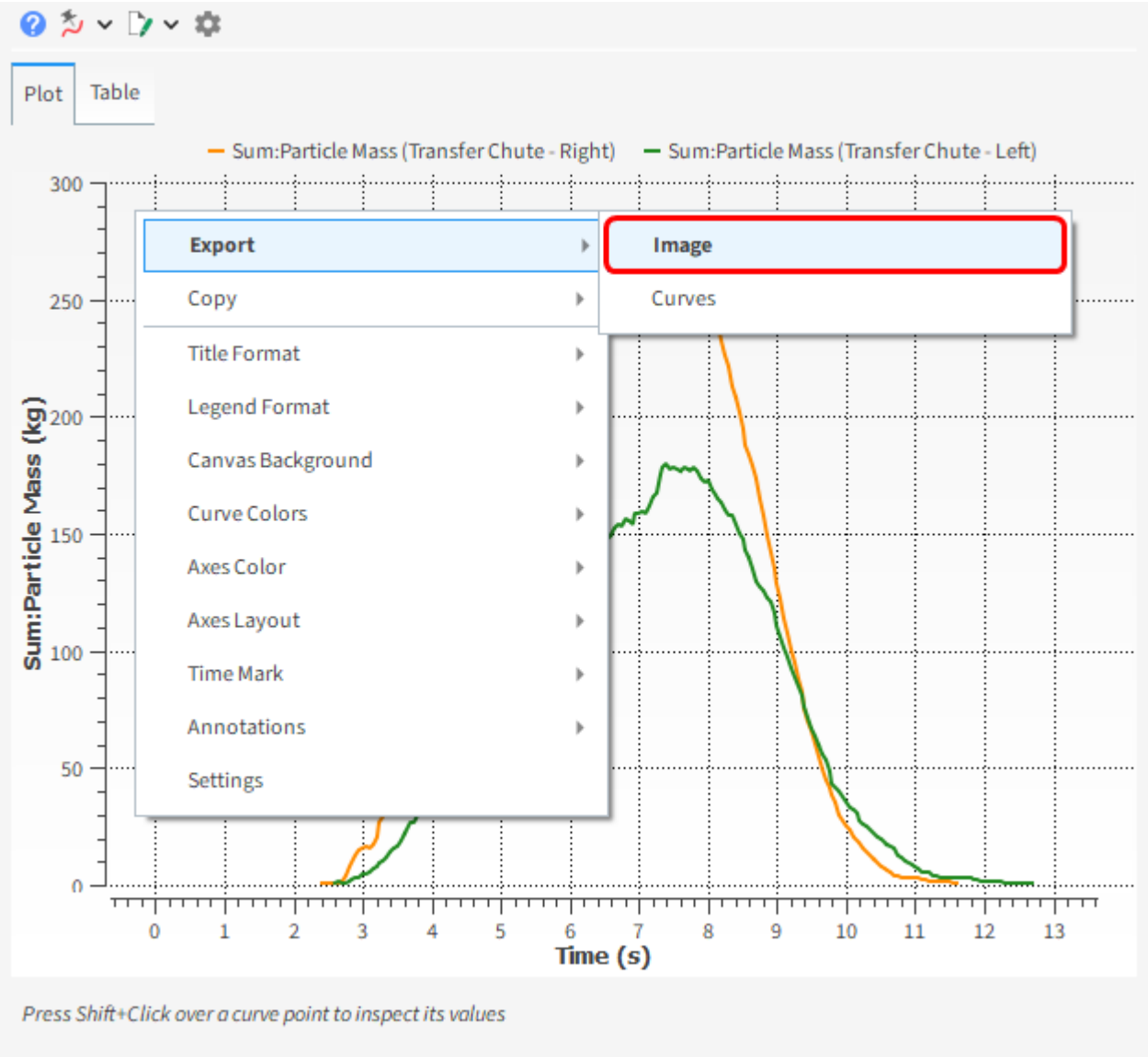
The **Time Plot** that appears shows that there is a balance difference between the two sides of the conveyor, which can cause operational problems and lead to additional wear on the belt surface.



1.1.15. POST PROCESSING - EXPORT IMAGES

It is possible to **Export** Rocky plots and **Save** 3D views as images.

1. To export an image of the plot, right-click an empty area within it, point to **Export**, and then click **Image**.



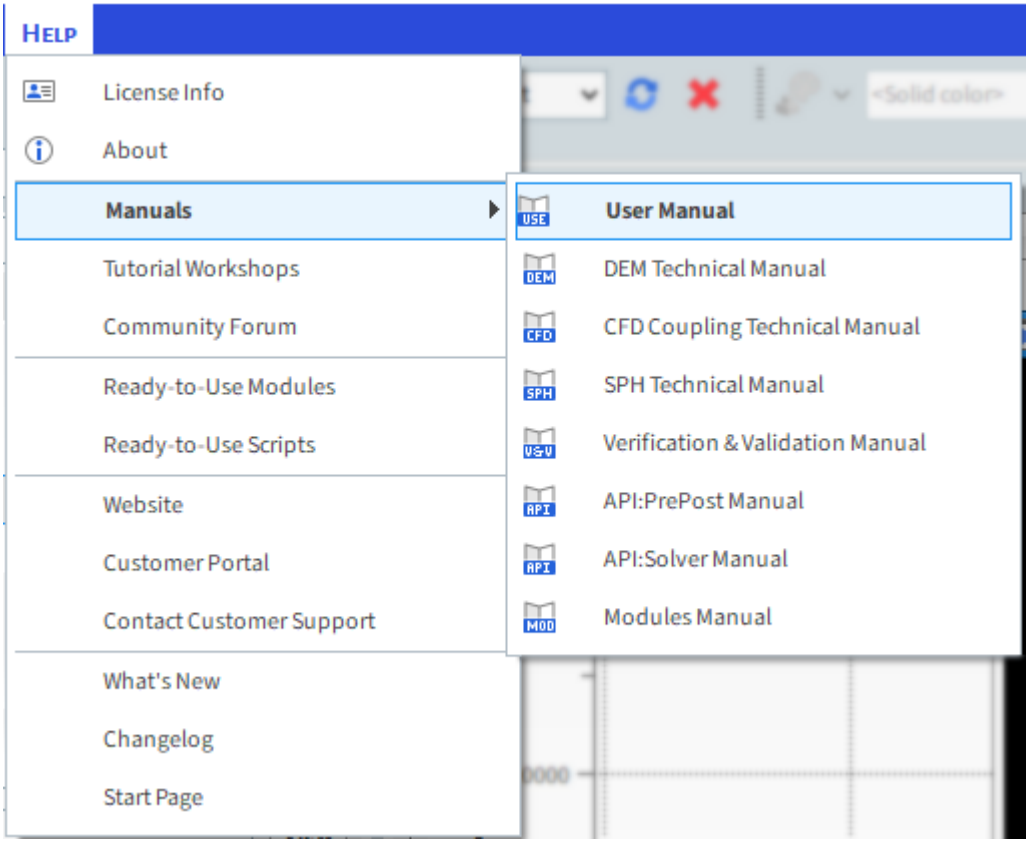
2. From the **Image** dialog, choose the **Snapshot Size,** and then click **OK**.
3. From the **Snapshot** dialog, set a **File name, Save as type** image extension, and location for your file, and then click **Save**.

1.1.16. HELP - USER MANUAL

This completes **Part B** of this tutorial.

For further information on any topic presented, we suggest searching the **User Manual**, which providesin-depth descriptions of the tools and parameters.

To access this manual, from the main **Toolbar** click **Help**, point to **Manuals**, and then click the **User Manual**.



1.1.17. CONCLUSION

Rocky was used to study a transfer chute design.During

this tutorial, it was possible to:

- Create an animation of your simulation
- Visualize Properties in a 3D View window
- Plot Properties and Curves
- Filter data using User Processes
- Use post-processing tools to analyze and export the results

What's Next?

Now that you understand the basics of setting up and running a Rocky project, you are ready tomove on to next tutorial.

Ansys / ROCKY