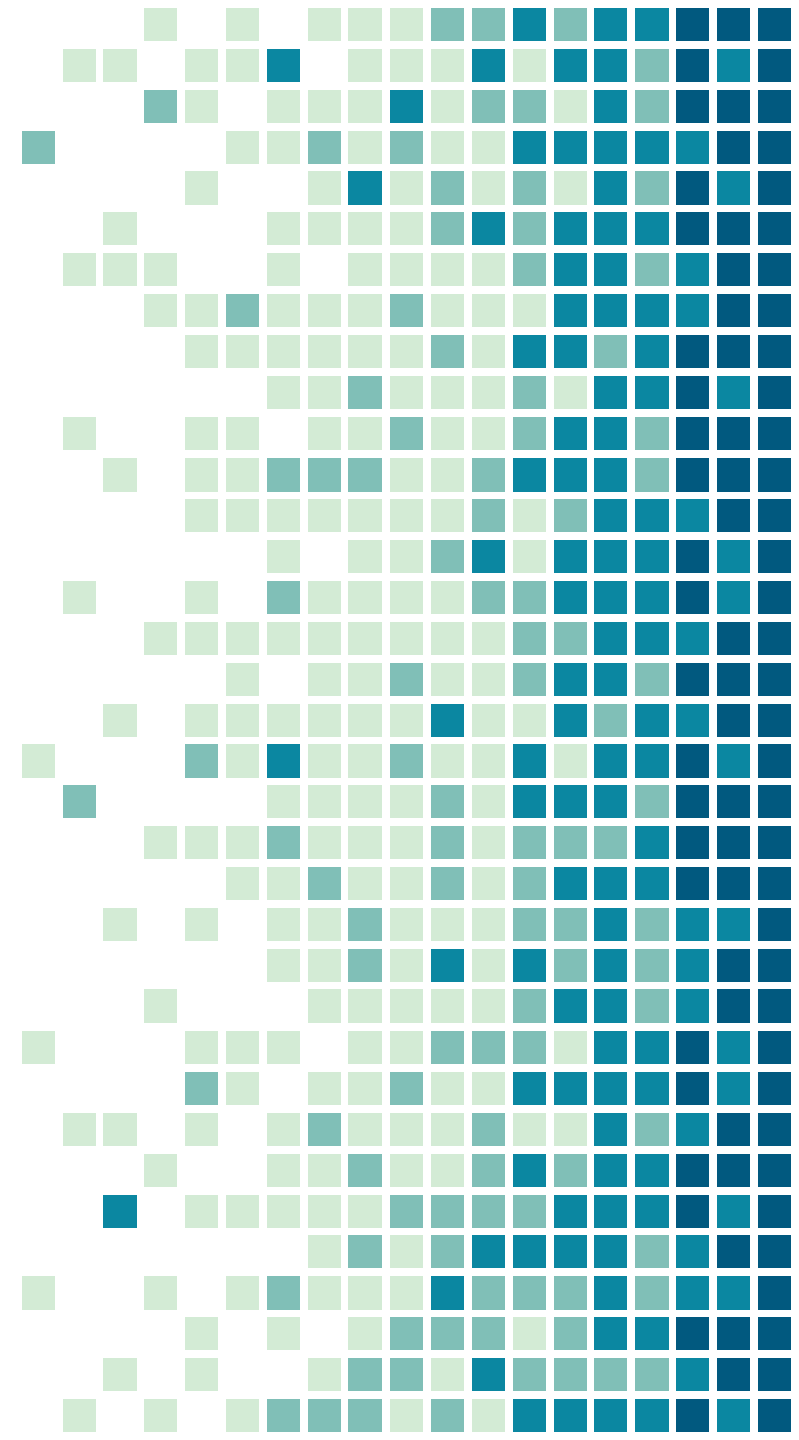


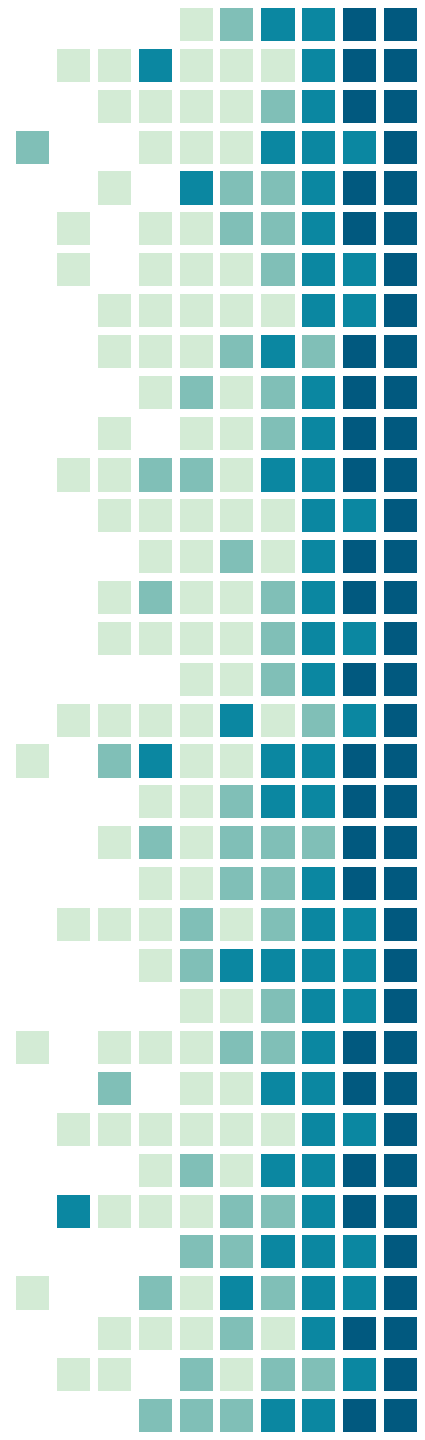
# VeloxSim Rigid v2.0 Tutorial Guide – Rigids

Dr. Sam Wong – [sam.wong@veloxsim.com](mailto:sam.wong@veloxsim.com)



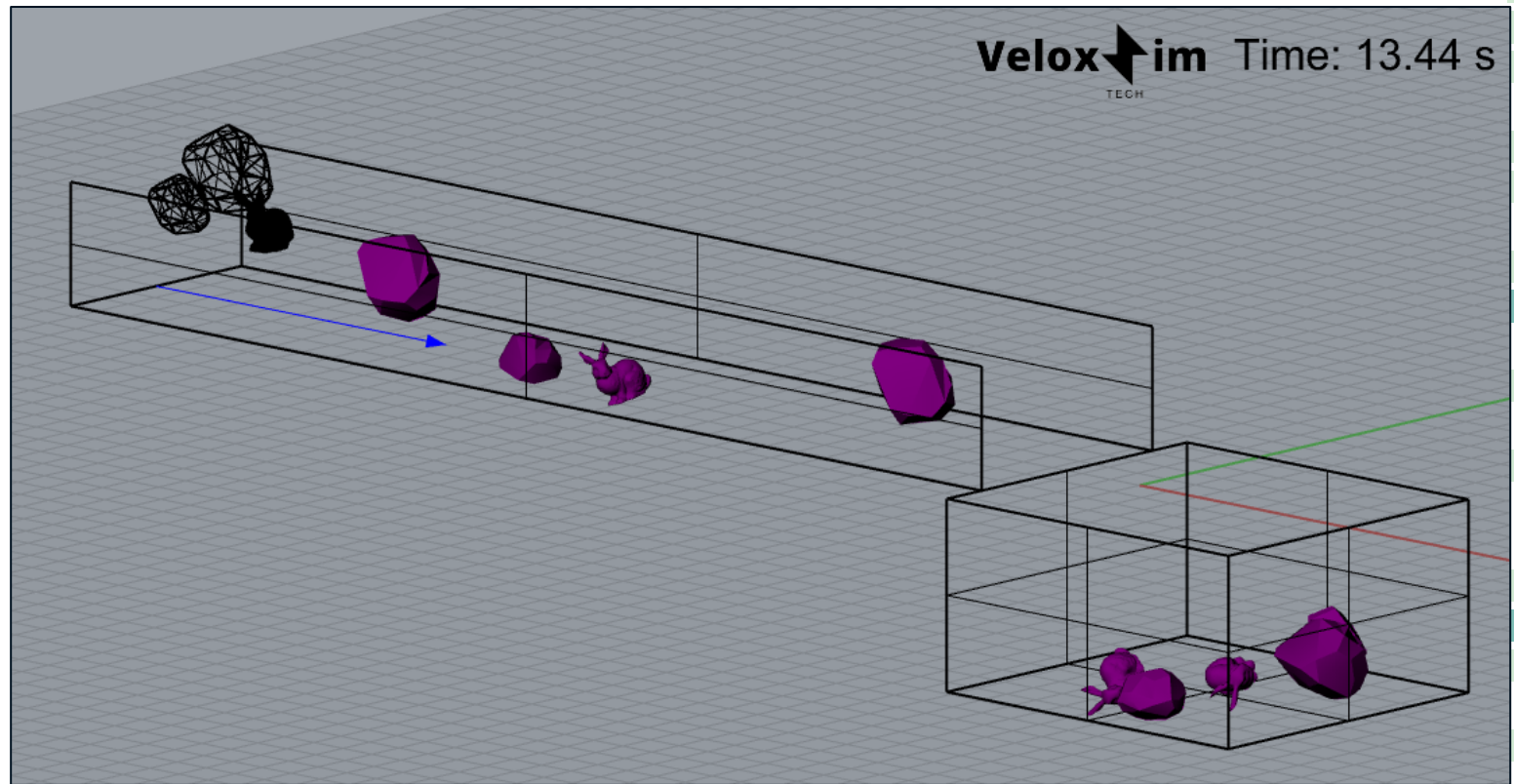
# Limitations

- 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.
- If the rigid is close to 1m in size, please use larger particle size to pack the rigid.



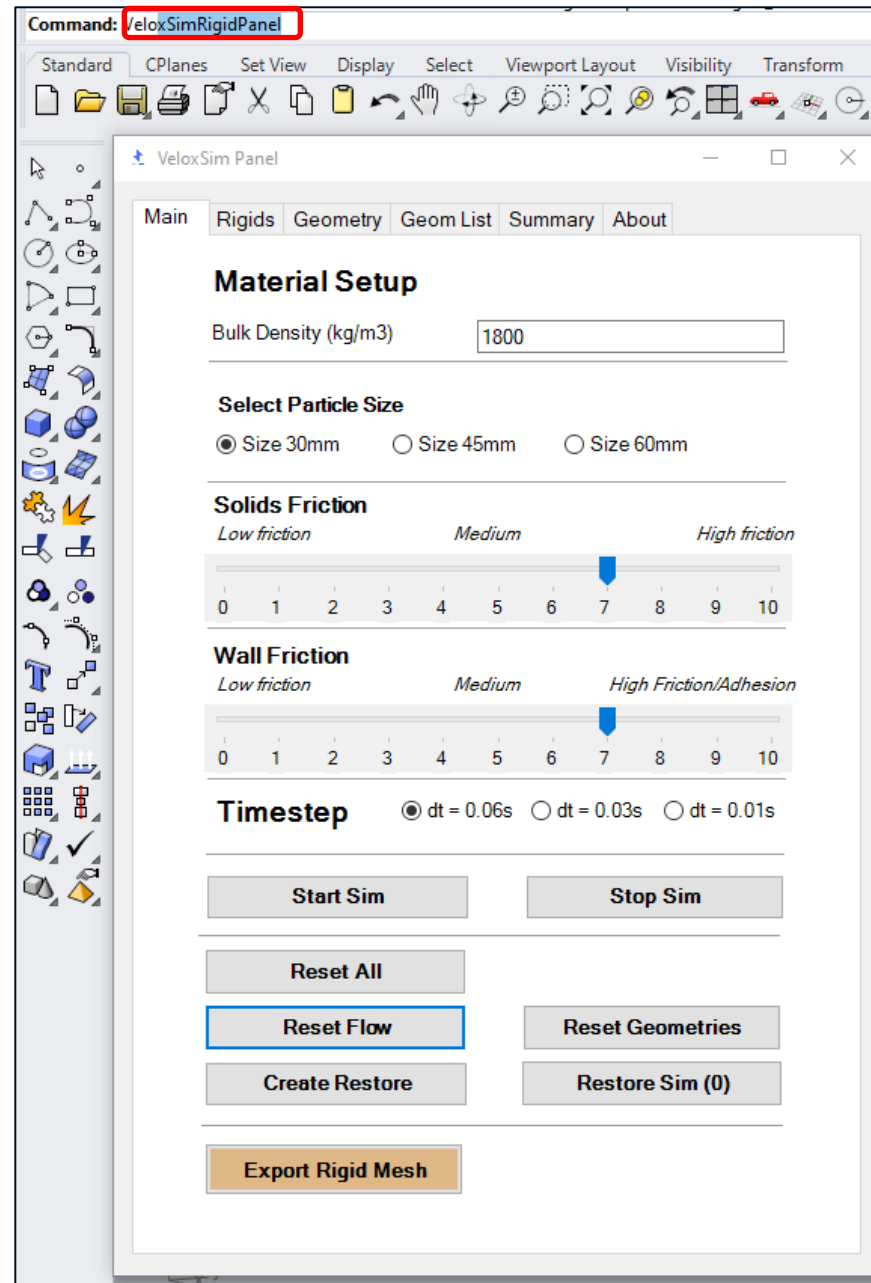
# Tutorial – Generate Rigid

- This simple tutorial goes through the simple steps of generating rigids, set a moving conveyor belt in VeloxSim Rigids



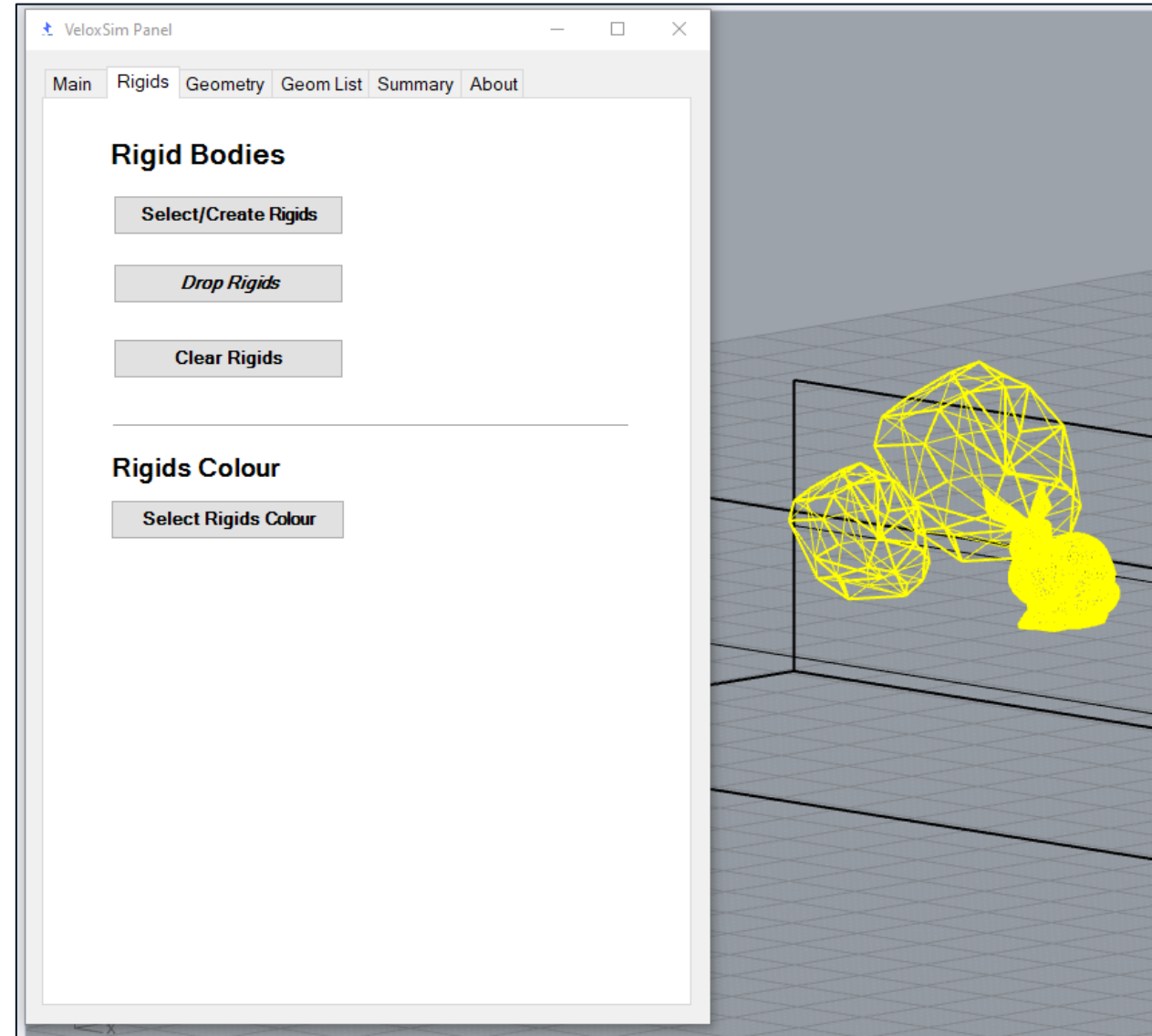
# Quick Start Tutorial

1. Open Rhino7/6
2. Type “VeloxSimRigidPanel” in the command line to open the VeloxSim panel
3. Enter the key inputs
  - Bulk density of the material ( $\text{kg/m}^3$ )
  - This will automatically calculate the solids density based on a maximum 40% packing factor
4. Select the particle size
5. Select the friction factors between the particles and particles-wall
6. Choose available timestep size



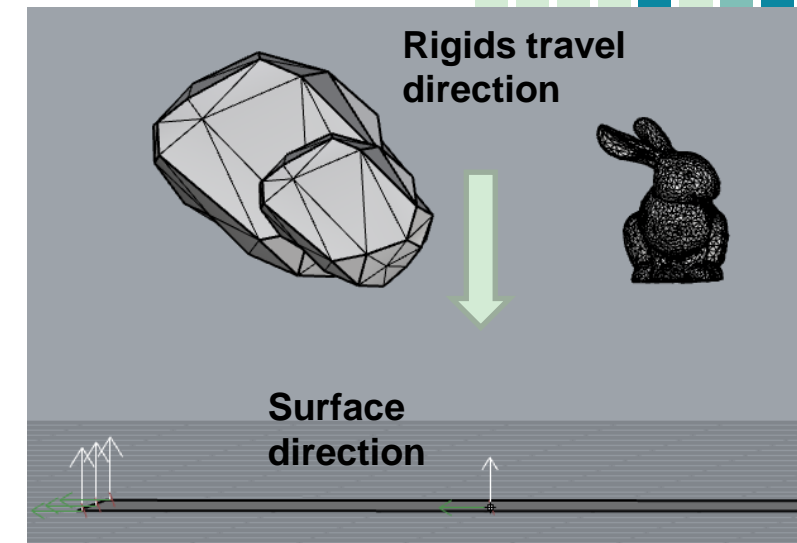
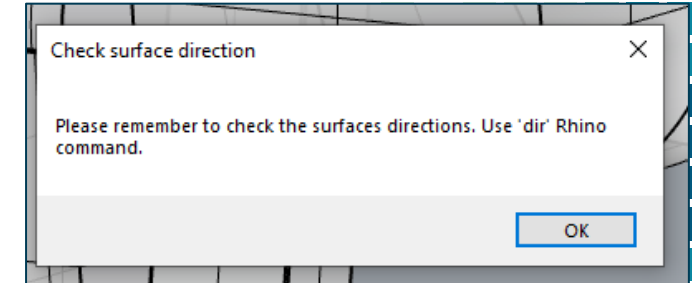
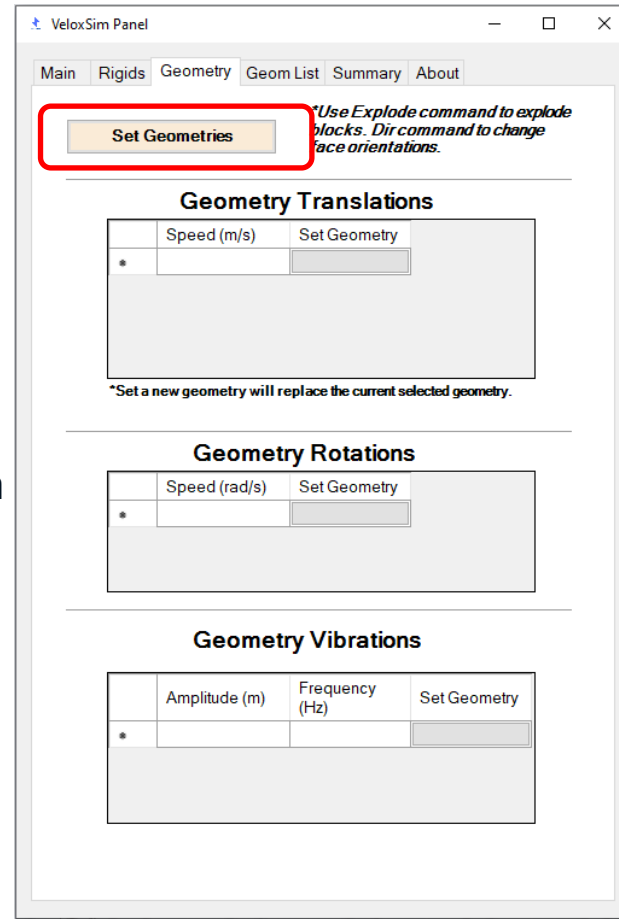
# Quick Start Tutorial – Create Rigid

1. In the VeloxSim panel, switch to the Rigid tab.
2. Creating solids is straightforward, select closed meshes (only work with closed meshes at the moment), and click “Select/Create Rigid”.
3. When simulation starts, the engine will create and drop the rigid.
4. To manually drop the rigid during simulation, click “Drop Rigid”.
5. “Clear Rigid” will reset the selected rigid meshes.
6. You can change the colour of the rigid by clicking “Select Rigid Colour”.



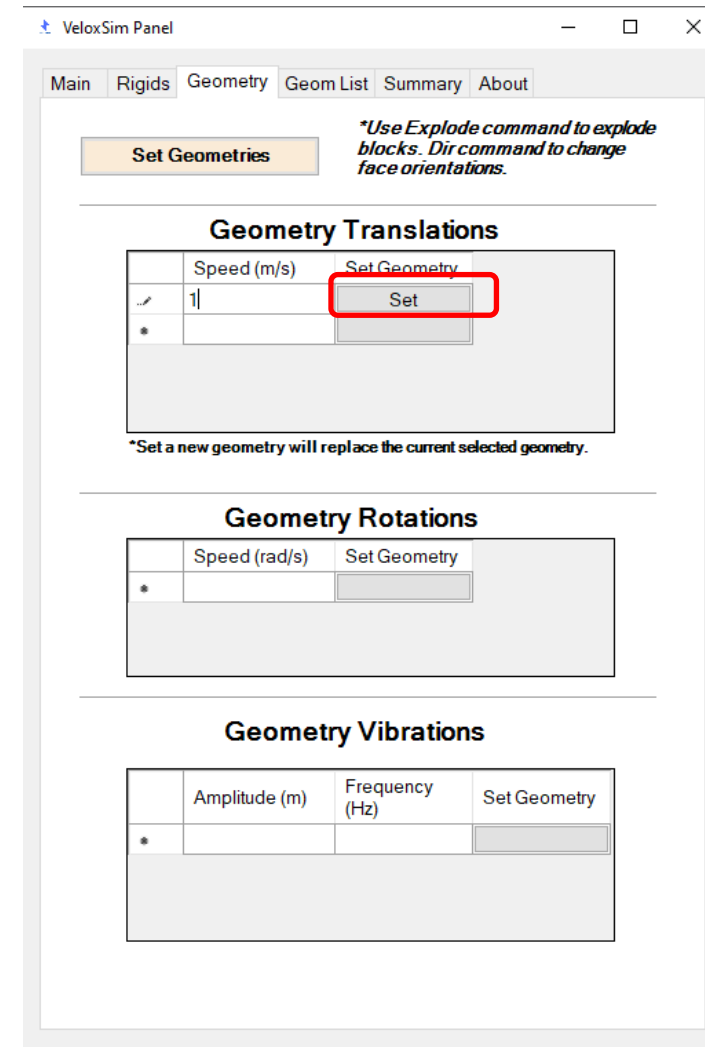
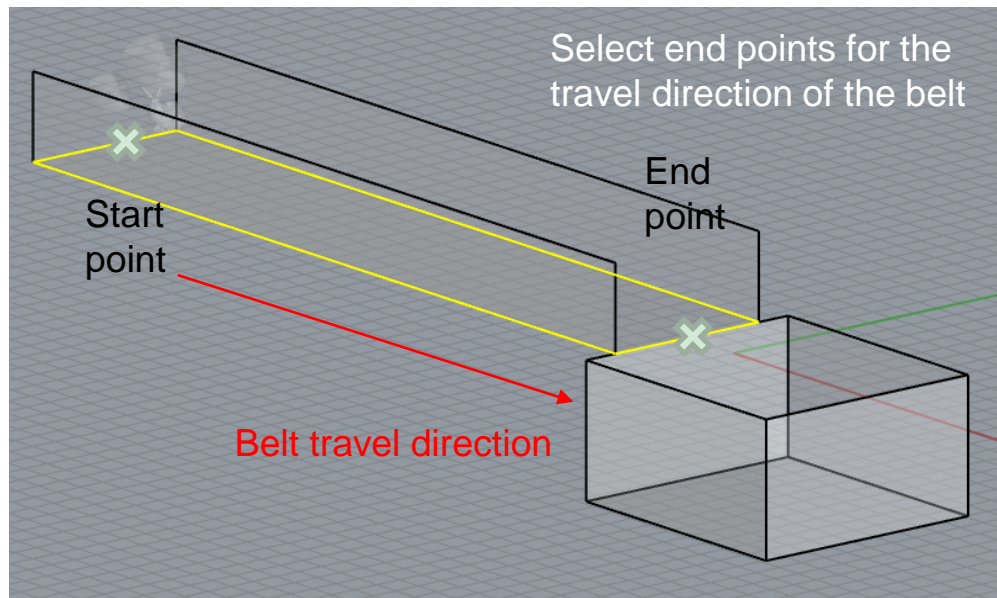
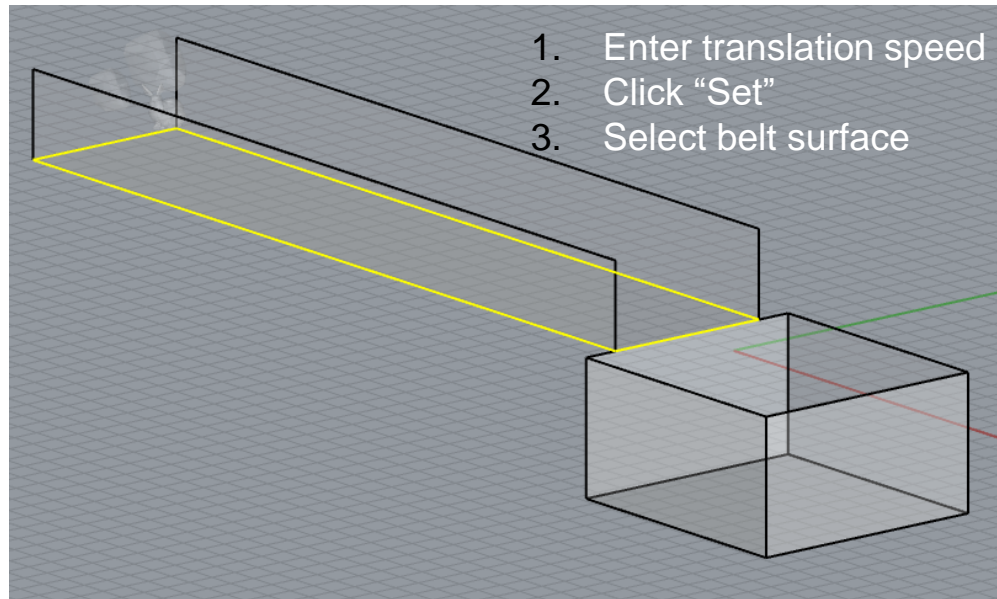
# Quick Start Tutorial – Collision Geometries

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





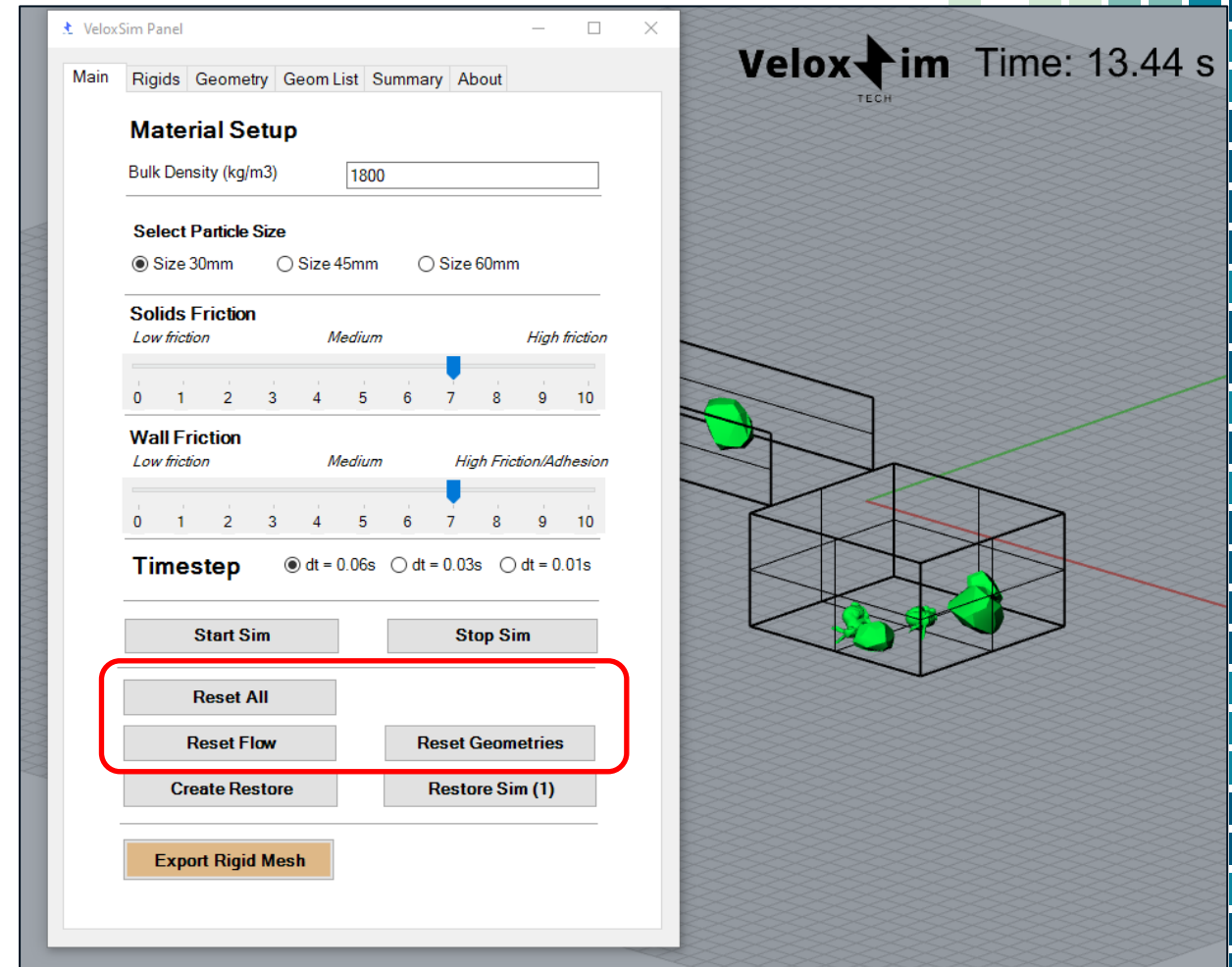
# Quick Start Tutorial – Moving Belt



**After completing the setup,  
Click "Start Simulation"**

# Quick Start Tutorial – Resetting

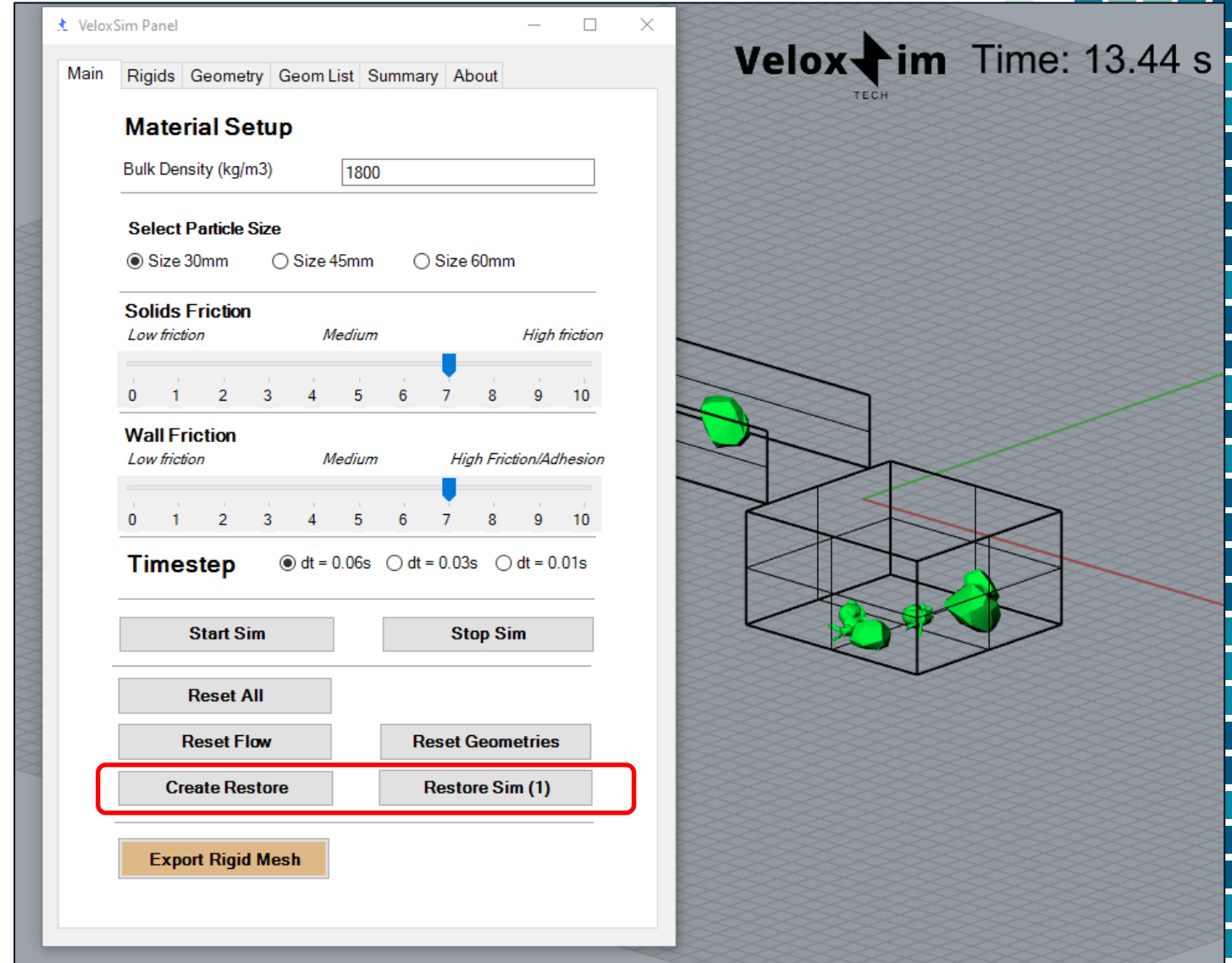
- Reset All – This will all the settings, including inlet, collisions geometries and discharge belt; and also restart the simulation
- Reset Flow – Only reset and restart the simulation
- Reset Geometries – Only reset the geometries





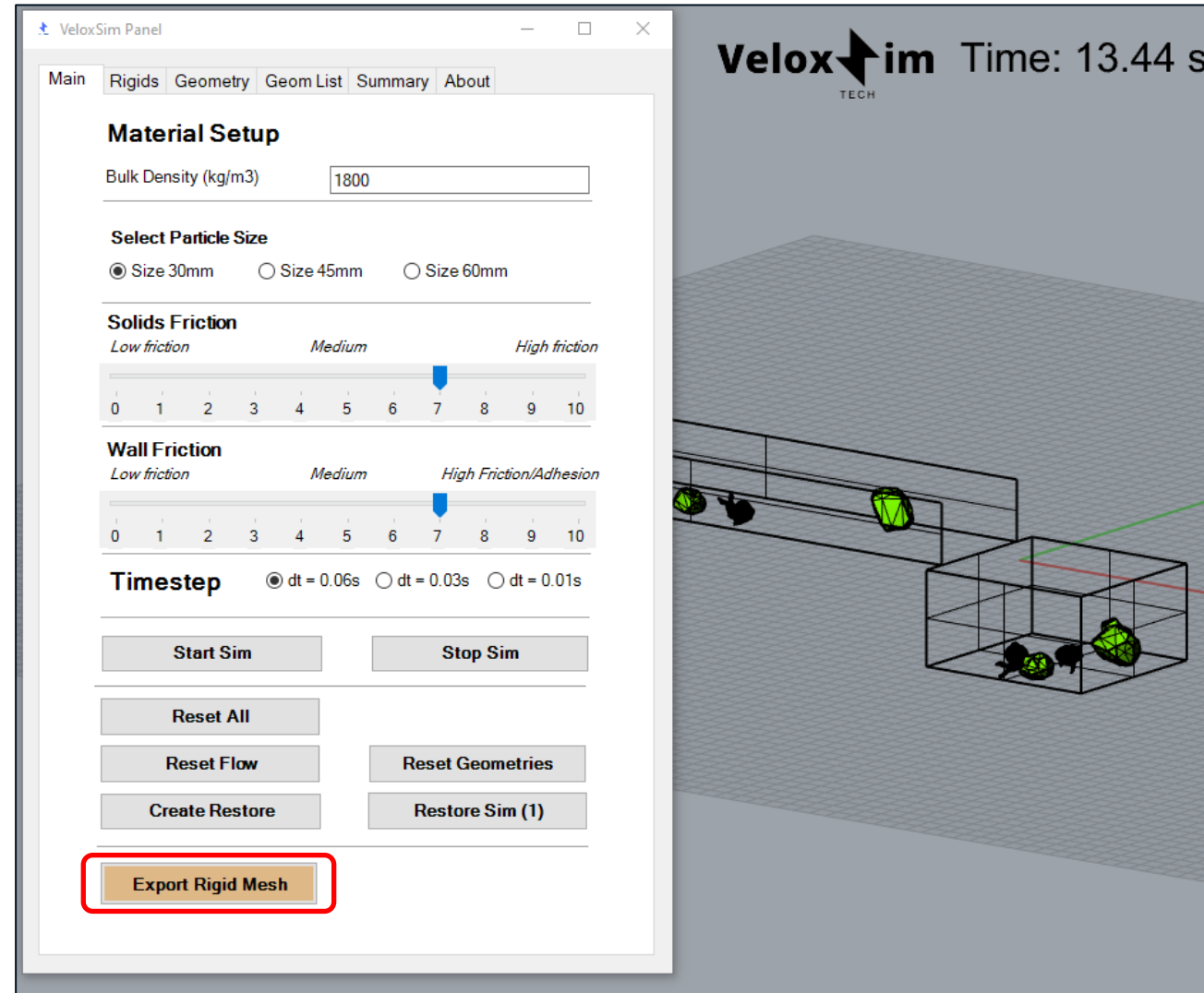
# Quick Start Tutorial – Create Restore Points

- 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 Rigids Meshes

- You can export the rigids in the simulation into meshes.



# 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.

