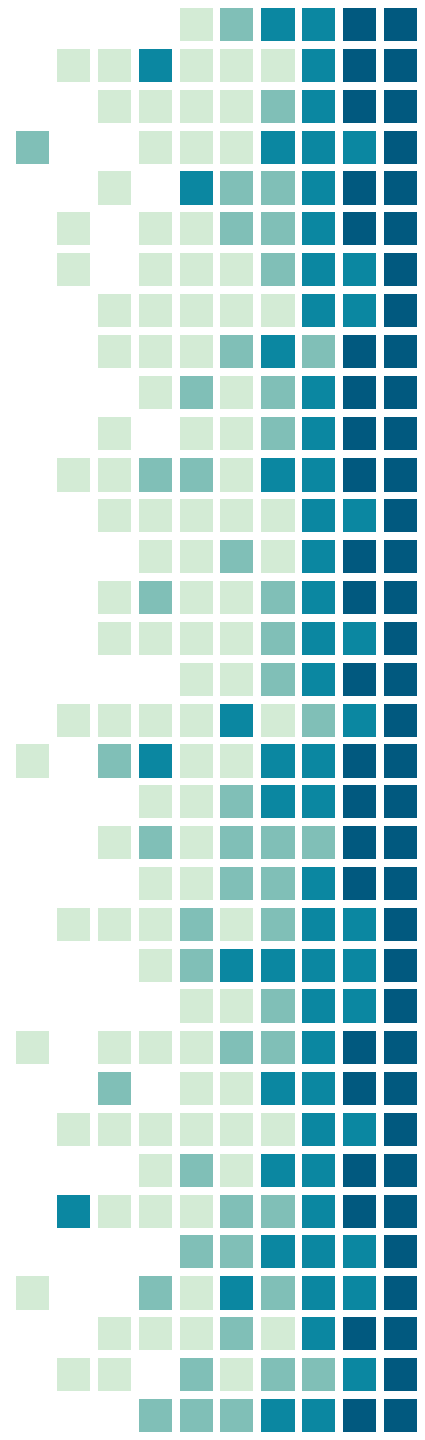# VeloxSim Particles V2.0 Tutorial Guide – Generic Chute

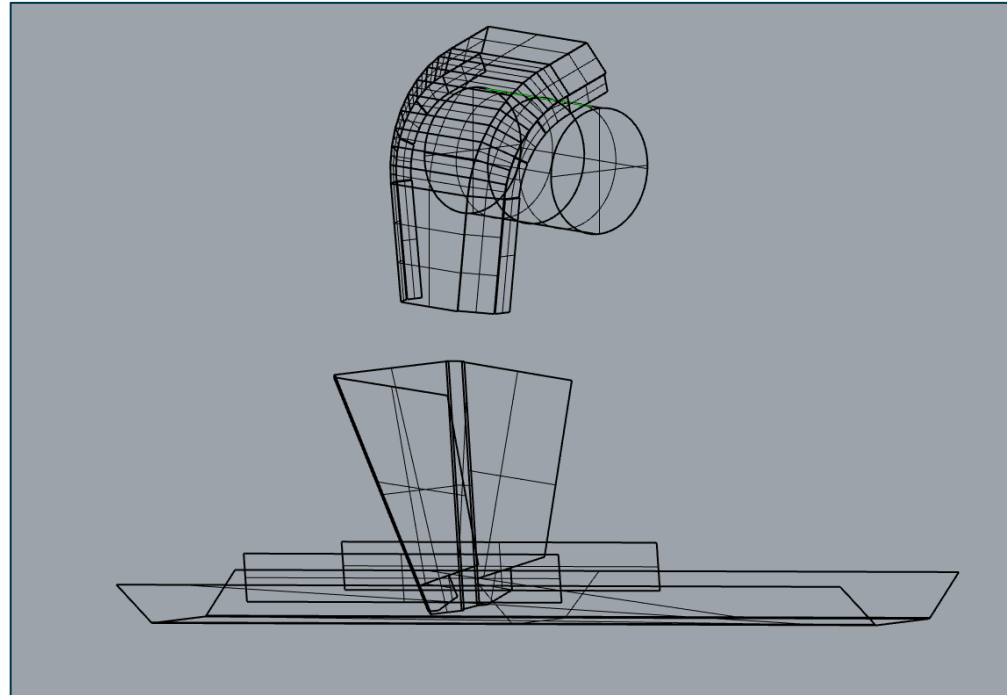Dr. Sam Wong – sam.wong@veloxsim.com

Limitations

- The unit of the Rhino model has to be in mm.

- Gravity direction is in the –ve World Z-axis

- To reduce numerical errors, the model has to be close to the world origin or (0,0,0)

- As a rule of thumb, just set the head pulley to be at (0,0,0) or within 1,000mm of the world origin.
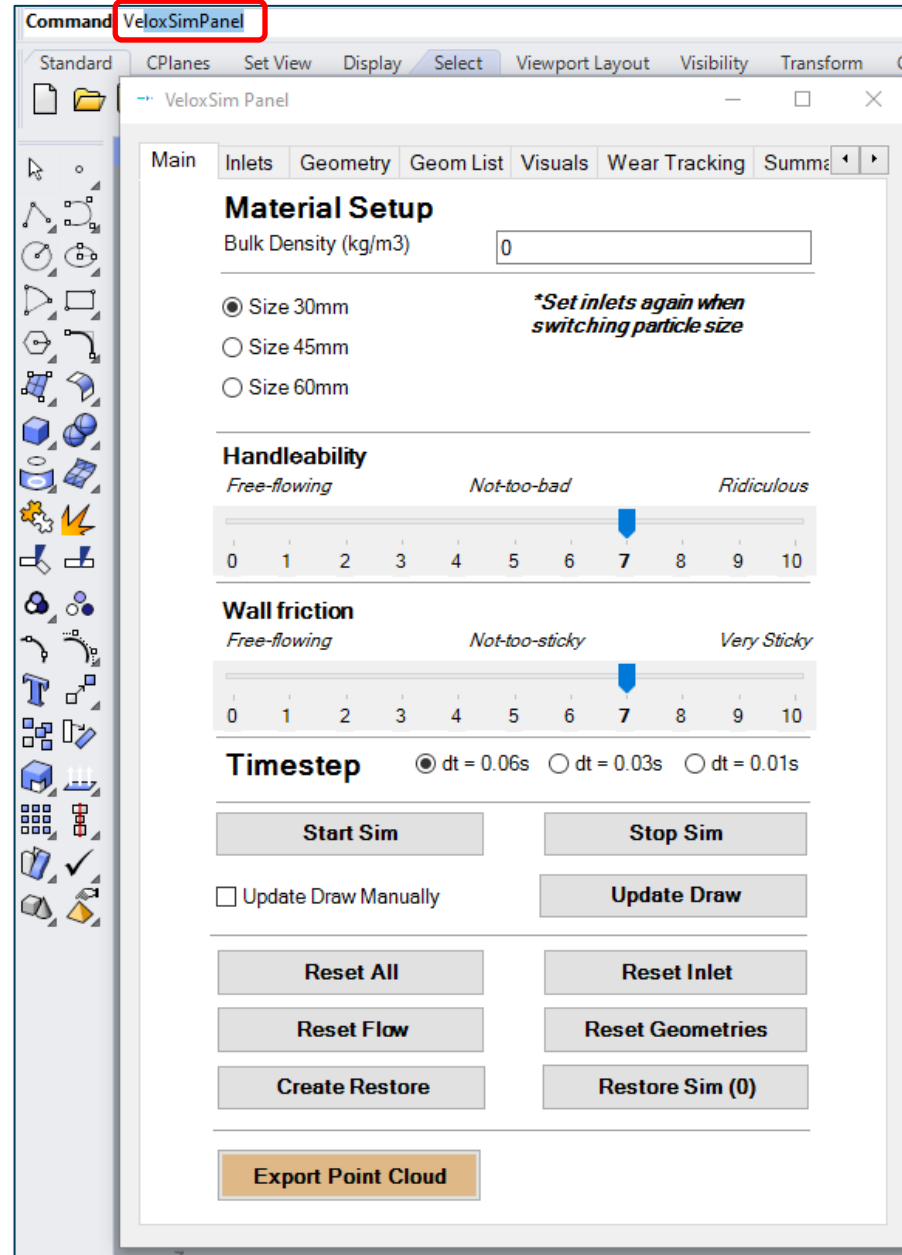
Generic Chute Tutorial

- This simple tutorial is to demonstrate how to set up a transfer chute simulation easily in VeloxSim Particles.

- Tonnage: **4,300tph**

- Bulk density: **1,800kg/m^3**

- Inlet belt speed: **3m/s**
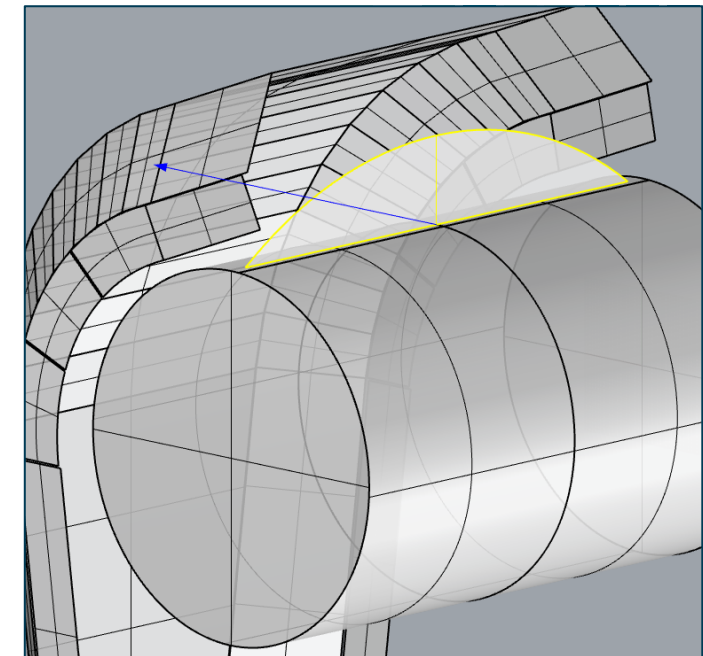
- Discharge belt speed: **3m/s**

Quick Start Tutorial

1. Open Rhino3D
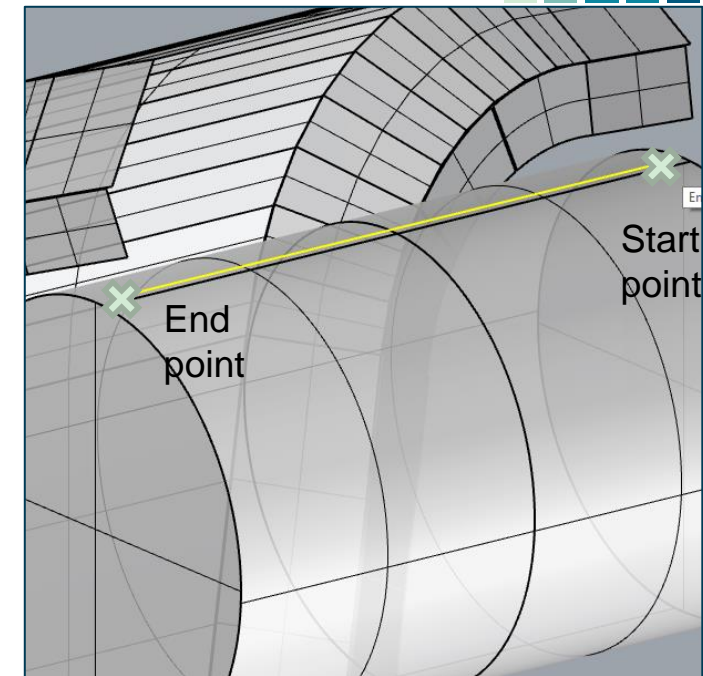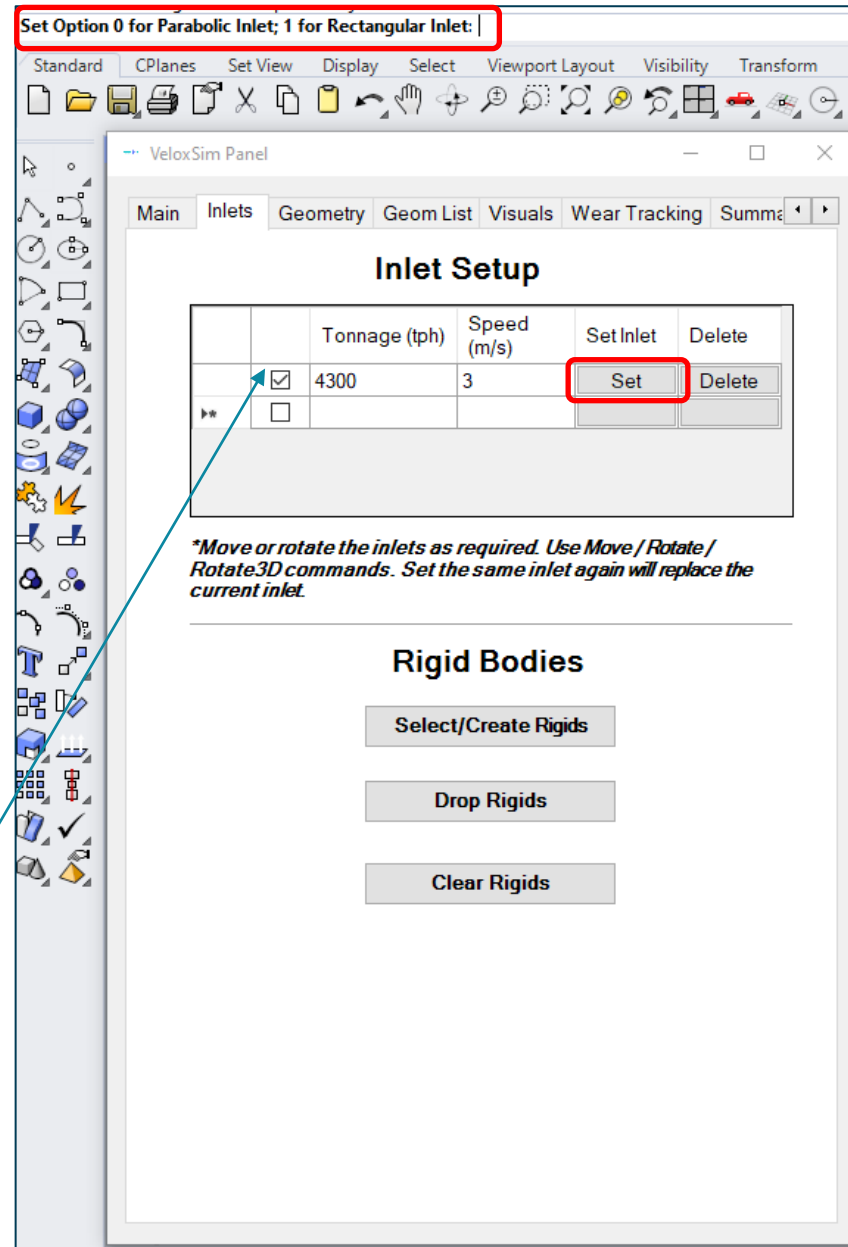
2. Type "VeloxSimPanel" in the command line to open the VeloxSim panel

3. Enter the key inputs

   - Bulk density of the material (kg/m^3)

4. Select the particle size

5. Select the handleability of the material using the sliding bar (No. 7 has been calibrated against one type of DEM material model for iron ore fines with 70° internal friction angle)
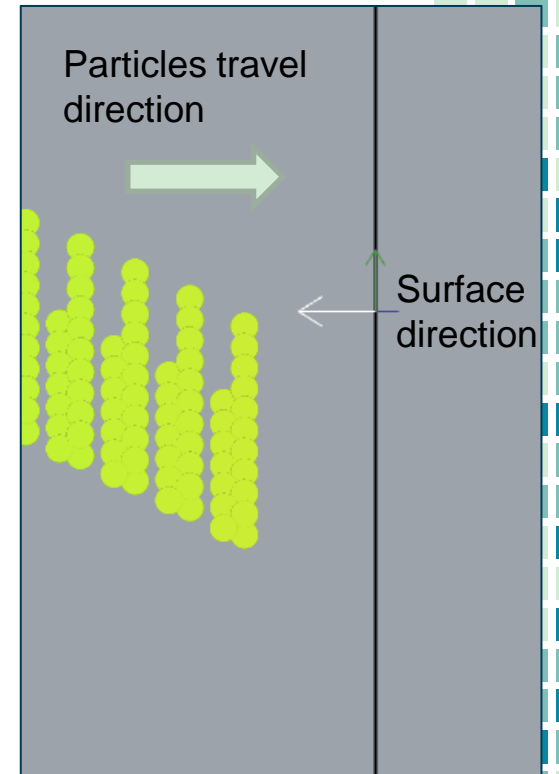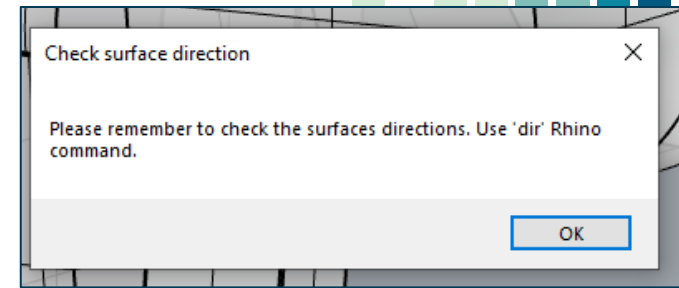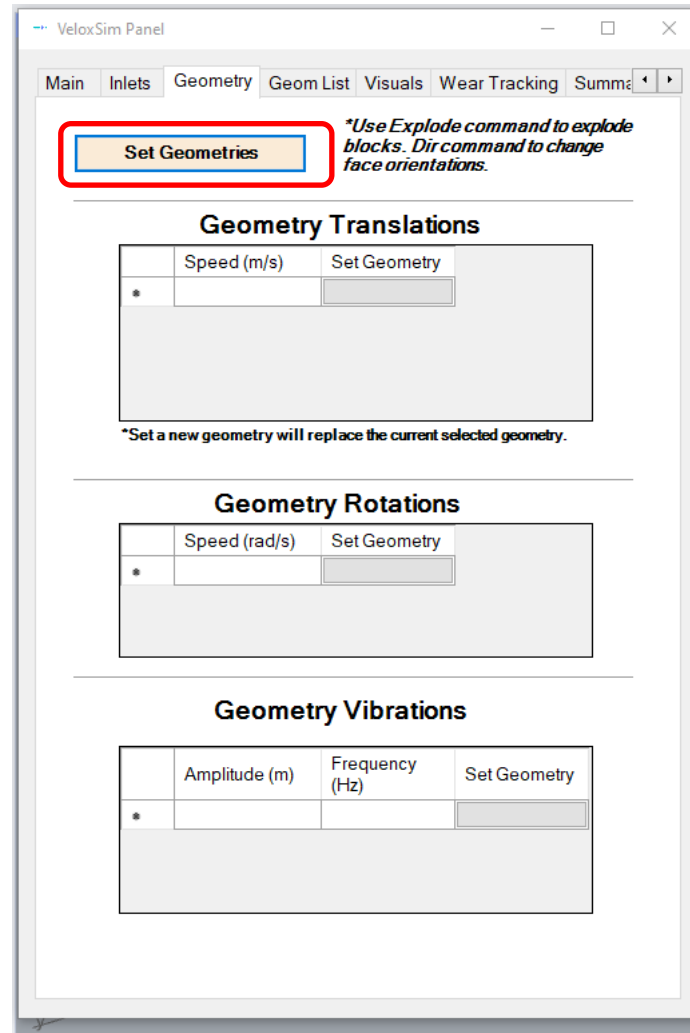
6. Choose available timestep size

Setup Feed Inlets

1. Select the "Inlets" tab. You are now able to set multiple inlets.

2. First, enter the required Tonnage and Speed.

3. Click "Set", you will prompted to choose a parabolic or rectangular inlet

4. Select the start and end points of the discharge burden

5. These start and end points should have taken into the account the belt edge distances

6. VeloxSim calculates the Cross-Sectional-Area (CSA) of the burden at the discharge point

7. Particles will be distributed across this inlet and injected into the simulation model

8. Make sure to tick the check box to enable the inlet

9. You can manipulate the direction and position of this inlet plane, VeloxSim will automatically update the particle injection points. User Move or Rotate / Rotate3D commands

Set Option 0 for Parabolic Inlet; 1 for Rectangular Inlet:

VeloxSim Panel

Main | Inlets | Geometry | Geom List | Visuals | Wear Tracking | Summa

Inlet Setup

		Tonnage (tph)	Speed (m/s)	Set Inlet	Delete
☑		4300	3	Set	Delete
▶	☐				

Move or rotate the inlets as required. Use Move / Rotate / Rotate3D commands. Set the same inlet again will replace the current inlet.

Rigid Bodies

Select/Create Rigids

Drop Rigids

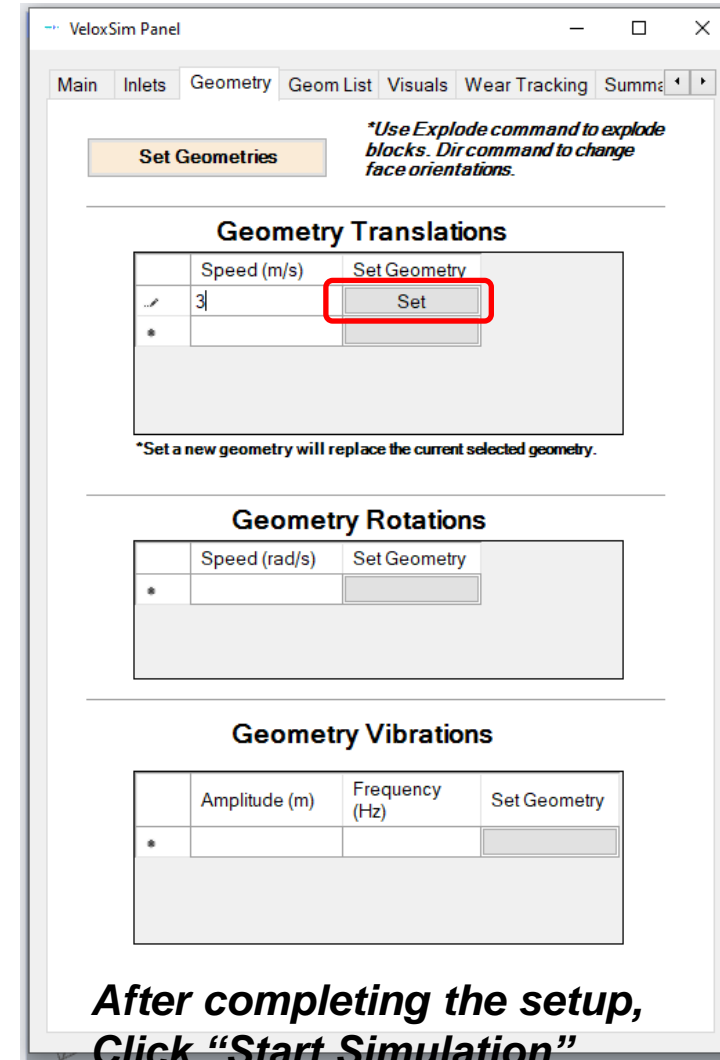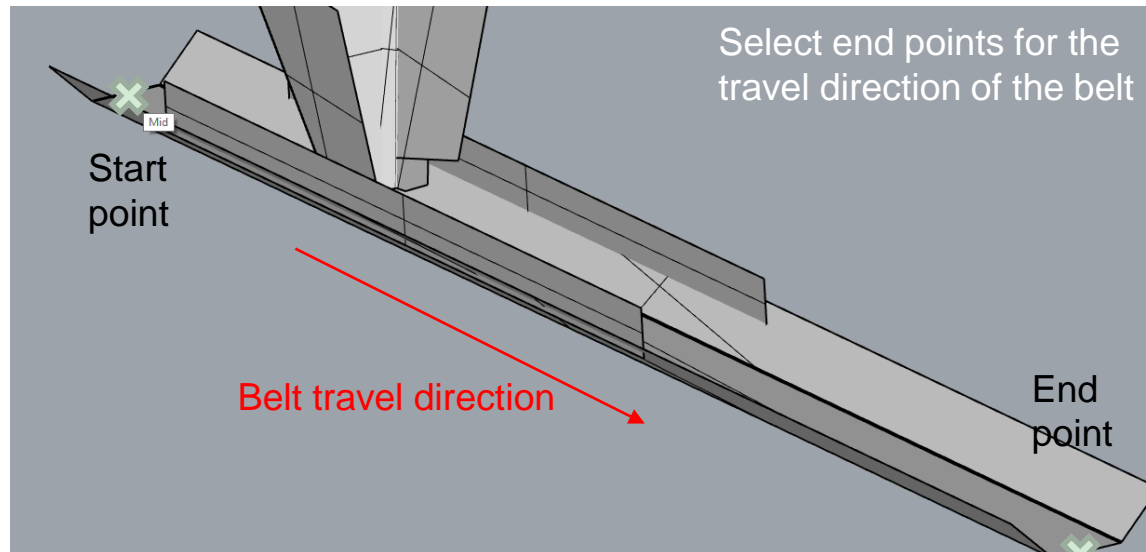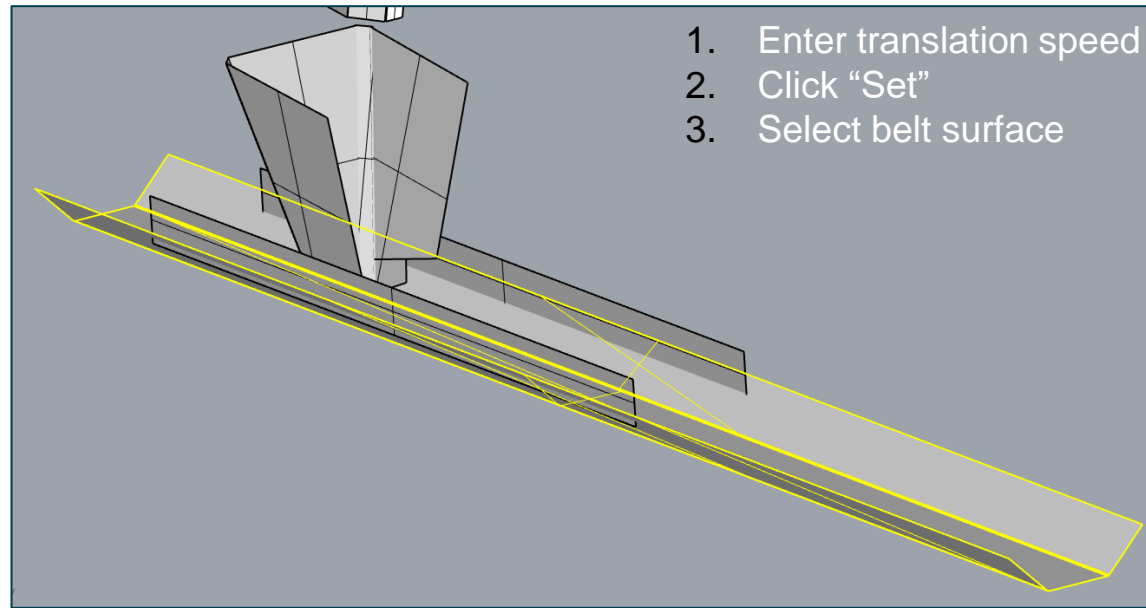Clear Rigids

Start point

End point

Quick Start Tutorial – Collision Geometries

1. To set the collision geometries, click "Set Geometries"

2. You will be prompted to select surface geometries in Rhino

3. Selected surfaces will be automatically meshed by VeloxSim

4. You will also be reminded to check the surfaces direction using the "*Dir*" command on the selected collision surfaces

5. The direction of the surface should be facing inwards, in the direction of where the particles are hitting, as shown in the snapshot on the right.

6. TIP: Use "*Explode*" command to split a group of surfaces into individual surfaces

VeloxSim Panel

Main | Inlets | Geometry | Geom List | Visuals | Wear Tracking | Summa

Set Geometries

Use Explode command to explode blocks. Dir command to change face orientations.

Geometry Translations

	Speed (m/s)	Set Geometry
*		

Set a new geometry will replace the current selected geometry.

Geometry Rotations

	Speed (rad/s)	Set Geometry
*		

Geometry Vibrations

	Amplitude (m)	Frequency (Hz)	Set Geometry
*			

Check surface direction

Please remember to check the surfaces directions. Use 'dir' Rhino command.

OK

Particles travel direction

Surface direction

Quick Start Tutorial – Discharge Belt

1. Enter translation speed
2. Click "Set"
3. Select belt surface

Select end points for the travel direction of the belt

Start point

Belt travel direction

End point

VeloxSim Panel

Main | Inlets | Geometry | Geom List | Visuals | Wear Tracking | Summa

Set Geometries

Use Explode command to explode blocks. Dir command to change face orientations.

Geometry Translations

	Speed (m/s)	Set Geometry
	3	Set
*		

Set a new geometry will replace the current selected geometry.

Geometry Rotations

	Speed (rad/s)	Set Geometry
*		

Geometry Vibrations

	Amplitude (m)	Frequency (Hz)	Set Geometry
*			

After completing the setup, Click "Start Simulation"
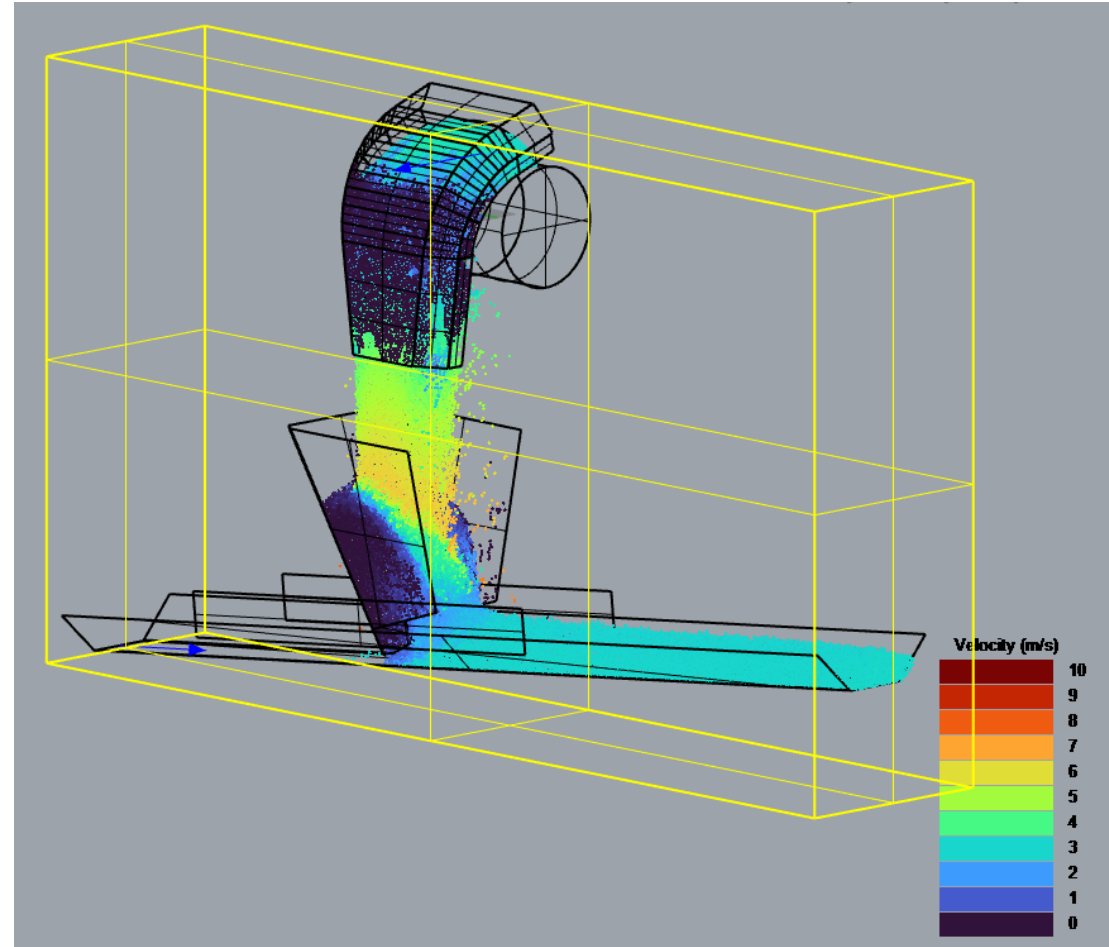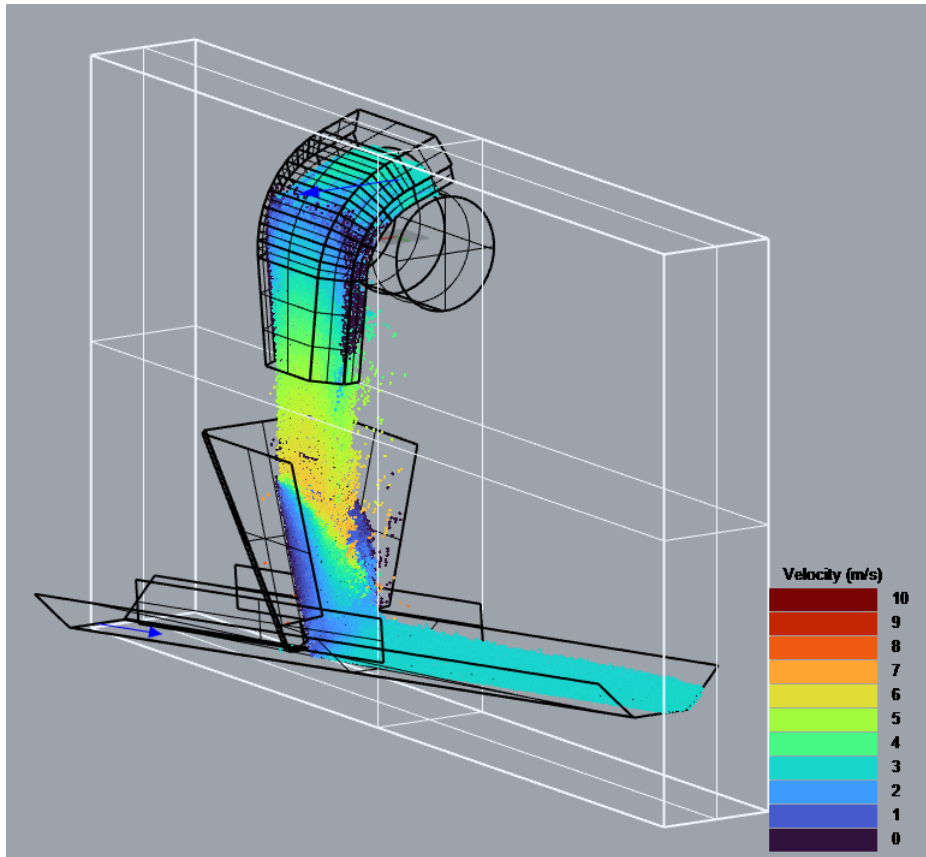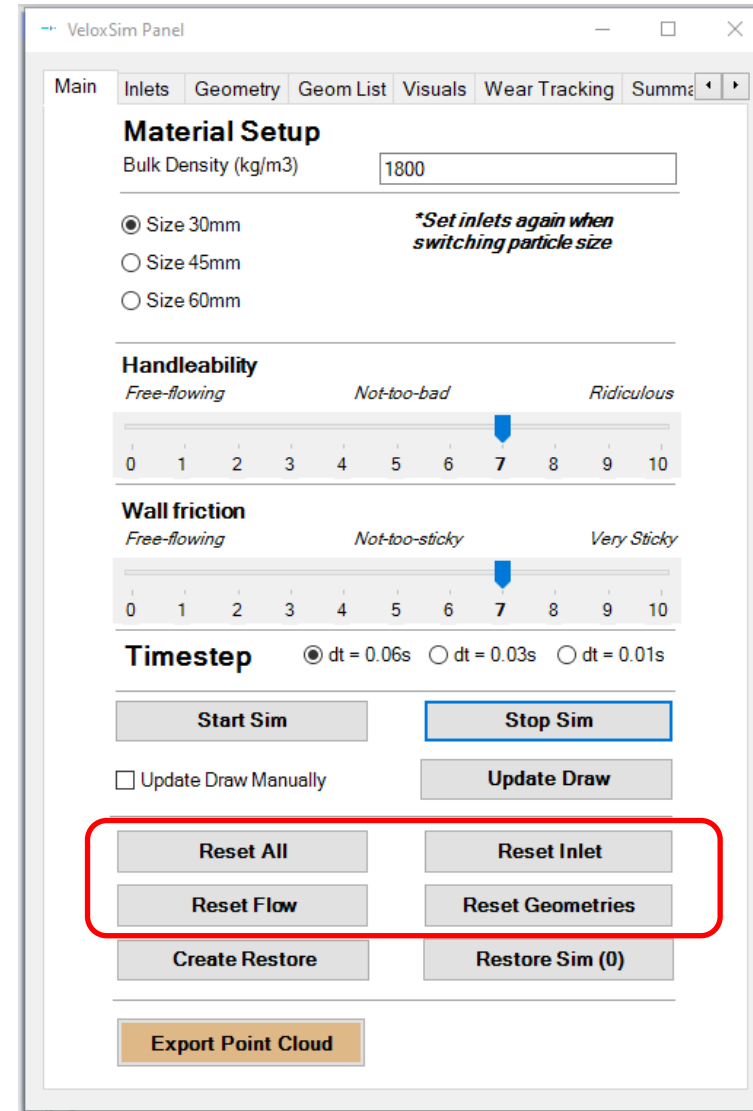
Quick Start Tutorial – Adjusting View Box

- After the simulation has started, a view box will be added to model under the layer named "VeloxSim_viewBox".

- Only particles within the domain of this box will be drawn by Rhino.

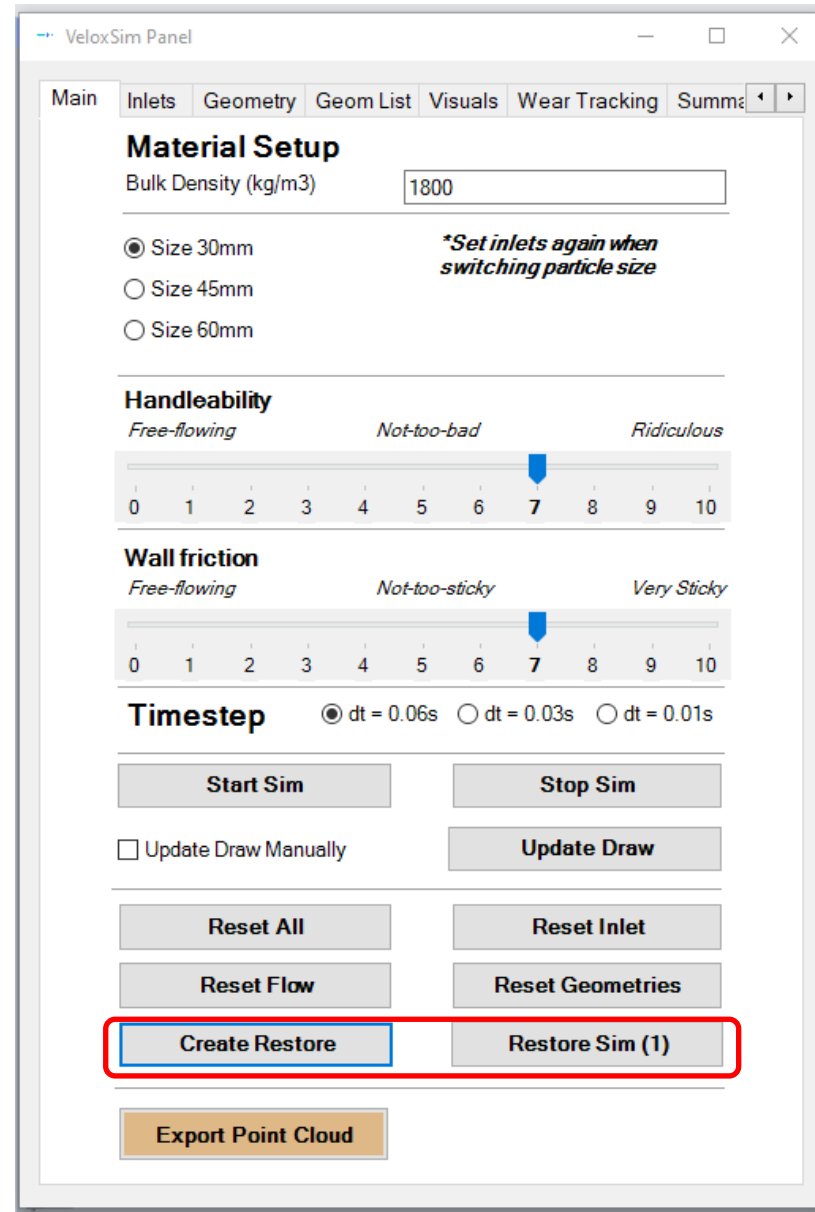- Hence, you can use it to clip the flow, just like the image below

Quick Start Tutorial – Resetting

- Reset All – This will all the settings, including inlet, collisions geometries and discharge belt; and also restart the simulation

- Reset Inlet – Only reset inlets

- Reset Flow – Only reset and restart the simulation

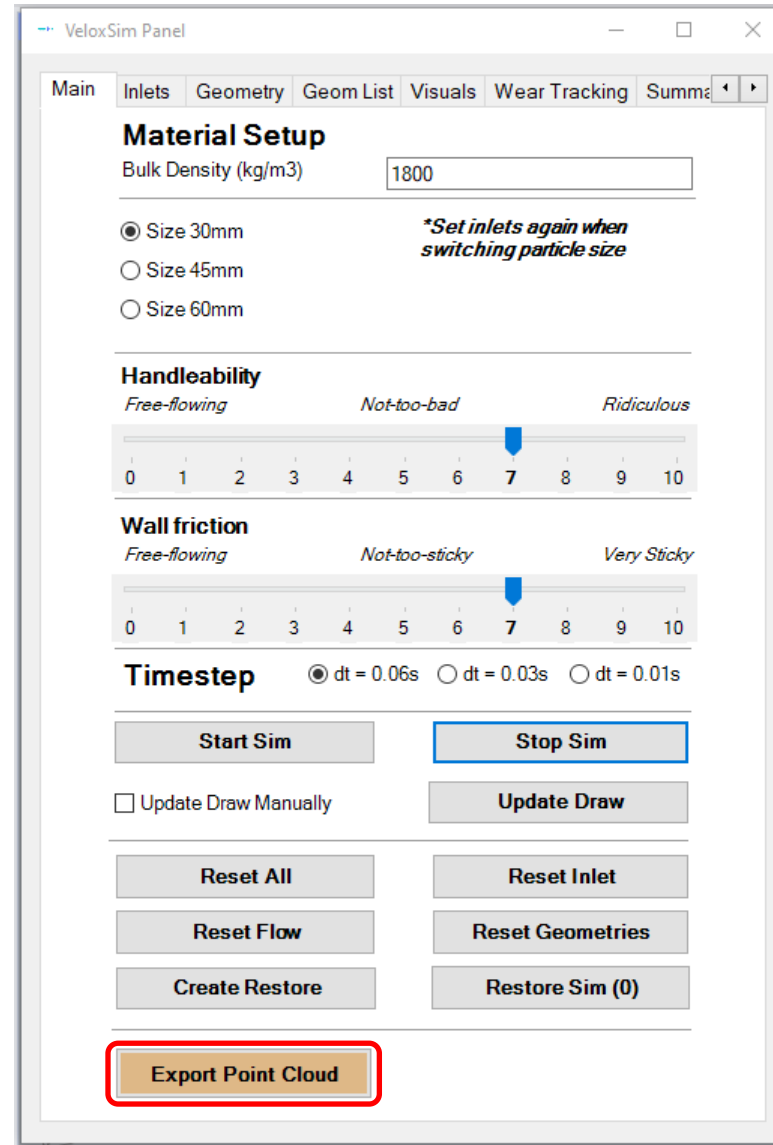- Reset Geometries – Only reset the geometries

Quick Start Tutorial – Create Restore Points

- You can create restore points in the simulation.

- Click "Create Restore" to generate a restore point.

- The number of restore point available is indicated beside the "Restore Point" button.

Quick Start Tutorial – Export Point Cloud

- You can export the particles in the simulation to a point cloud.

- The exported point cloud will be group in the layer named "VeloxSim Point Cloud".

FAQ

Q: Particles are tunnelling through a surface.

A: This is usually due to the surface in question is not facing inwards to the direction of the particles. Use Rhino command "Dir" to check the direction of the surface, if the direction is wrong, flip the direction by toggling "f" when you are still in the "Dir" command. Otherwise, try redrawing that surface.