



EDEM 2.4 User Guide

Copyrights and Trademarks

Copyright © 2011 DEM Solutions. All rights reserved.

Information in this document is subject to change without notice. The software described in this document is furnished under a license agreement or nondisclosure agreement. The software may be used or copied only in accordance with the terms of those agreements. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or any means electronic or mechanical, including photocopying and recording for any purpose other than the purchaser's personal use without written permission.

DEM Solutions
49 Queen Street
Edinburgh
EH2 3NH
UK

www.dem-solutions.com

EDEM incorporates CADfix translation technology. CADfix is owned, supplied by and Copyright © TranscenData Europe Limited, 2007. All Rights Reserved. This software is based in part on the work of the Independent JPEG Group. EDEM uses the Mersenne Twister random number generator, Copyright © 1997 - 2002, Makoto Matsumoto and Takuji Nishimura, All rights reserved. EDEM includes CGNS (CFD General Notation System) software. See the Online Help for full copyright notice.

EDEM® and Particle Factory® are registered trademarks of DEM Solutions. All other brands or product names are the property of the respective owners.

Contents

GETTING STARTED	6
INTRODUCTION TO EDEM	6
EDEM USER INTERFACE.....	7
ARRANGING THE USER INTERFACE	8
CAMERA AND MOUSE ACTIONS	9
CHOOSING UNITS.....	12
MANAGING LICENSES.....	13
EDEM CREATOR	15
ABOUT THE EDEM CREATOR	15
<i>Tabs Pane</i>	16
<i>Viewer</i>	16
<i>Viewer Controls (Creator)</i>	16
<i>Data Browser (Creator)</i>	18
<i>Creator Toolbar</i>	19
<i>Creator Menu Bar</i>	19
GLOBAL SETTINGS	21
THE MATERIALS DATABASE	28
DEFINING PARTICLES.....	29
THE MODEL DOMAIN	32
CREATING GEOMETRY SECTIONS	32
<i>Importing Geometry Sections</i>	38
<i>Merging Geometry Sections</i>	40
<i>Copying Geometry Sections</i>	40
PARTICLE FACTORIES.....	40
<i>Creating Particle Factories</i>	40
EDEM SIMULATOR	46
ABOUT THE EDEM SIMULATOR.....	46
<i>Simulator Pane</i>	46
<i>Viewer</i>	47
<i>Viewer Controls (Simulator)</i>	47
<i>Solve Report (Simulator)</i>	48
<i>Simulator Toolbar</i>	49
<i>Simulator Menu Bar</i>	49
SIMULATOR SETTINGS	50
<i>Selective Saves</i>	51
<i>Simulator Grid</i>	53
<i>Collision Detection</i>	53
<i>Processors</i>	54
<i>Starting and Stopping a Simulation</i>	54
REDUCING SIMULATION TIME.....	55
EDEM ANALYST	57
ABOUT THE EDEM ANALYST.....	57
<i>Tabs Pane (Analyst)</i>	58
<i>Viewer</i>	58
<i>Viewer Controls (Analyst)</i>	58
<i>Data Browser (Analyst)</i>	59
<i>Analyst Toolbar</i>	60

<i>Analyst Menu Bar</i>	61
REVIEWING YOUR SIMULATION.....	61
MODEL TAB	62
COLORING	67
SELECTIONS AND BINNING	69
<i>Manual Selections</i>	69
<i>Grid Bin Groups</i>	70
<i>Geometry Bins</i>	71
<i>Imported Geometry Bin Groups</i>	72
<i>Mass Flow Analysis</i>	72
<i>Selection Queries</i>	73
<i>Type Filter</i>	77
<i>Representation</i>	77
TOOLS TAB	79
<i>Ruler</i>	79
<i>Protractor</i>	79
<i>Domain Grid</i>	80
CLIPPING	80
SAVING IMAGES.....	83
CREATING VIDEOS	84
EXPORTING DATA.....	87
<i>General Settings</i>	88
<i>Creating a Query</i>	89
TRUNCATING RESULTS.....	93
GRAPHING	93
<i>Creating Graphs</i>	93
<i>Creating a Histogram (Bar Chart)</i>	94
<i>Drawing a Line Graph</i>	100
<i>Drawing a Scatter Plot</i>	101
<i>Creating a Pie Chart</i>	105
RUNNING EDEM IN BATCH MODE.....	110
EDEM'S COMMAND LINE INTERFACE (CLI)	110
CUSTOM PROPERTIES	112
<i>Creating a Custom Property</i>	112
<i>Viewing Custom Properties</i>	112
APPENDIX A: ESTIMATING SIMULATION TIME	113
APPENDIX B: ATTRIBUTE DEFINITIONS	115
<i>Particle Attributes</i>	115
<i>Geometry Attributes</i>	117
<i>Contact Attributes</i>	118
<i>Collision Attributes</i>	120
<i>Bonding Attributes</i>	121
APPENDIX C: EDEM FILE TYPES.....	122
REFERENCES.....	124
<i>General DEM</i>	124
<i>Contact Models</i>	124
<i>Particle Body Forces</i>	125

TECHNICAL SUPPORT 126
GLOSSARY 127
INDEX..... 131

Getting Started

Introduction to EDEM

EDEM™ is a DEM (Discrete Element Method) simulation software platform designed for the simulation and analysis of bulk particle handling and processing operations.

With EDEM you can quickly and easily create a parameterized model of your bulk solids system. CAD models of real particles can be imported to obtain an accurate representation of their shape. Add the mechanical, material and other physical properties to form your model particles. These can be stored in a library allowing you to build a collection specific to your processes.

EDEM manages information about each individual particle (mass, velocity and so on) and the forces acting on it. It can also take into account the particle's shape, rather than assuming that all particles are spherical. For post-processing, EDEM provides data analysis tools, 3D visualizations of the particle flow, and video creation.




EDEM's Particle Factory™ technology provides a unique method for efficiently generating assemblies of particles in association with the machine geometry imported from your CAD or CAE system as a solid model or mesh. Machine components can be grouped, moved, and copied; and you can specify the kinematics of each group individually.

User Defined Libraries programmed in C++ using EDEM's API (Applications Programming Interface) enable extension of EDEM's default physics capability to simulate almost any particle-particle, particle-fluid, particle-structure, or particle-electromagnetic interaction using external models, custom, or third-party solvers.

With EDEM's powerful post-processing tools you can visualize and graph any combination of variables. Having identified important system behavior you can easily modify models to refine your simulation. You can then use the video export feature to create video files of your simulation. EDEM can even record multiple simulations in the same video.

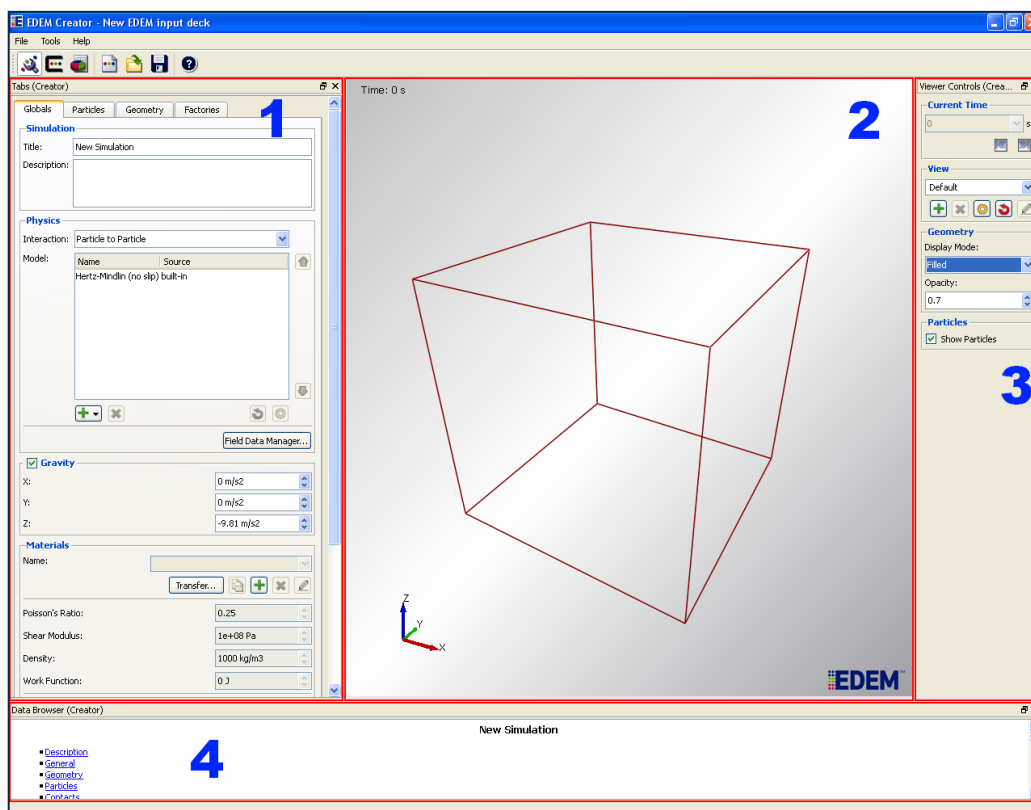
EDEM User Interface

EDEM is made up of three sections: the Creator, the Simulator and the Analyst.

-  **Creator** – set up and initialize models
-  **Simulator** – configure and control the EDEM solver engine
-  **Analyst** – EDEM's data analysis and visualization tool

You can move between each section by clicking on the toolbar button indicated above.

The layout of the user interface is common to each section. It consists of: the Tabs Pane, the Viewer, the Viewer Controls and the Data Browser.

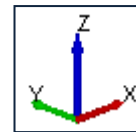


1 The Tabs Pane

The Tabs Pane is displayed at the on the left side of the EDEM window. It is made up of a number of individual panes. The contents of each pane vary according to which section of EDEM is currently active.

2 The Viewer

The Viewer displays 3D representations of your particles and geometry. The orientation, position and zoom factor of the Viewer are controlled using the mouse. The arrows in the bottom-left corner of the viewer indicate the current orientation: red (x), green (y) and blue (z).



See *Camera and Mouse Actions* on page 9 for more details.

3 The Viewer Controls

The Viewer Controls are used to determine how items are displayed in the Viewer: For example, the fill-style of a particle or the opacity of a geometry. The content of the pane varies according to which section of EDEM is currently active.

4 The Data Browser

The Data Browser is an .html page that displays detailed information about the contents of the Viewer: For example, particle properties and interactions or the dimensions of a geometry section. The content of the page varies according to which section of EDEM is currently active.

Arranging the User Interface

Floating and Docked Panes

The Tabs pane, Viewer controls and Data browser can each be floating or docked. By default all panes are docked. To undock a pane double click on its title bar (control area) and place it anywhere in screen. Double click the title bar to re-dock it in its original position or drop it over the left or right edge or the bottom of the screen to dock it in that position.

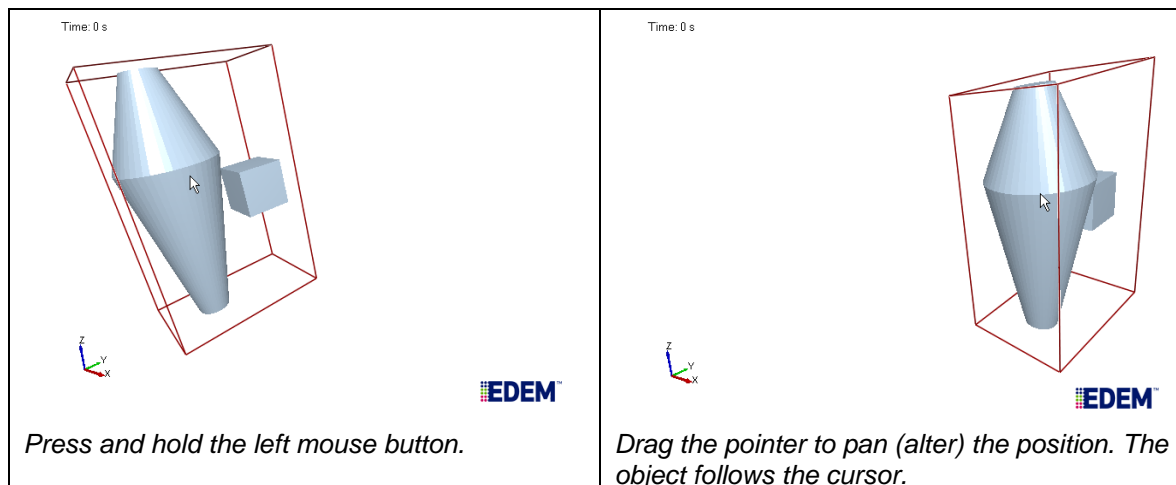
Opening and Closing Panes

Right-click anywhere on the toolbar. Each of the panes is listed. Select or un-select a pane to open or close it. In addition each pane can be closed using the close button displayed on its title bar.

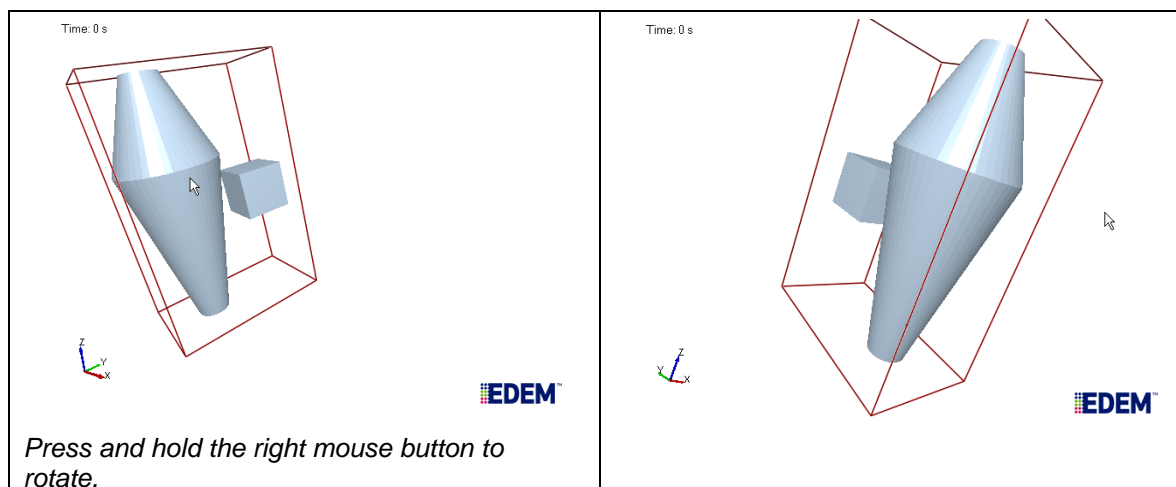
Camera and Mouse Actions

Use the mouse (with modifier keys) to move the camera around the model in the viewer. You can pan, track, zoom and rotate the camera. Select Tools > Options and then the Mouse Configuration tab to configure these controls. See the Creator Tools menu on page 19 for details.

Pan



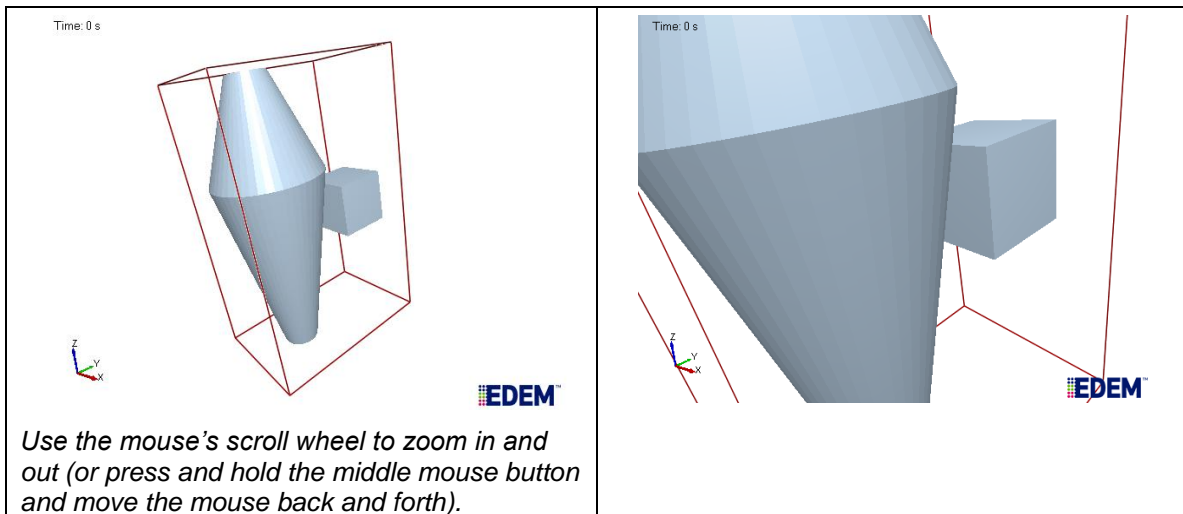
Free Rotate



Track

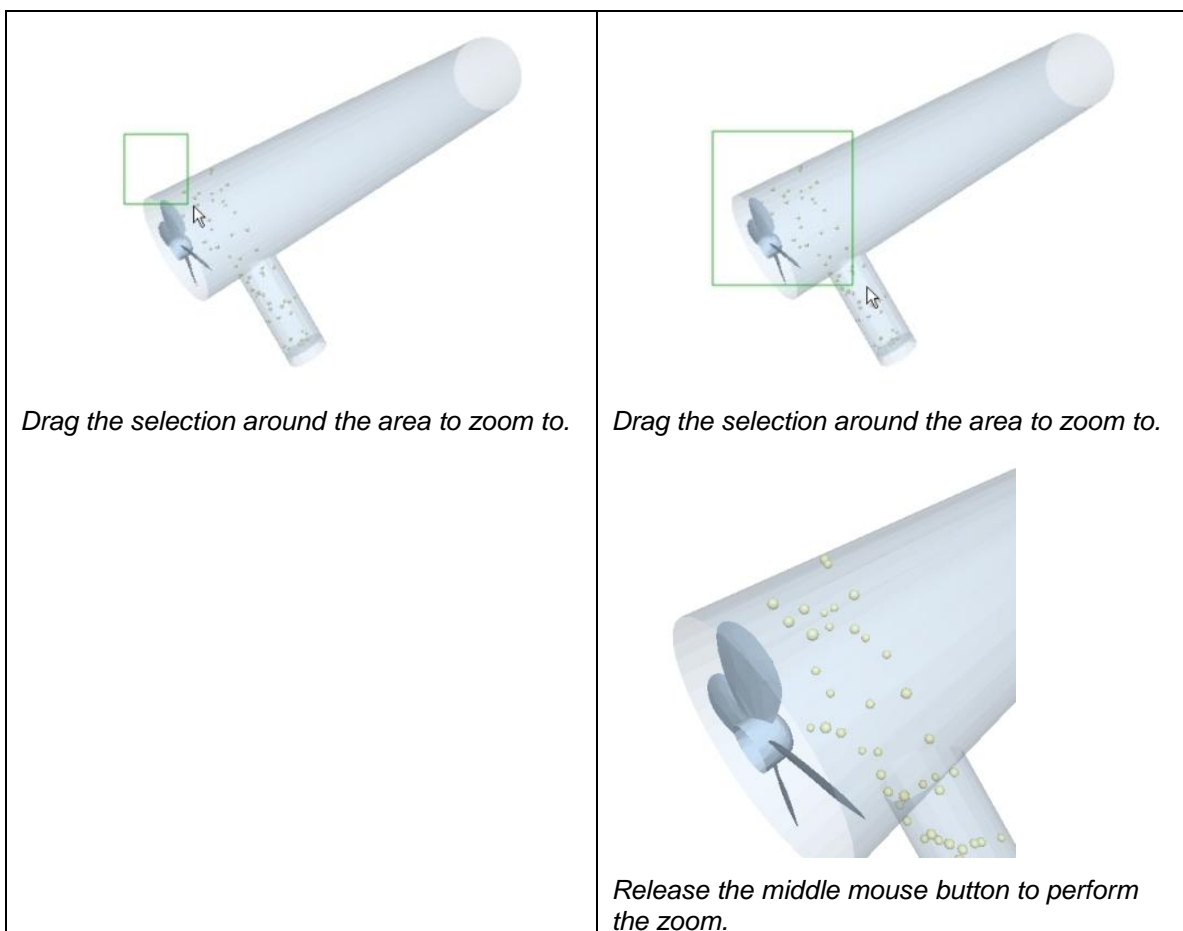
Track lets you move the camera while keeping a fixed distance from the target. This can be useful to track objects that may be hidden behind other objects. Usually, when you zoom in to an object the camera stops at the target position. To maintain the same distance between the camera and the target, hold down the shift key then use the left mouse button to track.

Zoom



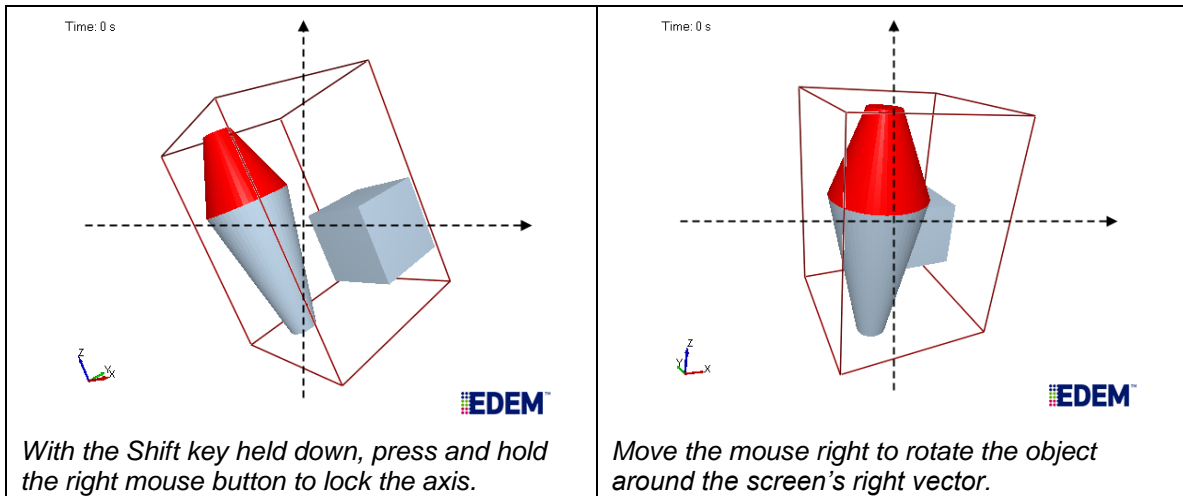
Zoom to Box

Alternatively, while holding down the Ctrl button, press and hold the middle mouse button to draw a box to zoom to:



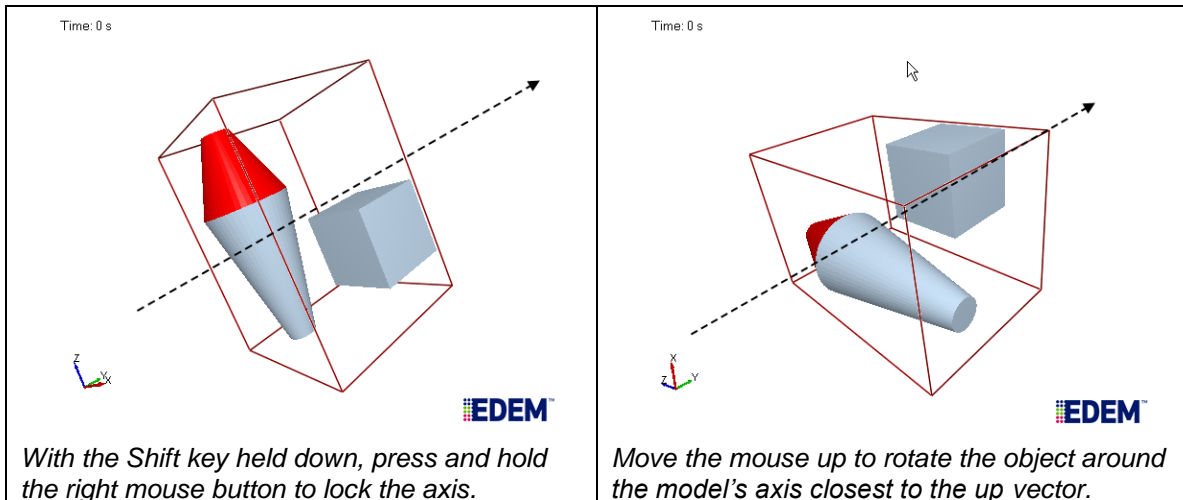
Rotate on Axis: Screen Axis

Select Tools > Options > Mouse Configuration and select to rotate on the Screen Axis. The screen's axes are shown below as dotted lines.



Rotate on Axis: Model Axis

Select Tools > Options > Mouse Configuration and select to rotate on the Model Axis. The model's right axis is shown below as a dotted line.



Choosing Units

The units of measurement used throughout EDEM can be configured. By default all quantities are measured in standard SI units (Système International d'Unites).

Changing Units

Select Tools > Options > Units to change the units. Changes are propagated throughout the software*. You can set units individually. Alternatively, all units can be set to CGS (centimeter-gram-second) or back to the default SI.

Property	Units available	SI unit	CGS unit
Acceleration	mm/s ² , cm/s ² , m/s ² , in/s ² , ft/s ² , fpm/s	m/s ²	cm/s ²
Angle	rad, deg	rad	rad
Angular Acceleration	rad/s ² , deg/s ²	rad/s ²	rad/s ²
Angular Velocity	rad/s, deg/s, rpm	rad/s	rad/s
Charge	nC, C	C	C
Density	g/cm ³ , kg/m ³ , lb/in ³ , lb/ft ³ , slug/ft ³	kg/m ³	g/cm ³
Energy	J, erg, kwh, btu, ft-lbf, in-lbf	J	erg
Force	N, dyn, kgf, lbf, ozf, pdl, gf	N	dyn
Frequency	Hz, kHz, MHz	Hz	Hz
Heat Flux	W, J/S	W	W
Length	mm, cm, m, in, ft	m	cm
Mass	mg, g, kg, lb, oz	kg	g
Mass Generation Rate	mg/s, g/s, kg/s, lb/s, oz/s, mg/min, g/min, kg/min, lb/min, oz/min, kg/h, uk ton/h, us ton/h, metric tonne/h	kg/s	g/s
Moment of Inertia	lb/ft ² , kg/m ² , g/cm ² , lb/in ² , slug/ft ²	kg/m ²	g/cm ²
Shear Modulus	pa, ksi	pa	pa
Stiffness	N/m, lb/ft	N/m	N/m
Stress	pa, Nm ² , lb/in ²	pa	pa
Temperature	K, °C, °F	K	K
Time	s, min	s	s
Torque	Nm, Dyne.cm, gf-cm, kgf-m, lbf-in, lbf-ft	Nm	Dyne.cm
Velocity	mm/s, cm/s, m/s, in/s, ft/s, ft/min	m/s	cm/s
Volume	mm ³ , cm ³ , m ³ , in ³ , ft ³ , L	m ³	cm ³
Work Function	J, eV	J	J

* **Note:** Graphs are not automatically updated when units are changed. Once units have been changed, click the 'Create Graph' button to regenerate graphs.

Managing Licenses

EDEM uses a software license management system called Sentinel™ RMS to manage licenses. As part of the EDEM installation process, licensing software is installed either on the same machine as EDEM or on a license server machine on your network.

Specifying the License Server to Use

When you start EDEM for the first time, a popup window is displayed for you to specify which license server to use.

- Select a license server (or click Refresh to refresh the list) then click OK.
- To change to a different license server, start EDEM using the License Server Reset option from the Start menu.

You can also specify the license server from the command line. For example, to start EDEM and use a license server called HERACLES, type the following command:

```
edem.exe --server HERACLES
```

To view the popup window with the list of license servers, from the command line type:

```
edem.exe --server
```

License Features

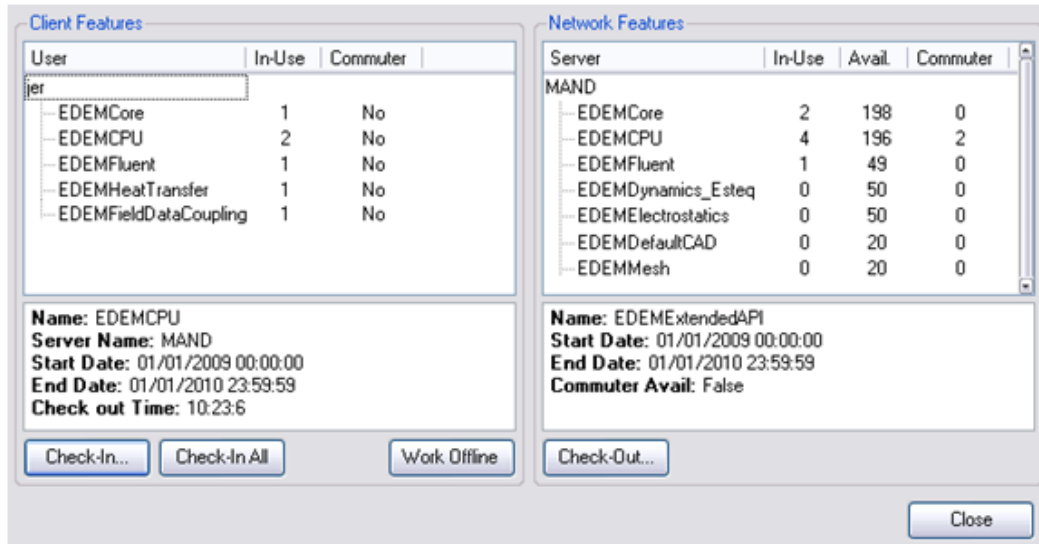
An EDEM license consists of a number of license *features*. These determine which EDEM features are available to you. Every license has at least two features: an EDEMCore and an EDEMCPU feature. The EDEMCore feature lets you start and use EDEM. The EDEMCPU feature controls how many processors are available to use to run a simulation. Depending on what was purchased, your license server may also have other features.

EDEM communicates with Sentinel RMS automatically whenever it needs a license feature. You can also reserve features in advance using the *EDEM License Manager*.

EDEM License Manager

The EDEM License Manager lets you reserve (“Check-Out”) license features as well as work offline (when you have purchased the commuter feature) to use EDEM when disconnected from the license server.

To view the EDEM License Manager, select Tools > License Management:

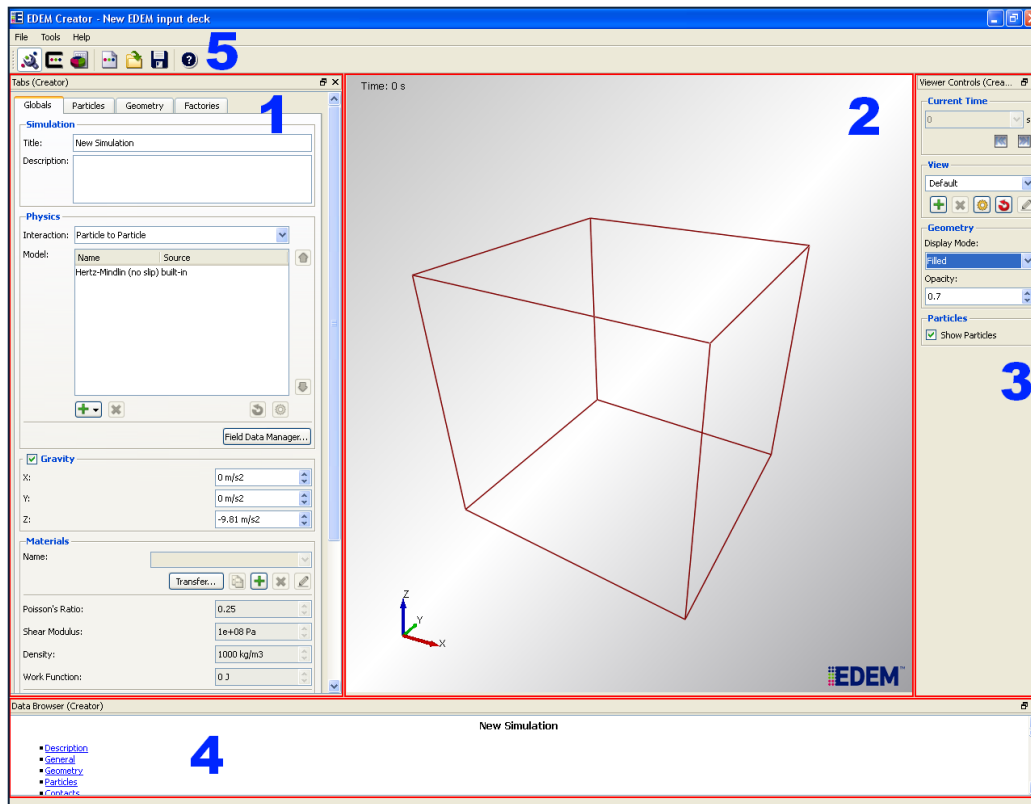


License features available from the license server are shown in the Network Features area on the right. Local features currently being used are shown in the Client Features area on the left. For each license feature, the EDEM License Manager shows how many are available, how many are currently being used, and how many are commuted. For any selected feature, the lower panes show the start date, expiry date, host name and check out time (if applicable).

EDEM Creator

About the EDEM Creator

The EDEM Creator is used to setup and initialize your model. Create or import particles and geometry and define the other model parameters.



- 1 Tabs Pane
- 2 Viewer
- 3 Viewer Controls
- 4 Data Browser
- 5 Toolbar and Menu bar

Tabs Pane

The Tabs Pane is displayed at the on the left side of the EDEM window. It contains four tabs: Globals, Particles, Geometry, and Factories.

Viewer

The Viewer displays 3D representations of your particles, geometry, and fields (if applicable). The rotation, position and zoom factor of the Viewer are controlled using the mouse.

Viewer Controls (Creator)

The Viewer Controls are used to determine how items are displayed in the Viewer. The content of the pane varies according to which tab in the Tabs pane is currently active.

Current Time

Lists all full time steps in the simulation. The selected time step is displayed in the Viewer.

View

Controls to change, add, and configure camera views.

- Select a view from the pulldown menu to display the Viewer from a number of different angles.
- To create a custom view, position your model as required then click the **+** button. Click the configure button to modify the view's projection, lighting, and other settings. Click the view's name to rename it, or click the **x** button to delete.
- For any view, if you manually change the angle or view configuration you can return to the original angle and settings by clicking the reset button.
- Click the configure button to open the Configure View window:

Setting	Description
Projection	<p>Set the projection type to either Perspective or Orthographic.</p> <ul style="list-style-type: none"> ▪ <i>Perspective Projection</i> displays objects as they would appear in the real world: objects that are further appear smaller. ▪ <i>Orthographic Projection</i> maps objects without affecting their relative sizes. Unlike perspective projection, with orthographic projection the size of the viewing area does not change from one end to the other. In other words, the distance from the camera does not affect how large an object appears. This is useful in architectural and CAD applications where the final image needs to reflect the measurements of objects, rather than how they might look.
Field of View	For perspective projection, set the angular field of view.
Camera Position	Specify the X, Y, and Z coordinates to determine the position of the camera.

Target Position	The target is the location of the point the camera is looking at. Specify the X, Y, and Z coordinates to determine the position of the target. Click the Domain Center button to place the target at the center of the domain.
Up Vector	The Up Vector indicates to the camera which way is up. Specify the camera's Up Vector using either world coordinates or by using the dial control.
Specular Reflection	EDEM's headlight is a constant lighting effect shining from the camera. It combines with a specular reflection technique to shade displayed objects. When the headlight is perpendicular to a flat geometry plane, the plane can become too shiny to be distinguished from the background. Click the checkbox to enable or disable the headlight's specular reflection. Specular reflection is disabled by default for all views except the Default view.
Import/Export	Allows customized camera settings created in one simulation deck to be saved for importing into future simulations. Use the Export button to save camera settings to a file (no file extension is required). Use the Import button to load all custom views in a file. Custom views can be imported into the Creator, Simulator, and Analyst, but must be imported into each section individually.

Geometry

- **Display mode.** Configure the display of all the geometry in the Viewer. Choose between filled, mesh or points style display.
- **Opacity.** Configure the opacity level of all the geometry in the Viewer.

Particles

- **Show Particles.** Display particles already created by the Simulator.
- **Show Scale Grid.** Enable or disable the display of a grid around the particle.
- **Show Origin.** Displays the origin. All multi-sphere particles must be centered on the origin.
- **Highlight Surface.** Highlight the currently selected particle surface.
- **Show Template.** Used to display particle templates in the Viewer.

Fields

- **Display field.** Select a field from the pick-list then tick to enable or disable display of the field in the viewer. Vector fields are displayed as arrows; scalar fields are displayed as points.
- **Color.** Select a color to display the field as. The default is Blue.
- **Scale.** Set the length of the field's vector line.

Factories

- **Display Lattice.** Enable or disable the display of the particle positioning lattice. This option is only available when cubic, BCC or FCC lattice positioning is selected.

Data Browser (Creator)

The Data Browser is an .html page that displays detailed information about your model. The information displayed differs between the Creator, Simulator and Analyst. Right click in the Data Browser and choose Save to save the information in an .html file.

There are five sections in the Data Browser within the Creator:

Description

The description as defined in the Globals tab.

General

- **Dimensions.** The number of dimensions in your domain.
- **Gravity.** Details of the gravity acting in your model as specified in the Globals tab.
- **Materials.** A list of all materials used in the model and their properties, as specified in the Materials Editor.
- **Energy.** The total energy in the model at the current time. It is the sum of the kinetic energy, potential energy and the energy in any contacts taking place.

Geometry

- **Domain.** The dimensions of the domain as specified in the Geometry tab.
- **Geometry Totals.** Details of the number of elements that make up each section of geometry.
- **Sections.** Each section in the model and its properties are listed. Properties are as specified in the Geometry tab.








Particles

- **Particles.** A list of each particle type used in the model and their properties. Details of each surface within the particle are also listed. Properties are as specified in the Particles tab.
- **Factories.** Details of each factory used in the model and their properties. Properties are as specified in the Factory tab. The total number of particles created by each factory is also listed.
- **Particle totals.** The total number of particles in the model at the current time. Totals for each particle type are also listed.

Contacts

- **Interactions.** A list of the interaction types taking place in the model as specified in the Materials Editor.
- **Contacts.** Lists the total number of contacts in progress at the current time. The contacts are broken down into contact types: For example number of particle A - particle B contacts or number of particle B - surface A contacts. The total number of collisions that have taken place during the time step are also listed.

Creator Toolbar

Icon	Name	Description
	Creator	Click to switch to the Creator.
	Simulator	Click to switch to the Simulator.
	Analyst	Click to switch to the Analyst.
	New	Click to start a new model.
	Open	Click to open an existing model.
	Save	Click to save your model.
	Help	Click to view the online help.

Creator Menu Bar

File Menu

- **Open.** Open an existing model
- **New.** Start a new model.
- **Save.** Save the current model.
- **Save As.** Save the current model with a specific name or to a specific location.
- **Creator.** Select to switch to the Creator.
- **Simulator.** Select to switch to the Simulator.
- **Analyst.** Select to switch to the Analyst.
- **Recent Files.** A list of recently opened files. Select a file to open it.
- **Quit.** Quit EDEM.

Tools Menu

Materials Database

Access the Materials Database. See *The Materials Database* on page 28.

License Management

Access the License Management tools to reserve licensable features and manage commuter licenses. See *Managing Licenses* on page 13.

Options

Access the global options dialog containing settings for the following items.

Particle Display

- **Display Templates.** Mesh files created in other packages can be used as particle templates. Templates can be used as a guide (outline) to help define the shape of your particle in the Creator as well as provide, if desired, the shape for calculation of the center of gravity and mass moments of inertia, or as a display option in the Analyst.

Refer to *Appendix C: EDEM File Types* on page 122 for a list of file-types that can be imported. When a model containing a particle template is saved, an extra file (.ptf) is created alongside the usual three files (.dem, .dfg and .idx). If this file is deleted, the template information will no longer be stored within the model.

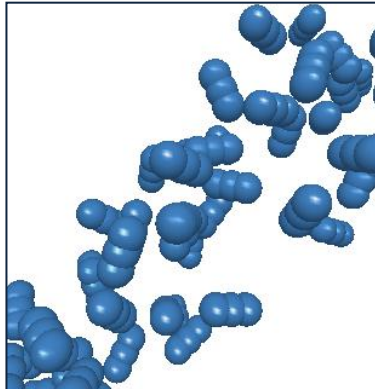


Figure 2-1: Default Particle Display

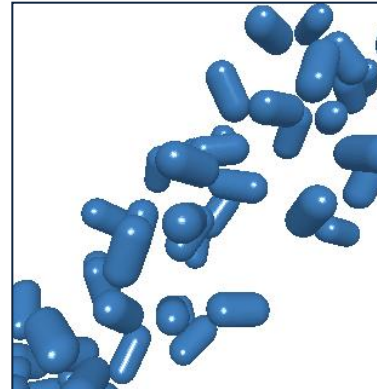


Figure 2-2: Particle Display Using a Template

Mouse Configuration

Used to configure how the mouse is used to alter the display in the Viewer.

- **Camera Actions.** Change the mouse button and modifier key used to pan, track, zoom and rotate the camera. Modifier keys assign keyboard buttons to press to perform the specified camera action. Choose any unique combination of three mouse buttons and three keys for each motion.
- **Rotate on Axis.** Change the mouse button and modifier key used to rotate on an axis. You can also set whether to rotate on the model axis or the screen axis for both the horizontal and vertical rotations.
- **Mouse Sensitivity.** The mouse sensitivity can be altered independently for tracking and zooming. You can also specify modifier keys for each sensitivity setting.

See *Camera and Mouse Actions* on page 9 for more information.

File Locations

The file location options are used to determine the location of the materials database and any user-defined elements.

- **Materials Database.** The location of the materials database, used to define any material used in a model. See *The Materials Database* on page 28.

- **Contact Models.** The location of custom (user-defined) contact models. See the user section of the DEM website.
- **Particle Body (External) Force.** The location of custom (user-defined) forces. See the user section of the DEM website.
- **Particles.** The location of particles saved independently of the model for reuse in other projects.
- **Factories.** The location of any user-defined factories.

Background

This option is used to change the fill color of the Viewer background. The background can be colored with a solid or gradient fill. The color change will affect the Creator, Simulator and Analyst.

Units

The units configuration dialog is used to set the measurement units used throughout EDEM. See *Choosing Units* on page 12 for more details.

Data Browser

The data browser options are used to configure the color of the data browser text and background.

Analyst Options

- **Timestep:** Models can be loaded into the Analyst with the first or last time step appearing first.
- **Axis Key:** Turn the display of the axis key in the Viewer on or off.
- **Bounding Box:** Turn the display of the boundary box (marking the domain edges) in the Viewer on or off.
- **Timestamp:** Turn the display of the time stamp in the Viewer on or off.

Global Settings

Simulation

Set the simulation title and a description of the simulation.

Physics

Interaction

Select the type of physics interaction to add:

- Particle-to-Particle contact models
- Particle-to-Geometry contact models
- Particle Body Forces

Contact Models

A contact model describes how elements behave when they come into contact with each other. Using the Interaction pulldown menu and + pick-list, you can build a list (stack) of contact models. The top element is applied first, then the next one down and so on. Use the up and down buttons to move the contact model up and down the list (check the sections below since some models need to be at the start or end of the list). To remove an item, click the X button. Click the preferences button to configure the selected model. Every simulation must have at least one base particle-to-particle and one particle-to-geometry contact model. EDEM is supplied with several integrated contact models; you can also add your own custom plug-in contact models. Refer to the user section of the DEM website and the *EDEM Programming Guide* for details.

Hertz-Mindlin (no slip)

Hertz-Mindlin (no slip) is the default contact model.

Interaction	Configurable Parameters	Position
Particle to Particle, Particle to Geometry	None.	Last

Hertz-Mindlin (no slip) with RVD Rolling Friction

Hertz-Mindlin (no slip) with RVD Rolling Friction calculates contact forces in the same way as the Hertz-Mindlin (no slip) model, but offers an alternative implementation of rolling frictions at contacts. Since this model includes Hertz-Mindlin, remove the default no-slip contact model from the list.

Interaction	Configurable Parameters	Position
Particle to Particle, Particle to Geometry	None.	Last

Hertz-Mindlin with Heat Conduction

Hertz-Mindlin with Heat Conduction calculates the heat flux between particles in contact. The model calculates a heat flux based on the particle temperatures and their overlap. This model must be used with Update Temperature particle body force to complete the calculation. The Update Temperature particle body force allows the inclusion of an external heat source and is where each particle temperature is calculated based on the heat flux calculated in the contact model and a given external heat flux.

Make sure the Update Temperature particle body force model is added to the particle body force model. Since this model includes Hertz-Mindlin, remove the default no-slip contact model from the list.

Interaction	Configurable Parameters	Position
Particle to Particle	Set the thermal conductivity for each type of particle. Note the unit depends on the temperature unit; for example, the	Last

Interaction	Configurable Parameters	Position
	unit will be W/mK when the temperature unit is Kelvins.	

Hertz-Mindlin with Bonding

Hertz-Mindlin with Bonding bonds particles together. Since this model includes Hertz-Mindlin, you should remove the default no-slip contact model from the list.

To use, enable the bonding model for both particle-particle and particle-geometry interactions. Although particles cannot be bonded to geometry elements, the bonded model ensures that the particles contact the geometry based on the particle’s physical radius and not the contact radius.

Interaction	Configurable Parameters	Position
Particle to Particle <i>(also enable for Particle to Geometry)</i>	<p>Select an active bond then set the Bond Formation Time and the following parameters:</p> <ul style="list-style-type: none"> ▪ <i>Normal Stiffness</i>: the tensile/compressive stiffness along the bond’s principal axis ▪ <i>Shear Stiffness</i>: Shear stiffness in the orthogonal plane to the bond’s principal axis ▪ <i>Critical Normal Stress</i>: the maximum normal stress the bond can withstand before it fails ▪ <i>Critical Shear Stress</i>: The maximum tangential stress the bond can withstand before it fails ▪ <i>Bonded Disk Radius</i>: The radius of the cylindrical bond between the particles <p>Set the bond stress and stiffness values to represent the material that is been modeled. A high stiffness value will produce high bond forces and stresses. A lower than normal timestep may be required to accurately capture these high forces.</p> <p>When the Bond Formation Time is reached, all defined particles in contact will be bonded together. Before this time, particles interact based on the Hertz-Mindlin contact model.</p> <p>The bond formation time can be updated to restart the bonding process. Existing bonds won't be affected, but any unbonded particles in contact will be bonded whenever the updated bond formation time is reached.</p> <p>Note that a bond between two particles will cease to exist whenever one or both of the bonded particles leaves the simulation domain. If periodic boundaries are applied when the particle leaves the domain, then the particle will exist on the opposite side of the domain; in this case the two particles will remain bonded across the periodic boundary.</p>	Last


Hertz-Mindlin with JKR Cohesion

Hertz-Mindlin with JKR (Johnson-Kendall-Roberts) *Cohesion* is a contact model that allows users to represent the cohesive nature of fine and moist materials. Originally implemented to allow the simulation of Van der Waals forces which influence the flow

behavior or fine, dry powders. This model may also be used to recreate the influence that a moisture content has on the bulk flow of larger-scale materials such as iron ore or wet grains. Remove the Hertz-Mindlin (no slip) contact model from the list when using this model.

Interaction	Configurable Parameters	Position
Particle to Particle, Particle to Geometry	Click + to add cohesion to particle-particle or particle-geometry interactions. Set the surface energy for each interaction.	Last

Use


1. Select the required category from the Interaction pulldown in the Physics section of the Creator.
2. Click the **+** drop-down list then select *Hertz-Mindlin with JKR Cohesion*.
3. Click the configuration button  to define contact model parameters.

Linear Cohesion

Linear Cohesion is a cohesion contact model that modifies the default Hertz-Mindlin contact model for particle-particle and particle-geometry interaction by adding a normal cohesion force.

Interaction	Configurable Parameters	Position
Particle to Particle, Particle to Geometry	Click + to add cohesion to particle-particle or particle-geometry interactions. Set the energy density for each interaction.	Any

Use

1. Select the required category from the Interaction pulldown in the Physics section of the Creator.
2. Be sure a model for contact forces is listed in the Model section. If not, click the **+** drop-down list then select (for example) *Hertz-Mindlin (no-slip)*.
3. Click the **+** drop-down list then select *Linear Cohesion*.
4. Click the configuration button  to define contact model parameters.


Linear Spring

Linear Spring is a damped linear spring contact force model. Remove the default Hertz-Mindlin (no-slip) contact model from the list when using this model.

Interaction	Configurable Parameters	Position
Particle to Particle, Particle to Geometry	Set the characteristic velocity of the spring. Be sure the velocity is the same for both particle-to-particle and particle-to-	Last

Interaction	Configurable Parameters	Position
	geometry.	

Use

1. Select the required particle interaction from the Interaction pulldown in the Physics section of the Creator.
2. Click the + drop-down list then select *Linear Spring*.
3. Click the configuration button  to define contact model parameters.

Hysteretic Spring

The *Hysteretic Spring* model allows plastic deformation behaviors to be included in the contact mechanics equations, resulting in elastic-plastic deformation behavior. Unlike the Hertz-Mindlin and Linear Spring models, which are elastic contact models, with the Hysteretic Spring model particles behave in an elastic manner up to a predefined stress. Once this stress is exceeded, the particles behave as though undergoing plastic deformation. The result is that large overlaps are achievable without excessive forces acting upon them, thus representing a compressible material.


When using this model the user should be aware that the coefficient of restitution parameter set in the interaction section of general tab is now used in an entirely different energy dissipation method compared to default Hertz-Mindlin model. The coefficient of restitution here determines the fraction of energy stored during loading energy which can be recovered upon unloading, Lower values of this parameter provide higher residual deformation in material under confinement.

Remove the default Hertz-Mindlin (no slip) contact model from the list when using this model.

Interaction	Configurable Parameters	Position
Particle to Particle, Particle to Geometry	Click + to add particle-particle or particle-geometry interactions. Set the following interaction parameters: <ul style="list-style-type: none"> • Damping factor: this dimensionless parameter controls the amount of velocity dependent damping. Without this additional damping, small vibrations of particles can persist for a long time. Setting this parameter to zero suppresses only the velocity-dependent damping without affecting the main energy loss mechanism due to the nature of the linearized elasto-plastic model. The recommended value for this parameter is zero when modeling “aggressive” compression of materials and small non-zero value for small strain compression regimes. • Stiffness factor: This dimensionless parameter is defined as the ratio of tangential stiffness to normal loading stiffness and used in calculation of tangential forces at the contact. According to the literature, the typical value of this parameter is in the range between 	Last

Interaction	Configurable Parameters	Position
	0.7 and 1. Also set the yield strength in units of Pa for each particle or geometry that has an interaction. EDEM offers a reasonable default value for this parameter estimated from material Shear Modulus. The user can, however, overwrite this default value.	

Use


1. Select the required category from the Interaction pulldown in the Physics section of the Creator.
2. Click the + drop-down list then select *Hysteretic Spring*.
3. Click the configuration button  to define contact model parameters.

Moving Plane

The *Moving Plane* model simulates the in-plane motion of a geometry section when contacted by a particle.

Interaction	Configurable Parameters	Position
Particle to Geometry	Click + to add a section to use as a moving plane. For each section, set the X, Y, and Z linear velocity.	First

Use

1. Select Particle-to-Geometry from the Interaction pulldown in the Physics section of the Creator.
2. Be sure a model for contact forces is listed in the Model section. If not, click the + drop-down list then select (for example) *Hertz-Mindlin (no-slip)*.
3. Click the + drop-down list then select *Moving Plane*.
4. Click the configuration button  to define the sections to use as moving planes. For each section, set the X, Y, and Z linear velocity.

Particle Body Forces

A particle body force can be set to act on particles when a specified condition is met, for example when particles are in certain position or are traveling at a specified velocity. Refer to the user section of the DEM website and the *EDEM Programming Guide*.

Using the Interaction pulldown menu and + pick-list, you can build a list (stack) of particle body forces. The top element is applied first, then the next one down and so on. Use the up and down buttons to move the force up and down the list. To remove an item, click the X button. Click the preferences button to configure the selected force.

Configuring Particle Body Forces

Name	Description and Parameter Values
<i>Temperature Update</i>	Although not strictly a force, this built-in model allows the modeling of model external heat sources acting on the particles. It has to be used with the Hertz-Mindlin with Heat Conduction contact model as it also completes the calculation of heat transfer between particles in contact Set the specific heat capacity for each type of particle. Note the unit depends on the temperature unit; for example, the unit will be $\text{Jkg}^{-1}\text{K}^{-1}$ when the temperature unit is Kelvins.

Use

1. Select Particle-to-Particle from the Interaction pulldown in the Physics section of the Creator.
2. Click the **+** drop-down list then select *Hertz-Mindlin with Heat Conduction*.
3. Click the configuration button to define the thermal conductivity for each particle type.
4. Select Particle Body Force from the Interaction pulldown.
5. Click the **+** drop-down list then select *Temperature Update*.
6. Click the configuration button to define the specific heat capacity for each particle type.

Gravity

Enable the gravity checkbox to set gravity in a given direction (x, y, z). The default settings are x: 0, y: 0 and z: -9.81m/s^2 .

Materials

Every particle or section of geometry used in a model is made of a particular material. All materials and the interactions between them must be defined in the Materials section. Materials can be created directly in the simulation or imported from the Materials Database. All materials defined in the Materials section are stored within the model.

Adding and Removing Materials

To add a material to a model:

- Click the **+** button then type a name for the new material in the Name field.
- Enter the Poisson's ratio, Shear modulus, Density and Work Function (if applicable) into the relevant fields.

You can also click Copy to create a duplicate of the currently selected material. To remove a material from a model, select the material from the drop-down list then click the **X** button.

Defining Interactions Between Materials

Interactions are used to define how materials act when they come into contact with each other. Interactions must be defined for all materials used in your model, including the interaction that occurs when a material comes into contact with itself: For example when two particles of the same material collide.

To define an interaction between two materials:

- Select the first material from the Name drop-down list.
- Click the **+** button in the Interactions section.
- Select the second material from the Select Material dialog.
- Enter values for the Coefficient of Restitution, Coefficient of Static Friction and Coefficient of Rolling Friction into the relevant fields.

The Materials Database

The EDEM Materials Database is a repository for material and interaction data. It stores material data in one central location. The material data can then be imported into simulations as desired.

The Materials Database supplied with EDEM contains three example materials. These represent materials that have:

- Low-stiffness (such as rubber)
- Medium-stiffness (such as plastic)
- High-stiffness (such as steel)

By default, the Materials Database (MaterialsDB.ddb) is installed to the My Documents > EDEM folder. If you move the Materials Database (for example to a shared network folder for use by multiple clients), remember to update the location information in EDEM. To do this, select Tools > Options > File Locations then browse to the new location.

Adding Materials and Material Interactions to the Database

Materials and material interactions can be added to the database directly or transferred from an existing simulation.

Note that materials defined in your database are not automatically added to any model. All materials used in a model must be defined directly in that model or explicitly transferred from the materials database.

Adding and Removing Materials

To add a material to the database:

- Select Tools > Materials Database from the Creator to open the Materials Database.
- Click the **+** button and type a name for the new material in the Name field.

- Enter values for the Poisson's ratio, Shear modulus, Density and Work Function (if applicable) into the relevant fields.

To remove a material from the database:

- Open the Materials Database.
- Choose the material from the drop-down list and click the **X** button. The material is permanently removed from the database.

Click the Copy button to create a duplicate of the currently selected material.

Importing Materials from other Databases

Materials can be imported from other EDEM Material databases.

- Open the Materials Database.
- Click the Import button and navigate to the Materials Database (.ddb) of your choice.
- Click Yes to import all materials and interactions. Click No to import a limited selection of the database.

Transferring Materials to and from the Materials Database

Any material or interaction defined in a model can be transferred to the materials database for use in other simulations. Similarly, materials and interactions can be imported from the database into your model.

To transfer a material to or from the database:

- Click the Transfer button. The Transfer dialog box appears.
- Click on the material to transfer. The material's properties are displayed at the bottom of the dialog.
- Click the appropriate arrow to transfer the material to or from the database.

Conflicts

If a material in the model and one in the database have the same name but different properties they are highlighted in red to indicate the conflict. If you continue with the transfer, one material will overwrite the other.


If two materials have the same name and the same properties they are both highlighted in green.

Defining Particles

All types of particle used in a model are defined in the Particles pane. These are the base or prototype particles. A model can contain any number of different particle types.

Select Particle

Creating or Removing Particle

- To create a particle click the **+** button and type a name in the Name field. A single spherical surface is automatically created.
- To copy an existing particle, select it from the drop-down list and click the  button.
- To remove a particle, select it from the drop-down list and click the **X** button.

Importing a Particle

Any particle created in EDEM can be exported for use in other models. To import a particle click the Import button and navigate to your chosen file.

Using a Particle Template

Mesh files created in other packages can be used as particle templates. A template can be used as a guide to create your particle and its properties automatically calculated from it. Refer to *Appendix C: EDEM File Types* on page 122 for a list of file-types that can be imported. We recommend using a template if your model uses multi-sphere particles.

To use a template:

- Go to the Tools > Particle Display menu.
- Click the Import button and navigate to the file of your choice. Any templates used in your model must first be imported here.
- Select the Show Template option in the Viewer controls pane at the right of the screen and choose your template from the drop-down list.
- Create surfaces to match the template outline.

When a model containing a particle template is saved, an extra file (.ptf) is created alongside the usual three files (.dem, .dfg and .idx). If this file is deleted, the template information will no longer be stored within the model.

Surfaces

A particle is defined using one or more spherical surfaces. Multiple surfaces can be overlapped to create multi-sphere particles.

- Click the **+** button then type a name for the surface in the Name field.
- Type the surface Radius.
- Select Contact Radius when the particle has a different contact radius due to another (long-range) force acting upon it. This option should only be used with a relevant user-defined contact model. When two contact radii overlap the contact force calculation is entered into, however internally the physical radius of the spheres is used to calculate the magnitude of the contact force.
- For multiple surfaces, use the Position X, Y and Z options to define their relative positions. Note that multi-sphere particles should be centered on the origin along the principle axes to ensure that the moments of inertia are calculated correctly.

Properties

Particle properties refer to the whole particle, not just to individual surfaces. Particle properties can be inserted manually or calculated automatically.

Automatically

- Click Calculate Properties then select whether to calculate using the spherical surfaces that make up your particle, or (preferably) from an imported template.
- Check whether you want to automatically center the particle when calculating the moments of inertia. Unchecking this option may result in unphysical results.
- Click OK. EDEM calculates the mass, volume, and moments of inertia.

Since random sampling is used to calculate particle properties, re-calculating for particles with multiple surfaces may result in slightly different values.

Manually

Type values for each property into the relevant field. If a value turns red it is not recommended.

Property	Description
Material	Choose a particle material. The drop-down list is populated with any material defined in the Globals pane
Mass	Mass of the particle.
Volume	Volume of the particle measured.
Moment of Inertia X/Y/Z	The moment of inertia is a measure of a body's resistance to angular acceleration. A particle consists of a number of spheres, i , each with mass m_i and radius r_i . If each sphere is a distance d_i from a particular axis of rotation, then the moment of inertia of the particle about that axis is given by:

Particle Limits

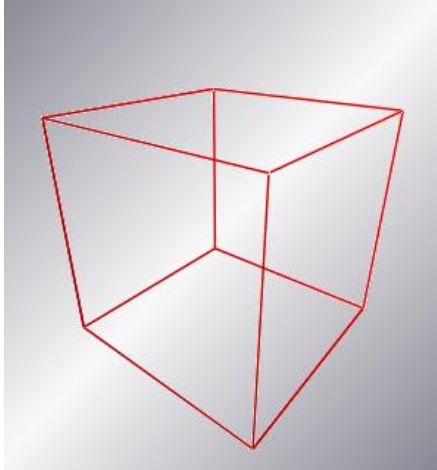
Under certain simulation conditions the velocity of a particle can become too large and cause surrounding particles to behave erratically: For example, when the simulation time step is too large. There are two ways to help combat this. Particle velocities can be capped at a specified limit or particles exceeding that limit can be removed from the simulation. Particle velocity and angular velocity can be treated separately.

Particle Export

Once a particle has been defined it can be exported for use in other EDEM simulations. Click the Export button and save the file to your Particles folder (as specified using Tools > Options > File Locations). Files are exported in the .dem format.

The Model Domain

The model domain is the area where simulation takes place. It is defined in the Geometry tab and indicated by the red box in the Viewer:



Particles that move out of the domain during the course of the simulation are permanently removed by the Simulator (unless a periodic boundary option has been selected). Geometry can move in and out of the domain during a simulation.

Defining the Model Domain

Domain size has an effect on simulation time - the larger the domain, the longer the simulation will take to run. Use the auto-update option to automatically fit the domain around any sections of geometry defined, thus creating the smallest domain possible.

Note: Sections of geometry should not be placed totally coincident with the edge of the domain. Particles are removed as soon as they reach the domain edge, so cannot interact with any geometry placed there.

Periodic Boundaries

Periodic boundaries enable you to set what happens to a particle once it leaves the domain. If the option is turned on for a particular direction, any particle leaving the domain in that direction will instantly re-enter it on the opposite side.

Creating Geometry Sections

Geometry sections are used to create the environment in which your particles act: For example, a pipe or a mixing drum. Sections can be defined in EDEM or imported from a CAD (or similar) package. Particle factories are also created using sections.

A model can have any number of sections allowing complex machinery to be created. A section can be defined as a box, a cylinder or as an n-sided polygon. Each section comprises a number of triangular elements: For example a square surface is made up of two elements and a ten-sided polygon of ten elements. Information on these elements can be examined at a later point: For example, contacts with a particular element. The individual elements can be seen by choosing the mesh display mode.

Create a New Geometry Section

To create a new section:

1. Go to the Geometry tab.
2. Click the **+** button then choose the shape. Select box, cylinder or polygon.
3. Type a name for the section in the Name field.

The section should now appear in the Viewer. Note the default size of some sections is quite small and may not be immediately visible in your domain.

Set the basic properties in the Details tab:

Property	Description
Volume	A section is described as a volume or a surface. Completely closed boxes and cylinders are volumes; polygons and open boxes or cylinders are surfaces. This field is completed automatically. Volumes or virtual surfaces can later be turned into particle factories.
Material	Select a section material from the drop-down list. The materials listed are defined using the Materials Section in the Globals pane.
Type	A section is either virtual or physical. A physical section is an actual surface or volume that particles can interact with. A virtual section (used to create particle factories) is a surface or volume of interest that does not actually exist and does not interact with anything in a simulation.
Center of Mass	For every closed-volume geometry (such as a box or closed cylinder), EDEM automatically calculates the center of mass when you first import or create the geometry. If required, set the center of mass parameters for X, Y, and Z components (for imported geometries) or click the auto-adjust checkbox for geometry sections defined in EDEM.

Specify the Section Properties

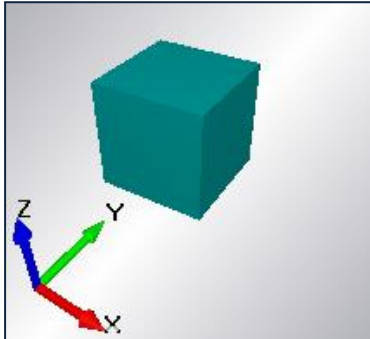
Once a section has been created a third tab (Box, Cylinder or Surface) will appear. Click on the tab and define size, shape and position of your section. Note geometry sections outside of or directly coincident with the domain boundary do not take part in the simulation, and will hence be 'invisible' to particles.

Section	Parameters
Box	<ul style="list-style-type: none"> ▪ Center. Set the position of the center of the box. ▪ Dimensions. Set the box height, width and length. ▪ Rotation. Set the box rotation about the x, y or z axes.

	<ul style="list-style-type: none">▪ Enable Sides. Any of the six sides of the box can be removed.
Cylinder	<ul style="list-style-type: none">▪ Radius Start. Set the radius of one end of the cylinder. The end can be open or closed.▪ Radius End. Sets the radius of one end of the cylinder. The end can be open or closed.▪ Start Point (x), End Point (x). Set the x-position of each end of the cylinder.▪ Start Point (y), End Point (y). Set the x-position of each end of the cylinder.▪ Start Point (z), End Point (z). Set the x-position of each end of the cylinder.
Polygon	<p>All polygons except a 4-sided polygon are defined as follows:</p> <ul style="list-style-type: none">▪ Edges. Set the number of edges. The minimum number is three.▪ Center. Set the position of the center of the polygon.▪ Rotation. Set the angle of rotation about the surface normal.▪ Surface normal. Define the surface normal to determine the orientation of the polygon. <p>A 4-sided polygon is specified as follows:</p> <ul style="list-style-type: none">▪ Edges. Set the number of edges to four.▪ Center. Set the position of the center of the polygon.▪ Dimensions. Set the length and the width.▪ Rotation. Set the angle of rotation about the x, y or z axes.

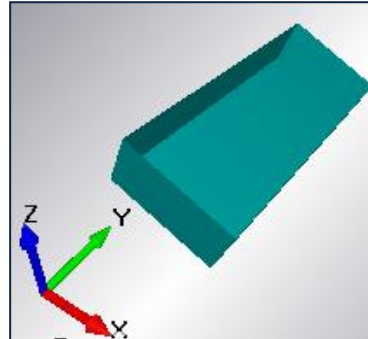
Example Geometry Sections

Box Section Examples



A closed cube

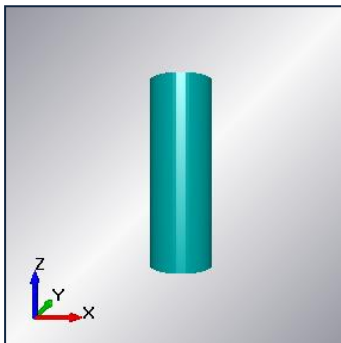
	Center (m)	Dimensions (m)	Rotation (rad)
X	0.5	0.2	0
Y	0.5	0.2	0
Z	0.5	0.2	0



An open, tilted cuboid.

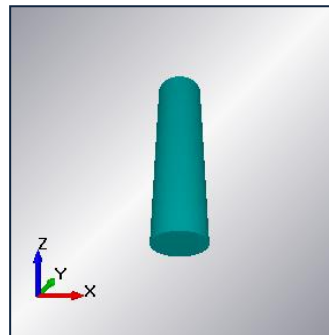
	Center (m)	Dimensions (m)	Rotation (rad)
X	0.5	0.5	-0.5
Y	0.5	0.2	0
Z	0.5	0.1	0

Cylinder Section Examples



A vertical cylinder

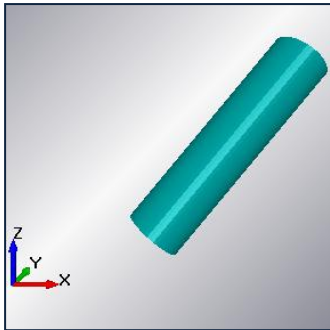
Radius Start:	0.1		
Radius End:	0.1		
Start Point (x):	0.5	End Point (x):	0.5
Start Point (y):	0.5	End Point (y):	0.5
Start Point (z):	0.2	End Point (z):	0.8



A cylinder tilted in the y-direction

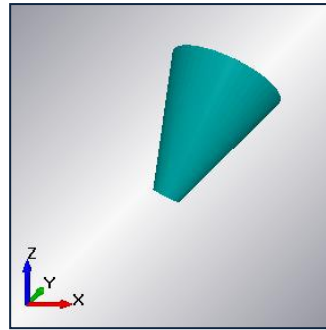
Radius Start:	0.1		
Radius End:	0.1		
Start Point (x):	0.5	End Point (x):	0.5
Start Point (y):	0.5	End Point (y):	1.0
Start Point (z):	0.2	End Point (z):	0.8

Cylinder Section Examples (Continued)



A cylinder tilted in the x-direction

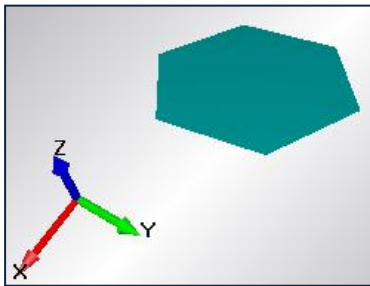
Radius Start:	0.1		
Radius End:	0.1		
Start Point (x):	0.5	End Point (x):	1.0
Start Point (y):	0.5	End Point (y):	0.5
Start Point (z):	0.2	End Point (z):	0.8



A funnel tilted in the x-direction

Radius Start:	0.1		
Radius End:	0.3		
Start Point (x):	0.5	End Point (x):	1.0
Start Point (y):	0.5	End Point (y):	0.5
Start Point (z):	0.2	End Point (z):	0.8

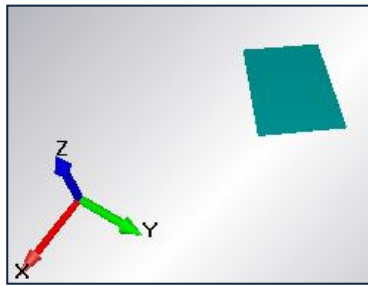
Polygon Section Examples



A 6-sided polygon.

	Center (m)	Surface normal
X	0.5	0
Y	0.5	0
Z	0.5	1

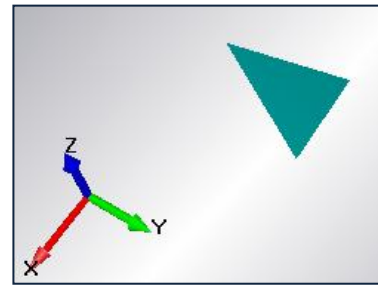
Radius = 0.4m, Rotation = 0 rad



An 4-sided, tilted polygon.

	Center (m)	Rotation (rad)
X	0.5	0
Y	0.5	0
Z	0.5	0.5

Side 1 = 0.5m, Side 2 = 0.3m



An 3-sided, tilted polygon.

	Center (m)	Surface normal
X	0.5	0
Y	0.5	0.5
Z	0.5	1

Radius = 0.3m, Rotation = 0 rad

Specify the Section Dynamics

Sections of geometry can be static or dynamic. Static sections remain in a fixed position during the course of a simulation whereas static sections move. Sections can move under translation or rotation.

Sections are strain limited, they will not move when a force provided by another element is applied. For example, a section will not move when another section comes into

contact with it or when a particle bounces off it. Sections pass through each other when they come into contact.

Specifying a Translation

- Select the section from the Sections drop-down list.
- Go to the Dynamics tab.
- Click the **+** button then type a name for the dynamic in the Name field.
- Set the dynamic type to Linear or Sinusoidal Translation (used to model oscillating motion).

Translation	Set-up
Linear	<ul style="list-style-type: none"> ▪ Set the Start time and End time. This determines the points in the simulation at which the motion will begin and end. ▪ Set the Initial velocity of the section. This determines the speed of the section at the specified start time. ▪ Specify the Acceleration of the section. Acceleration can be set to 0. ▪ Set the direction vector for the motion. This determines the direction the section will move in.
Sinusoidal	<ul style="list-style-type: none"> ▪ Set the Start time and End time. This determines the points in the simulation at which the motion will begin and end. ▪ Set the distance (also known as the amplitude) of the oscillation in each direction. ▪ Specify the frequency of the oscillation. ▪ Specify the oscillation offset: For example, two sections with identical oscillations offset by 180° will travel in opposite directions.

Specifying a Rotation

- Select the section from the Sections drop-down list.
- Go to the Dynamics tab.
- Click the **+** button then type a name for the dynamic in the Name field.
- Set the dynamic type to Linear or Sinusoidal Rotation (used to model oscillating motion).

Rotation	Set-up
Linear	<ul style="list-style-type: none"> ▪ Set the Start time and End time. This determines the points in the simulation at which the rotation will begin and end. ▪ Set the Initial velocity of the section. This determines how fast the section will start rotating at the specified start time.

	<ul style="list-style-type: none"> ▪ Specify the Acceleration of the section. Acceleration can be set to 0. ▪ Set the axis of rotation. This defines around which axis the section will rotate. ▪ Specify the point of action (the center of the rotation). If the geometry section is also moving under a translation, select the Moves With Body option to update center as the section moves. This will ensure that the center of the rotation remains in the same relative position to the geometry section.
Sinusoidal	<ul style="list-style-type: none"> ▪ Set the Start time and End time. This determines the points in the simulation at which the motion will begin and end. ▪ Set the angle of the oscillation in each direction. ▪ Specify the frequency of the oscillation. ▪ Specify the oscillation offset. ▪ Specify the center of the rotation. If the geometry section is also moving under a translation, select the Moves with body option to update center as the section moves. This will ensure that the center of the rotation remains in the same relative position to the geometry section.

Turning a Section into a Particle Factory

Particle factories are used to define where, when and how particles appear in a simulation. Any virtual surface or volume (physical or virtual) can be turned into a particle factory. See *Particle Factories* on page 40 for more details.

Importing Geometry Sections

Rather than defining sections using EDEM, geometry can be imported from a range of other packages. Refer to *Appendix C: EDEM File Types* for details.

1. Click the Import button in the Sections area of the Geometry tab.
2. Navigate to the file to import then click Open. If applicable, the Geometry Import Parameters dialog is displayed.
 - a) Choose to import as a quality or curvature mesh.
 - A quality mesh is usually made up of equilateral facets. It often contains more facets compared with a curvature mesh.
 - A curvature mesh usually matches the contours of a body well, but is often made of non-equilateral facets.
 - b) Set the facet options, depending on the method you selected. Set facet sag and facet length to a percentage of the model diameter, or use the default values. Click OK.

3. The Import Options dialog is displayed.
 - a) Select the unit of measurement the geometry is measured in.
 - b) Tick the Merge Sections checkbox to merge separate geometry sections into one single section.
4. Click OK. You may need to wait some time for the geometry import to complete, especially when you did not select default values in the previous step. Once imported, the geometry is displayed in the Viewer. Each section is listed in the section drop-down list.
5. Complete the Details options. See *Create a New Geometry Section* on page 33.
6. Set the Dynamics (if required). See
7. *Specify the Section Dynamics* on page 36.

Repositioning Imported Geometry Sections

Imported CAD geometry sections can be moved (translated) and rotated. You can also export the translation data to a text file.

- Select the imported section from the Sections drop-down list.
- Go to the Geometry tab then define the transformation as required.
- Tick the Display Bounding Box checkbox to display a blue box around the geometry section, aligned to its axis.

Movement	Parameters
Translation	<p>Set the position of a translation vector to determine where to move the geometry to. The direction of the translation is the vector defined by the start and end points. Once set, specify the distance to move the geometry along the vector.</p> <ul style="list-style-type: none"> ▪ Pick Start Point. Click to enable the target crosshair, then click a point on a geometry in the Viewer to set the start point for the translation. ▪ Pick End Point. Click to enable the target crosshair then click a point on a geometry in the Viewer to set the end point for the translation. ▪ Start Point X, Y, Z. Specify the X, Y, and Z coordinates of the start point of the vector along which the geometry is to be translated. ▪ End Point X, Y, Z. Specify the X, Y, and Z coordinates of the end point of the vector along which the geometry is to be translated. ▪ Distance. Specify the distance to move the geometry along the vector. ▪ Display Vector. Click to view the translation vector.
Rotation	<p>Set the position of a rotation axis to determine how to rotate the geometry. This axis is specified as a vector between two points around which all rotations are performed using a rotation angle. Once set,</p>

specify the angle to rotate the geometry around the axis.

- Pick Start Point. Click to enable the target crosshair, then click a point on a geometry in the Viewer to set the start point for the rotation.
- Pick End Point. Click to enable the target crosshair then click a point on a geometry in the Viewer to set the end point for the rotation.
- Start Point X, Y, Z. Specify the X, Y, and Z coordinates of the start point of the vector along which the geometry is to be rotated.
- End Point X, Y, Z. Specify the X, Y, and Z coordinates of the end point of the vector along which the geometry is to be rotated.
- Angle. Specify the angle to rotate the geometry around the vector.
- Display Axis. Click to view the rotation axis.

To export the transformation settings, select File > Export > Geometry Transformations. Choose which sections to export data from then specify a file. Click the export button to save the transformation data to a text file.

Merging Geometry Sections

Geometry sections can be merged at any time.

1. Select the section to merge into. The merged sections inherit this section name.
2. Click the Merge button in the Sections area of the Geometry tab.
3. Select the geometry sections to merge then click OK.

Copying Geometry Sections

Created and imported geometry sections can be copied.

1. Select the geometry section you want to copy.
2. Click the Copy  button.

A copied geometry appears in the same position as the original; it is not offset from the original's position. The name of the new section has a copy number at the end of the name of the original section.

Particle Factories

Creating Particle Factories

Particle factories are used to define where, when and how particles appear in a simulation. Any virtual surface or volume (physical or virtual) can be turned into a particle factory. Particles, based on the prototype particles previously defined, are then created on or in the specified section. A model can have any number of particle factories.

Click on the **+** button to create a new factory. Click the copy button to create a copy of the currently selected factory. Click the Import button to import your own custom particle factory (for details on custom particle factories, refer to the *EDEM Programming Guide*).

There are two types of particle factory: *static* and *dynamic*.

Static Factories

Static factories produce particles at a specified time. Simulation is paused during particle creation.

Parameter	Details
Number or mass of particles	<p>For a static factory, select either:</p> <ul style="list-style-type: none"> ▪ Fill Section. Creates as many particles as will fit on or in the chosen geometry section. ▪ Total Number. Produce a specified number of particles. ▪ Total Mass. Produce a specified amount of mass.
Start time	Set the time at which particle creation will take place.
Max attempts to place particle	<p>If you select random particle position, you must also define the maximum number of attempts to place a particle.</p> <p>During simulation, EDEM attempts to place particles in a random position in or on the specified section. If placing the particle in that position would cause an overlap with any other physical element in the model, the simulator will abandon the position and try to find another. It will attempt this for the specified number of times for each particle.</p>

Dynamic Factories

Dynamic factories produce particles over the duration of a simulation. Simulation continues as the particles are created.

Parameter	Details
Number or mass of particles	<p>For a dynamic factory, select either:</p> <ul style="list-style-type: none"> ▪ Unlimited Number. This creates as many particles as will fit on or in the chosen geometry section. If selected, the factory will attempt to create particles continuously from the specified start time until the end of the simulation. Particles will continue to be created even after a section surface or volume is full. This may cause the rest of the simulation to slow down. ▪ Total Number. Produce a specified number of particles.

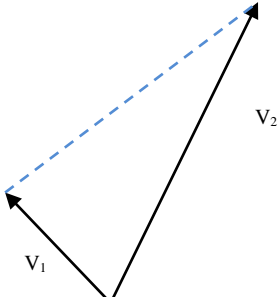
Generation rate	<ul style="list-style-type: none"> ▪ Total Mass. Produce a specified amount of mass. ▪ Target Number. Define the number of particles the simulator will attempt to place in or on the section every second. ▪ Target Mass. Define the mass generation rate the simulator should aim to create. When simulation starts, this rate is converted into mass per-timestep. If this is less than the smallest particle mass, the factory does not create a particle but adds the mass to the mass for the next timestep. Once there's enough mass, the factory creates a particle. If the mass per-timestep is greater than the smallest particle mass, the factory places particles until it reaches the mass per timestep. Be sure your factory is large enough to create the required mass rate: if not, a warning message may be displayed prompting you to update the rate.
Start time	Set the time particle creation starts.
Max attempts to place particle	During simulation, EDEM attempts to place particles in a random position in or on the specified section. If placing the particle in that position would cause an overlap with any other physical element in the model, the simulator will abandon the position and try to find another. It will attempt this for the specified number of times for each particle.

Initial Conditions

The initial conditions area is used to specify the section to use as a factory, and to define the starting conditions of the particles it creates.

Parameter	Options
Section	Select the section that will become a particle factory. Only suitable sections are listed. Any virtual surface or volume (physical or virtual) can be turned into a factory.
Type	<p>A factory can produce one or two different types of particle. Particle types must first be defined in the Particles pane. The options are:</p> <ul style="list-style-type: none"> ▪ Fixed. One fixed particle type. ▪ Binary. Two types of particle can be used. This option is useful when creating binary lattices.
Size	<p>The standard size of any particle type is defined in the Particles pane. Factories can also produce particles with a distribution of sizes based upon this standard size. All sizes are defined as factors of the originally specified size. The options are:</p> <ul style="list-style-type: none"> ▪ Fixed. All particles are of an equal size: For example, if the size ratio is set to two, all particles will be twice the standard radius. ▪ Random. Particle radii are randomly sized within a set size

Parameter	Options
	<p>range: For example, with a minimum size of 0.5 and a maximum of 1.5, particles no smaller than half and no bigger than one-and-a-half times the size of the standard size are produced.</p> <ul style="list-style-type: none"> ▪ Normal distribution. Particle radii have a normal distribution. The average (mean) particle radius or volume and the standard deviation must be defined. The mean and standard deviation should both be normalized i.e. the mean is the ratio of the mean to the input particle radius or volume and similarly for the standard deviation. Use the capping options to cap the maximum and minimum particle sizes. ▪ Log-normal distribution. Particle radii have a log-normal distribution. The average (mean) particle radius or volume and the standard deviation must be defined. The mean and standard deviation can be defined using a logarithmic or linear scale. With a logarithmic scale, the input values are exponential functions. For example, a mean value of 1 in the logarithmic scale denotes an actual scaling of 2.718 in the linear scale. Note that the mean and standard deviation should both be normalized i.e. the mean is the ratio of the mean to the input particle radius or volume and similarly for the standard deviation. <p>Note that changing the size of particles in this way does not affect Rayleigh timestep calculations. Setting a greatly reduced particle size may require a smaller timestep.</p>
Position	<p>When particles are created, they are placed on or in the specified section. Particles are never created on top of one another. When a factory tries to create a particle it will repeatedly attempt to find free space to place it.</p> <p>The position of the particles can be:</p> <ul style="list-style-type: none"> ▪ Random. Particles are positioned randomly or through a random walk. With a random walk, particles are placed one after another along a path in discrete steps of a defined length. This length is a factor of the maximum extent of the particle. The maximum angle that can be turned at each step is also defined. ▪ Cubic. Particles are created in a simple cubic lattice. Select the Display Lattice option in the Viewer controls pane to show the lattice in the Viewer. ▪ BCC lattice. Particles are created in a body-centered cubic lattice. It is defined in the same way as a cubic lattice. ▪ FCC lattice. Particles are created in a face-centered cubic lattice. It is defined in the same way as a cubic lattice. <p>Note only a random position is selectable when defining a total mass to create.</p>
Velocity	<p>Particles can be given a velocity at the time of creation. Options are:</p>

Parameter	Options
	<ul style="list-style-type: none"> ▪ Fixed. All particles have the same fixed velocity. The velocity must be defined as a vector. ▪ Random. Each particle is given a velocity at random within a defined range. ▪ Normal distribution. The particle velocities have a normal distribution. The particle direction, mean velocity and the standard deviation must be defined. ▪ Linear distribution. The particles are given velocities that fall on the line segment between the two velocity vectors provided by the user. <div style="text-align: center; margin: 20px 0;">  </div> <ul style="list-style-type: none"> ▪ Spray. The particles are formed in a conical spray. The spray direction is defined as a velocity vector. The angle is the angle of maximum deviation from the normal. The mean velocity and standard deviation must also be defined.
Orientation	<p>Particles can be given an orientation at the time of creation. This is useful for non-spherical particles. The options are as follows:</p> <ul style="list-style-type: none"> ▪ Fixed. All particles have the same fixed orientation. The orientation is defined in an <i>orientation matrix</i>. EDEM automatically adjusts the values you enter to maintain an orthogonal orientation matrix. ▪ Random. Each particle is given a random orientation.
Angular velocity	<p>Particles can be given an angular velocity at the time of creation i.e. they will be spinning when placed on or in the specified section. The options are as follows:</p> <ul style="list-style-type: none"> ▪ Fixed. All particles have the same fixed angular velocity. The velocity must be defined as a vector. ▪ Random. Each particle is given an angular velocity at random within a defined range. ▪ Normal distribution. The particle angular velocities have a normal distribution. The particle direction, average angular velocity and the standard deviation must be defined. ▪ Linear distribution. Particles are given angular velocities that

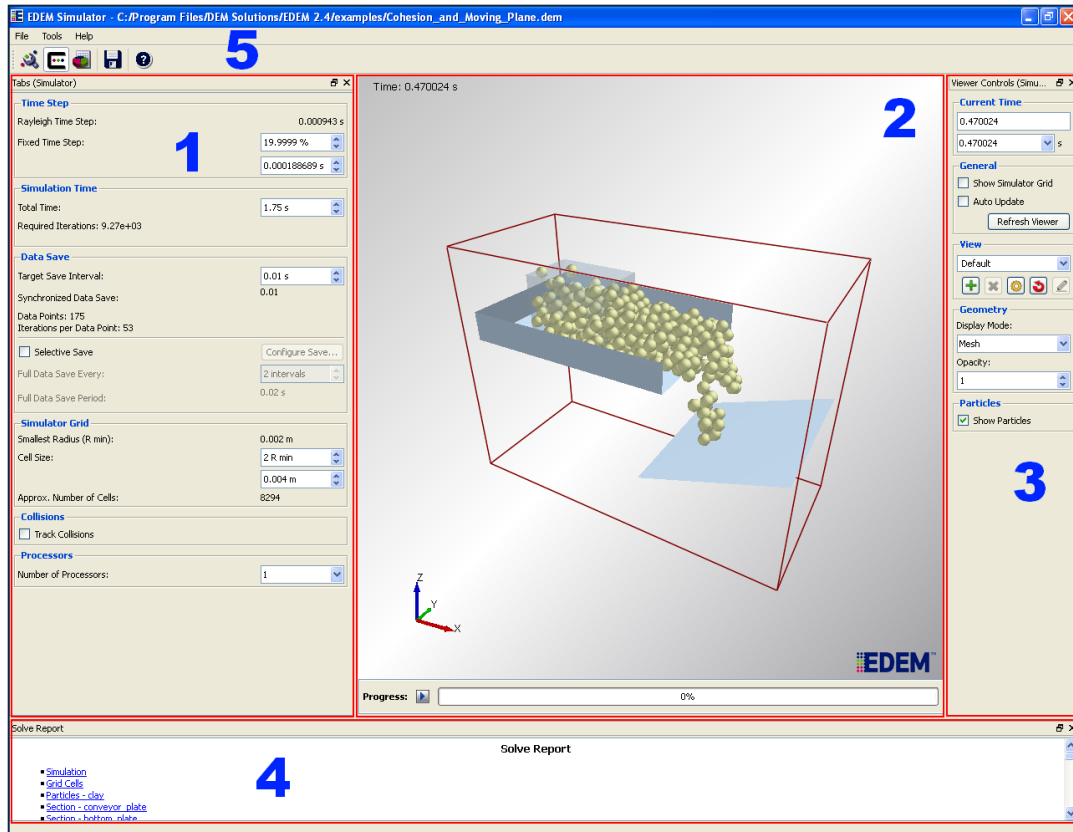
Parameter	Options
	fall on an arbitrary arc drawn between two velocity vectors.
Custom Property	<p>If you have defined any custom properties, particles can be given a custom property value at the time of creation. Options are:</p> <ul style="list-style-type: none"> ▪ Fixed. All particles have the same fixed value. ▪ Initial. Particles use the initial value of the property. ▪ Normal. Custom property values have a normal distribution. The mean and standard deviation must be defined. ▪ Random. Each particle is given a value at random within a defined range.

Table 2-1: Table of particle parameters

EDEM Simulator

About the EDEM Simulator

The EDEM Simulator is the discrete element solver used to carry out your simulation. Select the contact model and watch as your model is processed.



- 1 Simulator Pane
- 2 Viewer
- 3 Viewer Controls
- 4 Solve Report
- 5 Toolbar and Menu Bar

Simulator Pane

The Simulator Pane is where you set the time step, simulation time, and grid options.

Viewer

The Viewer displays 3D representations of your particles, geometry, and fields (if applicable). The rotation, position and zoom factor of the Viewer are controlled using the mouse. The simulation start/stop button and simulation progress bar are just below the viewer (see *Starting and Stopping a Simulation* on page 54 for more details).

Viewer Controls (Simulator)

The Viewer Controls are used to determine how items are displayed in the Viewer.

- **Current time.** The current simulation time is displayed. Simulation can be stopped and any previous full time step viewed. Simulation can be restarted from any previous full time step.
- **General**
 - **Show simulator grid.** Enable or disable the display of the simulator grid cells.
 - **Auto update.** If selected, the Viewer updates for every iteration. Simulation will be slower, however the slow down can be minimized by lowering the particle detail.
 - **Refresh Viewer.** When the auto update option is disabled, click to refresh the Viewer display.
- **View.** Controls to change, add, and configure camera views. See *Viewer Controls (Creator)* on page 16.
- **Geometry**
 - **Display mode.** Configure the display of all the geometry in the Viewer. Choose between filled, mesh or points style display.
 - **Opacity.** Set the opacity level of the geometry in the Viewer.
- **Show particles.** Display particles.
- **Fields**
 - **Display field.** Select a field from the pick-list then tick to enable or disable display of the field in the viewer. Vector fields are displayed as arrows; scalar fields are displayed as points.
 - **Color.** Select a color to display the field as. The default is Blue.
 - **Scale.** Set the length of the field's vector line.

Solve Report (Simulator)

The Solve Report is an .html page that displays detailed information about your model. Right click anywhere within the Solve Report and choose Save to save the information in an .html file. There are six sections in the Simulator's Solve Report:

Simulation

- **status.** Whether the simulation is processing or stopped.
- **current time.** Time into processing.
- **time step.** The time step used by the model as detailed in the Simulator.
- **estimated total computation time.** The estimated total time to complete the simulation.
- **elapsed computation time.** Time since the simulation started (excluding pauses).

Grid cells

- **status.** Either allocating grid cells or grid cells have been initialized.
- **total number.** The total number of grid cells in the model.
- **side length.** The dimensions of grid cells as specified in the Simulator.
- **created.** The number of grid cells created.

Particles

- **total number.** The total number of particles in the model.
- **total removed.** The total number of particles removed from the model: For example, particles that move beyond the domain boundaries.
- **minimum speed.** The speed of the slowest particle in the model.
- **maximum speed.** The speed of the fastest particle in the model.

Geometry Section

- **speed.** The speed of the geometry section.
- **number of particles in contact.** The total number of particles in contact with the geometry section at the current time step.








Energy

- **system energy.** The total energy in the model at the current time. It is the sum of the kinetic energy, potential energy and the energy in any contacts taking place. It should be equal to the total energy gained by creating particles plus the total energy gained by geometry movement minus the energy lost through contacts minus the energy lost by removing particles.
- **kinetic & potential energy.** The total kinetic and potential energy in the system.
- **total lost by removing particles.** Total energy lost by removing particles from the domain.
- **total lost through contacts.** Total energy lost by contacts between elements.
- **total gained by creating particles.** Total energy gained.
- **total gained by geometry movement.** The total energy gained through the movement of sections of geometry.

Factory

- **status.** The current factory status.
- **total particles created.** Number of particles accepted and currently being generated.
- **total particles regenerated.** Number of particle creation attempts.
- **total particles failed.** Number of particles that could not be placed once the *Max attempts to place particle* is reached.
- **total particles accepted.** Number of particles accepted in the grid.
- **creation rate/target creation rate.** The particle or mass creation rate as specified in the Factories pane.
- **mass created since rate change.** Amount of mass created by the factory (when creation rate is set to target mass rate of particles)
- **actual creation rate since rate change.** The actual creation rate (as opposed to the target creation rate) when creation rate is set to target mass rate of particles.

Simulator Toolbar

Icon	Name	Description
	Creator	Switch to the Creator
	Simulator	Switch to the Simulator
	Analyst	Switch to the Analyst
	New	Start a new model
	Open	Open an existing model
	Save	Save any changes made to Simulator settings
	Help	View the online help

Simulator Menu Bar

File Menu

- **Save.** Save any changes made to Simulator settings.
- **Creator.** Select to switch to the Creator.
- **Simulator.** Select to switch to the Simulator.
- **Analyst.** Select to switch to the Analyst.
- **Quit.** Quit EDEM.

Tools Menu

The Tools Menu in the Simulator is the same as for the Creator. See the Creator Tools menu on page 19 for details.

Simulator Settings

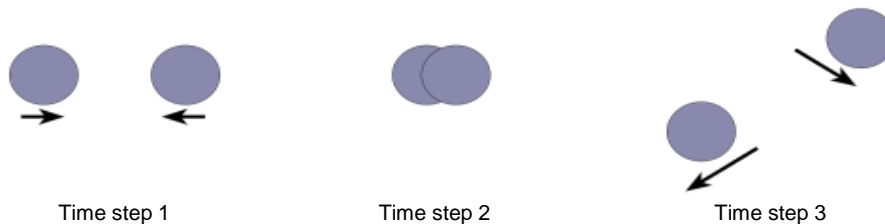
Time Steps

The time step is the amount of time between iterations (calculations) in the Simulator. The time step is fixed and remains constant throughout the simulation. The value is displayed as both the actual time step (in seconds) and as a percentage of the *Rayleigh* time step. The Rayleigh time step is the time taken for a shear wave to propagate through a solid particle. When using a simulation with a range of particle sizes, the Rayleigh time step is calculated based on the smallest particle size. Refer to *Appendix A: Estimating Simulation Time* for more information.

The smaller the time step, the more data points are produced. A large number of data points produce results with a very fine level of detail, however the simulation time will be longer due to the increased number of calculations required.

Choosing a Time Step

If the time step is too small, the simulation will take a long time to run. If the time step is too large, particles can behave erratically. For example, the figure below shows two particles moving towards each other. At time step 1 they are some distance apart. The particles are moving towards each other at some speed, and when their positions are recalculated at time step 2 they are apparently over-lapping. The particle forces and energies are recalculated at this point, however these values are very large due to the apparent overlap; this supplies each particle with a very large and incorrect velocity, and they move off erratically at time step 3. Subsequently, these particles can come into contact with other elements in the system, often causing an 'explosion' of incorrectly moving elements. To recover from such an event, set particle limits using the Creator's Particles tab.



Simulation Time

The simulation time is the amount of real time your simulation represents.

- **Total Time:** The total actual length of the simulation in seconds or minutes.
- **Required Iterations:** Indicates the number of iterations required to complete a simulation of a defined time using the defined time step. For example, $5e^{+05}$ iterations are required to produce one second of simulation with a time step of $2e^{-06}$.

Data Save

- **Target Save Interval:** How often data should be written out to a file. This determines the number of data points (the points at which data is collected from the simulation for analysis).
- *Synchronized Data Save:* The actual time that data will be saved. This is usually the same as the target save interval, but may differ depending on the timestep time.
- *Data Points:* The number of data points as determined by the target save interval, it is calculated as the total run time divided by the target save interval.
- *Iterations per Data Point:* The number of simulator iterations for each data point.

Selective Saves

When simulating, EDEM saves particle, geometry, and interaction data for post-processing and analysis. Often you may only be interested in a subset of this data: EDEM's *selective save* feature lets you select which attributes are saved and how often by defining *partial saves*. Using selective save can greatly reduce simulation data file sizes as well as enabling you to run even larger simulations.

Selective save works by combining full saves (where all data is written out) and partial saves (where only selected data is written out) to create smaller data sets. For example, assume you have a simulation with a total time of 10 seconds that writes-out every 1 second. By default, this creates a full save of all simulation data every second:

0 s	1 s	2 s	3 s	4 s	5 s	6 s	7 s	8 s	9 s
Full	Full	Full	Full	Full	Full	Full	Full	Full	Full

Setting the *Full Data Save Every* option to e.g. 3 intervals results in a full save only every 3 seconds (the first write-out is always a full save):

0 s	1 s	2 s	3 s	4 s	5 s	6 s	7 s	8 s	9 s
Full	Partial	Partial	Full	Partial	Partial	Full	Partial	Partial	Full

To enable selective save:

1. Tick the Selective Save checkbox.
2. Enter a value into Full Data Save Every to set the frequency of full data saves. For example, set this to create a full save every 5 intervals.
3. Click the Configure Save button. The Data Save Configuration window is displayed.
4. Select which particle, geometry, and interaction attribute data to save in a partial save. Double-click an attribute to include it in the save. Any other required attributes will also be enabled. Untick a data group to exclude related attributes.

The Data Save Configuration window also displays an estimate of the per-particle or per-triangle reduction in data size for each timestep.

Note that if you stop the simulation early then switch to the analyst, the current timestep will be a full save irrespective of the full save interval.

Particle Data

Data Group	Attribute Data
Base Particle Data	ID
	Position
	Orientation
	Number of Particles
	Volume
	Voidage
	Mass
	Moment of Inertia
	Potential Energy
Velocity	Velocity
	Kinetic Energy
Angular Velocity	Angular Velocity
	Rotational Kinetic Energy
	Total Energy
Force and Torque	Compressive Force
	Total Force
	Torque
Custom Properties	Custom Property

Geometry Data

Data Group	Attribute Data
Base Geometry Data	ID
	Position
	Surface Normal
	Node
	Number of Geometry Elements
	Velocity
	Angular Velocity
Force and Torque	Compressive Force
	Total Force
	Torque
Custom Properties	Custom Property

Interaction Data

Data Group	Attribute Data
Contacts (Particle-Particle)	All particle-particle contact attribute data
Contacts (Particle-Geometry)	All particle-geometry contact attribute data
Collisions (Particle-Particle)	All particle-particle collision attribute data
Collisions (Particle- Geometry)	All particle-geometry collision attribute data
Bonds	All bond element attribute data
Custom Properties	All interaction-based custom property data

Simulator Grid

Select the Show Simulator Grid option to display the grid in the Viewer.

The main computational challenge in DEM simulation is the detection of contacts. By dividing the domain into grid cells, the simulator can check each cell and analyze only those that contain two or more elements (and therefore a possible contact), thus reducing processing time. The results achieved by a simulation are not affected by the number of grid cells, only the time taken to reach them. The simulation process is:

1. Draw the grid, dividing the domain into cells of a specified size.
2. Identify active cells (those containing two or more elements) and check for contacts.
3. Calculate forces on elements.
4. Reposition elements as a result of any force acting upon them.
5. Active cells are again identified and the process repeats.

As the grid length decreases, fewer elements are assigned to each grid cell and contacts become easier to resolve. The fewer particles per grid cell, the more efficient the simulator. If there is no more than one particle in each grid cell then no contact detection needs to take place so the simulation will progress faster. The idealized length of a grid cell is $2R_{\min}$ where R_{\min} is the minimum particle radius in the simulation. If this results in more memory allocation than available, reduce the number of grid cells to avoid the time-consuming swapping of memory to the hard disk.

Collision Detection

EDEM uses the grid to detect collisions. The domain is split into grid cells whose dimensions are approximately that of the particle diameter. Each particle and geometry element is then assigned to one or more grid cell based on its position. Once this is complete for all particles and elements, each grid cell is investigated to see whether it contains more than one object. Any grid cell containing more than one object is analyzed further to check for objects in contact with each other. To do this, EDEM compares the distance between the center of the spheres and the known radii of the spheres.

Storing Collision Data

During a simulation data can be recorded about both contacts and collisions.

- **Contacts** are the impacts occurring between elements at data write-out points. In other words, the contact is *in progress* when the write-out takes place. The contact

has an associated force, position and so on - these are discrete values. If two elements stay in contact with each other for some time e.g. over 4 write-out points, four contacts will be stored and each of these may have a different force, position and so on.

- **Collisions** are complete impacts. When two elements collide it will register as one collision, regardless of how long the elements stay in contact for. Data is collected for the duration of the collision e.g. total energy loss, min/max/average normal force data and so on. Collisions may occur in-between write-outs and never register as contacts.

EDEM records all contact data as standard. To record collision data, enable Track Collisions. Storing collision data will *significantly increase* memory usage. Collision data is stored in memory until it is written out at the rate specified in the Simulator. If a large number of collisions are occurring and the write out frequency is too low, the amount of data being stored in memory will become very large and use up all memory resources. To minimize this, be sure data is written out frequently.

Processors

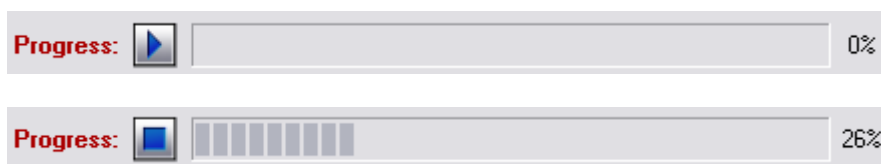
The number of processors available to the Simulator depends on:

- The number of physical processor cores in your system
- The number of processor keys you have purchased

Select the required number of processors from the Number of Processors pulldown. When you start a simulation, EDEM reserves the requested number of keys from the license pool (if available). If your BIOS has hyper-threading enabled, EDEM may report the number of processors incorrectly. Be sure to consider the number of physical cores in your system.

Starting and Stopping a Simulation

The start/stop button is used to start and stop the simulation:



The current simulation time is shown in the top left corner of the Viewer Controls pane. All previous time steps are also listed. You can stop and start the simulation from any time step (current or previous) using the start/stop button. Simulation can also be stopped using the keyboard shortcut Ctrl + Shift + K.

Starting a simulation reserves processor keys from the license pool. If there are not enough keys currently available, EDEM prompts you to decide whether to stop or continue with fewer processors (depending on how many processor keys are available).

Stopping a simulation saves simulation data (this may take a while, depending on the size of the simulation) and returns reserved processor keys to the pool of total keys available to all users.

Reducing Simulation Time

The following can help to reduce processing time.

- Close the Data Browser.
- Turn off the auto-update option. If this option is selected, processing time can still be reduced by lowering the particle detail.
- Reduce the simulator grid size.
- Use partial saves to write-out less data to the simulation file(s).

Simulation Parameters

The time taken by the software to perform a single iteration, $t_{iteration}$, is affected quite significantly by the simulation parameters. An obvious parameter is the number of elements (particle surfaces and geometry elements) to be simulated. EDEM has a near-linear dependence on the number of elements up to 20,000 (at which point memory considerations can affect the simulation speed).

Contact detection is the most significant overhead to an EDEM simulation and extra parameters such as geometry dynamics and contact models typically have a secondary effect. However, it is worth noting that Particle Factories use contact detection to place their particles so a high number of factories, or factories within regions of space with medium to high particle volume fractions will significantly affect the simulation speed.

The Data Browser

The data browser (solve report) at the bottom of the EDEM window is constantly updated with particle details. To speed up the simulation, close the data browser or turn off the display of factory information.

You can re-open the browser by right-clicking anywhere on the EDEM toolbar and selecting Data Browser from the pop-up menu. The display of factory information in the Data browser can be turned on and off using the controls in the Simulator's Tools > Solve Report menu.

The Auto-update option

When the auto-update option is selected (from the Viewer controls pane in the Simulator), the Viewer is updated for every iteration (time step). Potential particle positions are also drawn. Consequently simulation will be slower, however the slow down can be minimized by lowering the particle detail.

The particle detail controls are used to determine the number of polygons used to create each particle. The greater the number of polygons, the better the appearance of the particle. However, if the polygon number is high it will slow down the following:

- Simulation (if the auto-update option is selected)
- Animation playback in the Analyst
- Altering the view in the Viewer (position, rotation or zoom factor).

Grid size

The fewer particles per grid cell, the more efficient the simulator. If there is no more than one particle in each grid cell then no contact detection needs to take place so the simulation will progress faster. The idealized length of a grid cell is $2R_{\min}$ where R_{\min} is the minimum particle radius in the simulation. If this results in more memory allocation than available, reduce the number of grid cells to avoid the hugely time-consuming swapping of memory.

The grid options are in the Simulator.

Domain size

The domain is the area in which simulation takes place and is illustrated by the red box in the Viewer. The size of the domain size has an effect on simulation time - the larger the domain, the longer the simulation will take to run. Use the auto-update option to automatically fit the domain around any sections of geometry defined, thus creating the smallest domain possible. Similarly if you are only interested in the activity in a particular part of your geometry, reduce the domain to fit around that area.

Please note that sections of geometry should not be placed totally coincident with the edge of the domain. Particles are removed as soon as they reach the domain edge, so cannot interact with any geometry placed there.

The domain options are in the Geometry pane in the Creator.

Hardware and Drivers

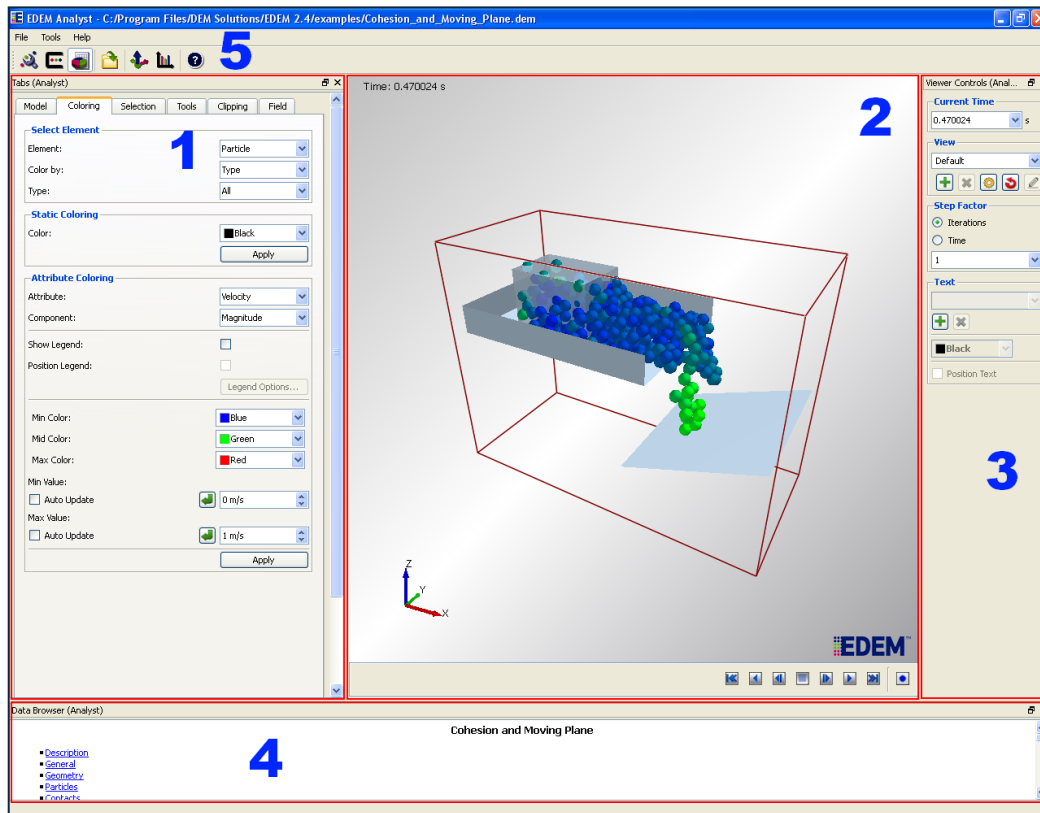
Earlier it is shown how $t_{\text{iteration}}$ is affected by the amount of memory available to the software. Of equal importance is the type of memory, the processor(s) and the communication between the two. A faster processor is capable of carrying out more calculations per second and adding extra processors allows for the total computational load to be divided. DEM Solutions has good links with hardware vendors and tests are continuously under way on possible speed improvements through hardware.

In addition to the hardware, maintaining up-to-date drivers can also help.

EDEM Analyst

About the EDEM Analyst

The EDEM Analyst is the post-processor used to examine the results of your simulation. Play back your simulation as an animation, graph your results, create a video or export data for examination elsewhere.



- 1 Tabs Pane
- 2 Viewer
- 3 Viewer Controls
- 4 Data Browser
- 5 Toolbar and Menu Bar

Tabs Pane (Analyst)

The Tabs Pane is displayed on the left side of the EDEM window. It has five tabs: Model, Coloring, Selection, Tools, Clipping, and Fields.

Viewer

The Viewer displays 3D representations of your particles, geometry, and fields (if applicable). The rotation, position and zoom factor of the Viewer are controlled using the mouse.

A number of animation controls appear just below the Viewer. See *Reviewing Your Simulation* on page 61 for details.

Viewer Controls (Analyst)

The Viewer Controls are used to determine how items are displayed in the Viewer.

- **Current Time.** Choose the time step displayed in the Viewer. Partial-save time steps are shown in a lighter color.
- **View.** Controls to change, add, and configure camera views. See *Viewer Controls (Creator)* on page 16.
- **Step Factor.** The step factor determines the number of iterations or the amount of time the animation controls step through. For example, setting the factor to 5 iterations will increase the speed of the animation playback as only every fifth iteration is displayed. Setting the time factor to every iteration will result in a smoother but slower playback. When your simulation contains partial saves (see *Selective Saves* on page 51), setting the step factor to match the partial save interval results in a constant display of full-save timesteps.
- **Text.** Add descriptive text to the Viewer. Click the **+** button and enter the text in the text field. Choose the text color using the color drop-down list. To position the text select the Position Text option and click a location anywhere within the Viewer window. To delete text, use the **X** button.

Data Browser (Analyst)

The Data Browser is an .html page that displays detailed information about your model. The information displayed differs between the Creator, Simulator and Analyst. Right click anywhere within the Data Browser and choose Save to save the information in an .html file.

The Data Browser within the Analyst has the following sections: Description, General, Geometry, Particles, Contacts, Bonds, and Selections.

Description

The description as defined in the Globals pane.

General

- **Dimensions.** The number of dimensions in your domain.
- **Gravity.** Details of the gravity acting in your model as specified in the Globals pane.
- **Materials.** A list of all materials used in the model and their properties, as specified in the Materials Editor.
- **Energy.** The total energy in the model at the current time. It is the sum of the kinetic energy, potential energy and the energy in any contacts taking place.

Geometry

- **Domain.** The dimensions of the domain as specified in the Geometry pane.
- **Geometry Totals.** Details of the number of elements that make up each section of geometry.
- **Sections.** Each section in the model and its properties are listed. Properties are as specified in the Geometry pane.

Particles

- **Particles.** A list of each particle type used in the model and their properties. Details of each surface within the particle are also listed. Properties are as specified in the Particles pane.
- **Factories.** Details of each factory used in the model and their properties. Properties are as specified in the Factory pane. The total number of particles created by each factory is also listed.
- **Particle totals.** The total number of particles in the model at the current time. Totals for each particle type are also listed.

Contacts

- **Interactions.** A list of the interaction types taking place in the model as specified in the Materials Editor.
- **Contacts.** Lists the total number of contacts *in progress* at the current time. The contacts are broken down into contact types: For example, the number of particle A - particle B contacts or particle B - surface A contacts. The total number of collisions that *have taken place* during the time step are also listed.








Bonds

- **Bond totals.** A list of the bond totals, indicating total number of bonds, total number of intact bonds, total number of broken bonds, and total intact and broken bonds for each particle type.

Selections

- **Selections.** All selections are listed. This section has links to pages with information on the selection and its associated queries, list of particles, and geometry elements (if applicable).

Analyst Toolbar

Icon	Name	Description
	Creator	Click to switch to the Creator.
	Simulator	Click to switch to the Simulator.
	Analyst	Click to switch to the Analyst.
	Open	Click to open an existing model.
	3D Viewer	Click to switch to the 3D Viewer.
	Create Graph	Click to switch to the Graph Creator.
	Help	Click to view the online help.

Analyst Menu Bar

File Menu









- **Open.** Open an existing model.
- **Truncate File.** Remove sections of data from your model. See *Truncating Data* on page 93 for details.
- **Export.** Opens a sub-menu with the following options:
 - **Simulation Deck.** Export a single full time step from your simulation to create a copy as a new EDEM simulation deck (you cannot export from partial save time steps). Exporting a deck allows it to be transferred to another location and run again using identical simulation parameters. To export a simulation deck:
 1. Enter a name for the deck then click Save.
 2. From the Export Options dialog, select the content options:
 - Un-tick the Export Custom Properties option to exclude any custom property data from the deck.
 - To export a time step other than the current time step, tick the Set Start Time checkbox then enter a starting time in seconds.
 3. Click OK to save the simulation deck.
 - **Results Data.** Export specified data for analysis. See *Exporting Data* on page 87 for details.
 - **Image.** Save a copy of the image currently displayed in the Viewer. See *Working with Images* on page 83 for details.
- **Print Image.** Print a copy of the image currently displayed in the Viewer.
- **Creator.** Select to switch to the Creator.
- **Simulator.** Select to switch to the Simulator.
- **Analyst.** Select to switch to the Analyst.
- **Recent Files.** A list of recently opened files. Select a file to open it.
- **Quit.** Quit EDEM.

Tools Menu

The Analyst's Tools Menu is the same as for the Creator. See the Creator Tools menu on page 19 for details.

Reviewing Your Simulation

The Analyst is used to review and examine your simulation. A number of animation controls appear just below the Viewer. These are used to examine every iteration of your simulation. The buttons are:

	Jump to start		Stop		Jump to end
	Play backwards		Step forward		Record animation (video)
	Step backward		Play forwards		

Note: Close the Data Browser to speed up playback.

Model Tab

Options in the Model tab are used to configure the display of your model in the Viewer. You can enable or disable the display of geometries, particles, contacts, bonds, or selection groups (including bins and bin groups).

Note: If there are any display conflicts, the relevant control is highlighted.

Geometry

Display options can be applied to all the geometry in your model, to individual sections, or to selection groups. The options apply to the geometry or selection group selected in the drop-down lists. Once you have set the geometry options, click the Apply button.

Display by

Select Type for geometry sections or Selection for geometry in a selection.

Type/Selection

Depending on the Display-by drop-down list, select either a geometry section, all geometry sections, or a selection.

Properties by selection

For selections, enable or disable geometry type display options for the selected selection.

Display checkbox

Enable or disable the display of the selected section or selection.

Display Mode

Choose between filled, mesh or point style display. The mesh option is used to display the triangular elements the section is made up of. The points option shows the vertex of each element.



Figure 4-5: Filled Geometry

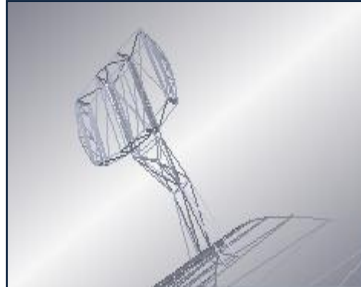


Figure 4-6: Mesh Geometry



Figure 4-7: Points Geometry

Opacity

Set the opacity level from 0 (fully transparent) to 1 (opaque).

Particles

Display options can be applied to all particles, particles of a particular type, or particles in a selection group. The options apply to the particle type or selection group selected in the drop-down lists. Once you have set the particle options, click the Apply button.

Display by

Select Type for types of particles or Selection Group for particles in a selection group.

Type/Selection Group

Depending on the Display-by drop-down list, select either a type of particle, all particles, or a selection group containing particles.

Properties by selection group

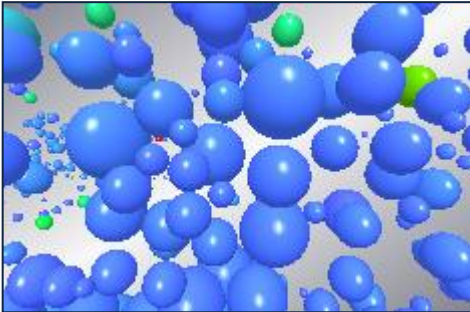
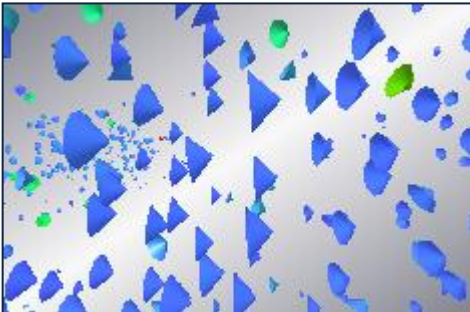
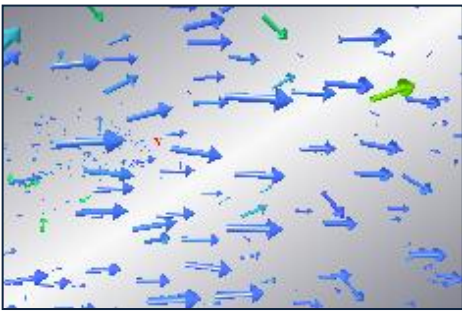
For display by selection groups, enable or disable particle type display options for the selected selection group.

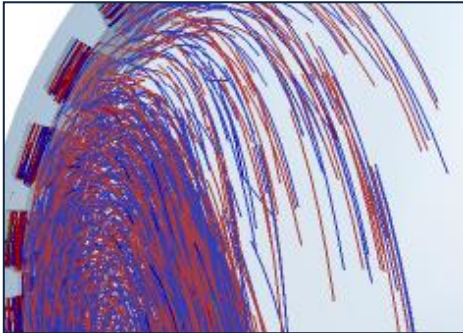

Display

Enable or disable the display of the selected particles or selection group.

Representation

Choose how to represent particles: either default, cone, vector, stream, or template. Click the Options button to set further display attributes.

Option	Description
Default	<p>By default, particles are displayed as they were defined in the Creator.</p> 
Cone	<p>Particles are displayed as cones. The direction, length and base width of the cone can be set to represent a variety of particle attributes.</p> <ul style="list-style-type: none"> ▪ <i>Direction.</i> Represents velocity, torque, total force, angular velocity, or custom property (consisting of exactly three values) ▪ <i>Length.</i> Represents: velocity, total force, moment of inertia, kinetic energy, mass, rotational kinetic energy, compressive force, torque, volume, angular velocity, or a custom particle property. ▪ <i>Base width.</i> Can represent: velocity, total force, compressive force, torque, mass, kinetic energy, rotational kinetic energy, volume, moment of inertia, angular velocity, charge, electrostatic force, or a custom particle property. <p>Particle display reverts to standard spheres when EDEM cannot show particles as cones (for example, if the selected attribute has no values).</p> 
Vector	<p>Particles are displayed as vectors. The direction and length of the vector can be set to represent a variety of particle attributes.</p> <ul style="list-style-type: none"> ▪ <i>Direction.</i> Can represent: velocity, torque, total force or angular velocity, or custom property (consisting of exactly three values) ▪ <i>Length.</i> Can represent: velocity, total force, moment of inertia, kinetic energy, mass, rotational kinetic energy, compressive force, torque, volume, angular velocity, or a custom particle property. 

Option	Description
Stream	<p>The path of the particle is displayed as a stream with variable coloring.</p> <ul style="list-style-type: none"> ▪ <i>Number of Steps.</i> Define the number of time steps the particles should be streamed for. The number indicates that the streams will be drawn for the previous n time steps up to and including the current time step. ▪ <i>Stream All Steps.</i> Streams are drawn across all time steps. ▪ <i>Stream Fade.</i> Fades the color intensity of the streams. 
Template	<p>Particles are displayed using one of the currently imported templates.</p> 

Contacts and Bonds

Contacts

Contacts between different types of particle and between particles and individual sections of geometry can all be examined independently.

Select Contact from the Element pulldown then use the Type 1 and Type 2 controls to choose a pair of elements. Enable the Display option to show the contacts in the Viewer.

The display can represent the following attributes:

- Contact Vector
- Normal Force
- Normal Overlap
- Tangential Force

Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Bonds

Bonding between different types of particle can be examined independently.

Select Bond from the Element pulldown then select the Bond State (either Intact Bond or Broken Bond). Use the Type 1 and Type 2 controls to choose a pair of elements. Enable the Display option to show the bonds in the Viewer.

For broken bonds, set the distance threshold to define the distance between the center of particles before broken bonds are no longer displayed.

The display can represent the following attributes:

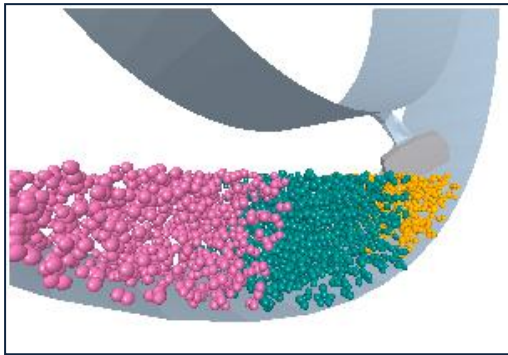
- Normal Force
- Normal Moment
- Tangential Force
- Tangential Moment

Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Coloring

Use the Coloring tab to color different elements of your model in a variety of ways. You can independently color any section of geometry, particle, contact type, bond, or selection group (including bins and bin groups).

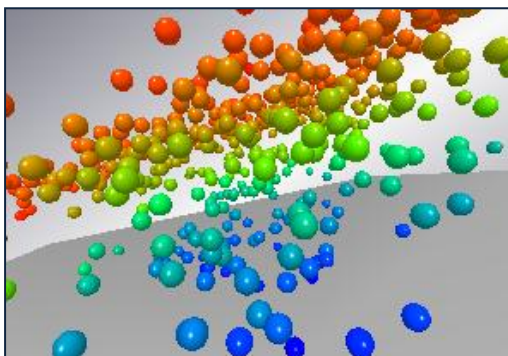
Static Coloring



Static coloring is used to set the standard color for any element: For example, a particular particle type or geometry in a selection group.

The color applies to whatever element and type or group is selected in the Select Element section. The figure on the left shows particles in three separate selection groups each colored differently.

Attribute Coloring



Attribute coloring is used to color any element of the model by a particular attribute: For example, the figure on the left shows all particles colored by velocity. The particles with the greatest velocity are colored red; those with the lowest colored blue.

Choose the element to color from the Select Element section, then select the attribute to color by.

The attributes and their components available for each element are listed in the table below. Refer to *Appendix B: Attribute Definitions* for more information.

Element	Attributes	Components
Particle	Angular Velocity	Magnitude, X, Y, Z
	Compressive force	n/a
	Moment of Inertia	Magnitude, X, Y, Z
	Kinetic energy	n/a
	Mass	n/a
	Potential energy	n/a
	Rotational kinetic energy	n/a
	Torque	Magnitude, X, Y, Z
	Total energy	n/a
	Total force	Magnitude, X, Y, Z
Particle	Velocity	Magnitude, X, Y, Z
	Volume	n/a
	Custom property	Depends on number of elements

Element	Attributes	Components
Geometry	Compressive force	n/a
	Total force	Magnitude, X, Y, Z
	Torque	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Custom property	Depends on number of elements
Contact	Contact vector 1	Magnitude, X, Y, Z
	Contact vector 2	Magnitude, X, Y, Z
	Normal force	Magnitude, X, Y, Z
	Normal overlap	n/a
	Tangential force	Magnitude, X, Y, Z
	Tangential overlap	n/a
	Custom property	Depends on number of elements
Bond	Normal force	Magnitude, X, Y, Z
	Normal moment	Magnitude, X, Y, Z
	Tangential force	Magnitude, X, Y, Z
	Tangential moment	Magnitude, X, Y, Z

Table 4-1: Attributes for Coloring Elements

A color should also be assigned for the minimum, median and maximum values. Elements will be colored depending on whereabouts in the range they fall.

The value range is determined by setting the minimum and maximum attribute values. The values can be entered directly or can be read from the model by clicking the update buttons. This takes the current maximum and minimum values of the selected attribute from the model. If the auto-update options are selected these values will update according to which time step is being viewed.

Attribute coloring can only be viewed when the current time step is either a full save or a partial save that included the required attribute.

Smooth Colors

Coloring of geometry attributes can be smoothed to better visualize the color gradation. With Geometry selected from the Element pulldown, select an attribute to color by then tick Smooth Colors. Without color smoothing enabled, each geometry triangle is displayed with one color only. With smoothing enabled, the color across each triangle is interpolated based on the value of the attribute at the nodes.

Coloring Legend

Enable the Show Legend option to display a color legend in the Viewer. Select the Position Legend option then click anywhere in the Viewer to position it. Click the Legend Options button to set the legend's width, height, number of intervals, alignment and title.

Selections and Binning

Selections allow data to be extracted from a particular area or element in the domain. Using selections, you can monitor any element passing through a particular area (known as a “bin”) or track particular elements wherever they move within the domain. You can then display, color, graph, or export data based on these selections and sub-selections.

There are five types of selections:

- Manual Selection
- Grid Bin Group
- Geometry Bin
- Imported Geometry Bin Group
- Mass Flow Analysis

Manual Selections

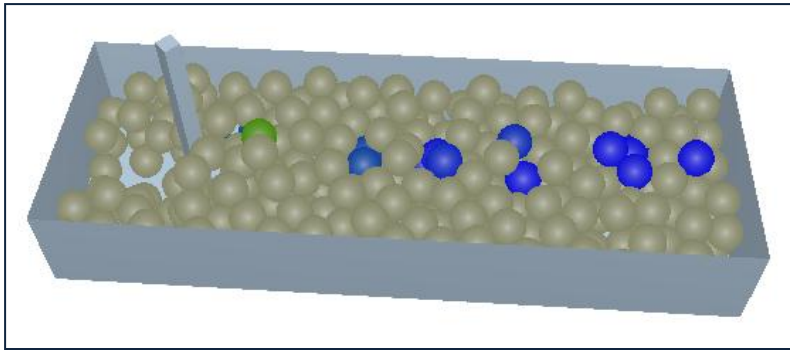
A manual selection lets you select and track individual particles, contacts, geometries, or bonds. You can then display, color, graph or export data based on these selections. You can also define and add queries, as well as display detail in the data browser.

1. From the Selection tab, click the **+** button then choose Manual Selection. Name the new selection.
2. Click the Enable Manual Selection checkbox. The cursor changes to a crosshair.
3. Move the pointer into the Viewer.
4. Single-click your selection, or press and hold the mouse button then drag the pointer to draw a selection box. All particle and geometry elements captured by the box that match the current filter will be part of the selection.

If an element is not displayed in the Viewer (after being turned off using the controls in the Models tab) it will not be captured by the selection box. All selected elements are highlighted in orange by default. Additional selection boxes can be drawn to add further elements to the selection.

All captured elements are listed in the Manual Selection area. Click the **x** button to remove them completely from the selection.

Select the Display in Data Browser checkbox to add information on elements in the selection to the Selection section within the Data browser.



This example shows a selection group where several particles have been individually selected. Coloring has been applied to show Velocity.

Grid Bin Groups

A Grid Bin Group splits the model domain into a lattice. Each grid is known as a bin group and each cell is known as a bin. Anything in or moving through each bin can be monitored.

To create a Grid Bin Group, from the Selection tab click the **+** button then choose Grid Bin Group. Define the selection then click the Apply button to apply settings and display the group in the Viewer.

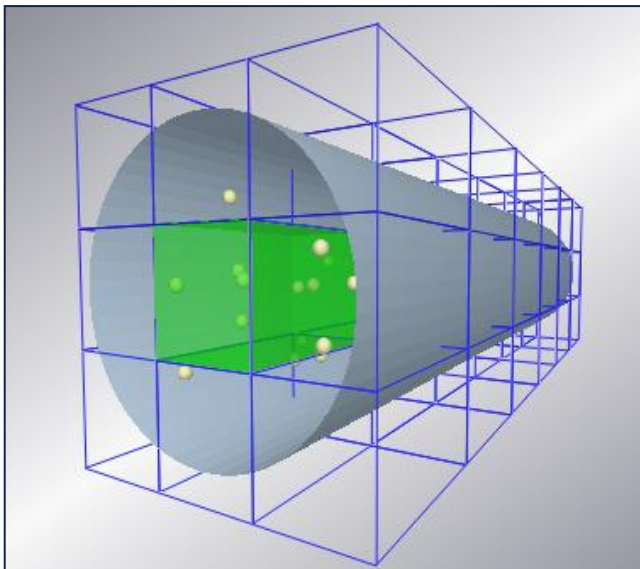


Figure 4-10: A Grid Bin Group with a Highlighted Bin at the Center

Grid Selection

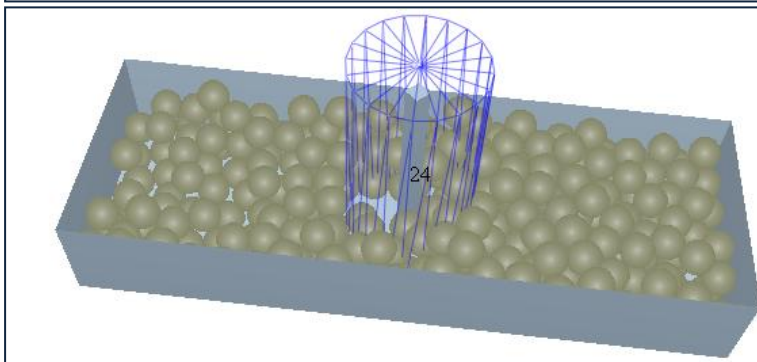
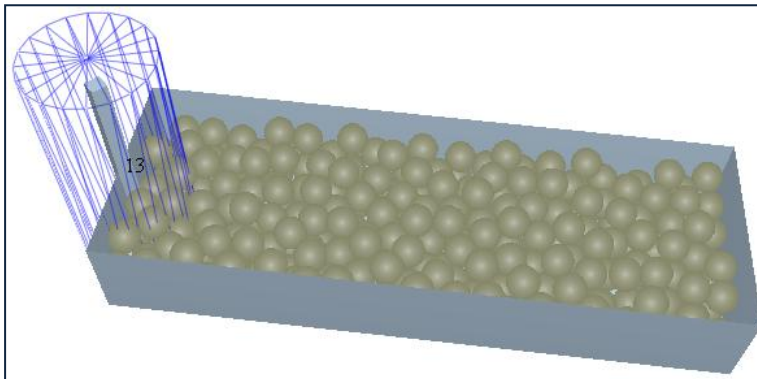
The Grid Selection section is used to define the size and number of cells in the bin group. By default the bin group covers the entire domain, however it can be limited to a specific area. The number of bins in the group is determined by the number of bins along each axis. Empty bins (those containing no elements) can also be removed. If the bin becomes non-empty at a different time step it is automatically added back into the group.

Geometry Bins

A geometry bin is a bin element based on a Box or Cylinder geometry type. You can link its position to any static or dynamic geometry section. The bin is displayed as a mesh of blue lines.

To define a geometry bin:

1. From the Analyst, click the Selection tab.
2. In the Selection Groups area, click + then choose Geometry Bin. Name the new bin.
3. Select the geometry type: either Box or Cylinder.
4. Set the bin's properties to define its size, shape, and initial position. For cylinders, set the number of sides to determine how many polygons are used to render the bin. Fewer polygons are faster to process, but at the expense of accuracy.
5. Set the filter options to determine what to include in the bin. By default all particles are included.
6. Select the display mode and on-screen query options. For example, set the display mode so that the bin is displayed only when it contains particles.
7. Select whether to link the bin's position to a geometry element defined in the Creator. This links to the center (or zero-point) of the geometry. If the geometry has dynamics, the bin will move along with it.



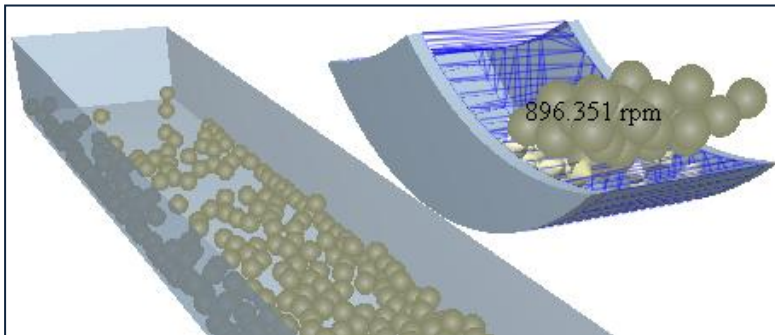
This example shows a geometry bin based on a cylinder. The bin indicates the number of particles and moves with the geometry it's linked to.

Imported Geometry Bin Groups

With the imported geometry bin group you can define a bin group based on an imported mesh file or CAD template created in another package. You can then link the bin group's position to any defined geometry.

To define an imported geometry bin group:

1. From the Analyst, click the Selection tab.
2. In the Selection Groups area, click **+** then choose Imported Geometry Bin Group.
3. In the CAD Template Selection section, click the Import button then navigate to a CAD template file. When prompted, set the units then tick the Merge Sections option to set the template as one bin, or leave it un-ticked to create one bin for each geometry element. The imported file must contain only solid geometry elements: you cannot import surfaces (non-closed elements such as a plane).
4. Set the filter options to determine what to include in the bin. By default all particles are included.
5. Select the display mode and on-screen query options. For example, set the display mode so that the bin is displayed only when it contains particles.
6. Select whether to link the bin's position to a geometry element defined in the Creator. If the geometry is dynamic, the bin will move along with it.



This example shows an imported geometry bin linked to a moving bucket geometry. A query has been added to display the average particle angular velocity.

Mass Flow Analysis

Mass flow rate is the rate at which particles pass through a bin group. It is measured in mass over time, for example, tons per hour or kilograms per second. Mass flow analysis enables the calculation of the rate of mass flow in a certain volume within the simulation. The user creates the volume (a cylinder) and places it within the model domain.

The calculation for mass flow rate for a single timestep is

Where:

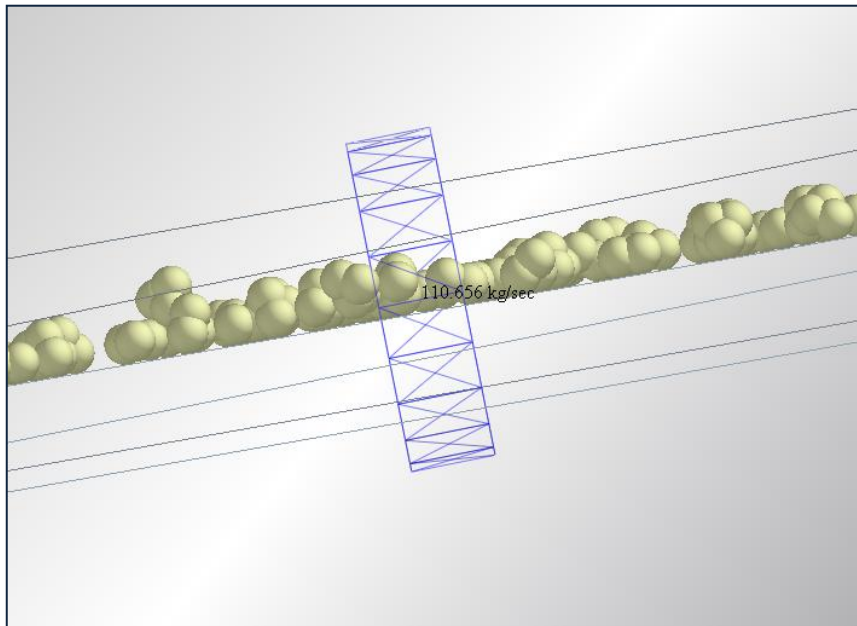
- \dot{m} = Magnitude of the mass flow rate.

- = The mass of particle i the selection bin.
- = The velocity of particle i in the selection bin as a vector.
- = The length of the cylinder as a unit vector.

For each particle its mass is multiplied by the dot product of its velocity and the unit vector of the cylinder from start to end. Each of these per particle values are then summed and divided by the length of the bin. The length of the bin is the distance from the start point to the end point of the cylinder.

To define the mass flow:

1. From the Analyst, click the Selection tab.
2. In the Selection Groups area, click + then choose Mass Flow Sensor. Name the new mass flow sensor.
3. Set the mass flow sensor's properties to define its size, shape, and initial position. For cylinders, set the number of sides to determine how many polygons are used to approximate the circumference of the cylinder. Fewer polygons are faster to process, but at the expense of accuracy.
4. Set the filter options to determine what to include in the calculation of the mass in the bin. By default all particles are included.
5. Select the display mode and on-screen query options. For example, set the display mode so that the bin is displayed only when it contains particles.



Selection Queries

A query is used to define a single element attribute, for example, particle velocity, total force on a geometry section or number of collisions. The query information can then be displayed in the Viewer and in the Groups section in the Data Browser. Bin cells can also be colored according to the results of the query.

To define a query, click the Edit button in the Options section. The Selection Query Editor is displayed:

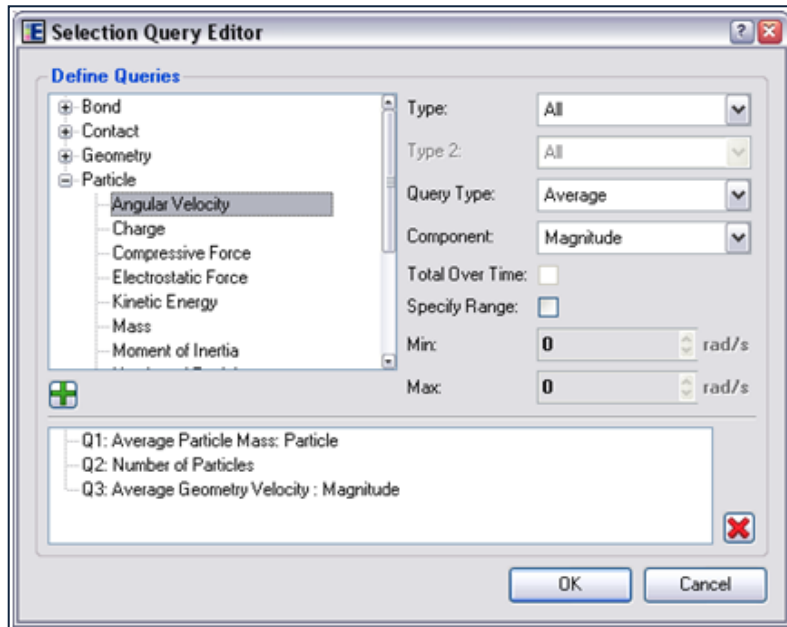


Figure 4-11: Selection Query Editor

1. Select an element and attribute from the list. The elements are categorized by type: contacts, geometry, bonds, and particles. See the table below and *Appendix B: Attribute Definitions* for details.
2. Use the Type/Section control to choose the section (for geometry elements) or type of element the information will be displayed for: For example, an attribute relating to particles of type A, a particular section of geometry or a contact between two specific particle types.
3. The Query type option is used to determine the type of data displayed. Choose from average, maximum, minimum or total: For example the average velocity or the maximum force. This option is only available for certain element attributes (see the tables below for details).
4. Select the component of the attribute you are interested in. Choose from Magnitude, X, Y or Z component: For example, the X component of the particle velocity or the magnitude of the normal force in a surface-surface contact. This option is only available for certain element attributes.
5. If applicable, select the Specify range option to constrain the data collected to a particular value range: For example, particle velocities between 1m/s and 10m/s.
6. Click the Add button once the query has been fully defined. Multiple queries can be created and added to the list. All queries are automatically added to a table in the selection's page in the Data browser.

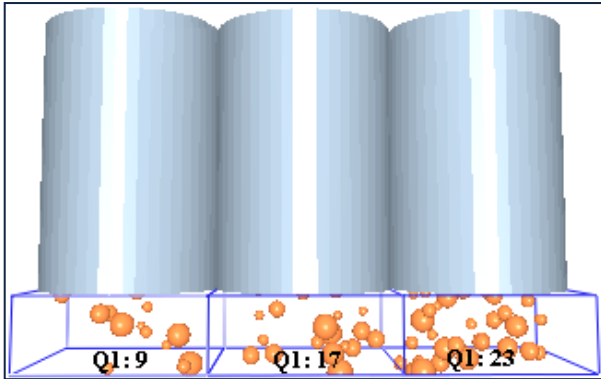


Figure 4-12: A selection with query results (number of particles) displayed on-screen

Contacts

Affects all contacts or any combinations of specific particle-particle or particle-geometry contacts.

Attribute	Query type	Component
Contact vector 1	Average, maximum, minimum or total	Magnitude, X, Y, Z
Contact vector 2	Average, maximum, minimum or total	Magnitude, X, Y, Z
Normal force	Average, maximum, minimum or total	Magnitude, X, Y, Z
Normal overlap	Average, maximum, minimum or total	n/a
Number of contacts	n/a	n/a
Tangential force	Average, maximum, minimum or total	Magnitude, X, Y, Z
Tangential overlap	Average, maximum, minimum or total	n/a
Custom property	Average, maximum, minimum or total	Depends on number of elements

Bonds

Affects all bonds or any combinations of specific particle-particle bonds.

Attribute	Query type	Component
Normal force	Average, maximum, minimum or total	Magnitude, X, Y, Z
Normal moment	Average, maximum, minimum or total	Magnitude, X, Y, Z
No. of broken bonds	n/a	n/a
No. of intact bonds	n/a	n/a
Tangential force	Average, maximum, minimum or total	Magnitude, X, Y, Z
Tangential moment	Average, maximum, minimum or total	Magnitude, X, Y, Z

Geometry

Affects all geometry or a particular section (as specified in the Creator).

Attribute	Query type	Component
Compressive force	Average, maximum, minimum or total	n/a
Number of geometry elements	n/a	n/a
Total force	Average, maximum, minimum or total	Magnitude, X, Y, Z
Torque	Average, maximum, minimum or total	Magnitude, X, Y, Z
Velocity	Average, maximum, minimum or total	Magnitude, X, Y, Z
Custom property	Average, maximum, minimum or total	Depends on number of elements

Particles

Affects all particles or a particular particle type (as specified in the Creator). To cumulatively sum the number of particles over a period of time, select the 'Total over time' option. As you step forward through the simulation the total number of particles will be recorded. Cumulative particle totals for bin groups only works when stepping forward through your simulation: if you jump back, the totals may not be accurate.

Attribute	Query type	Component
Angular Velocity	Average, maximum, minimum or total	Magnitude, X, Y, Z
Compressive force	Average, maximum, minimum or total	n/a
Moment of Inertia	Average, maximum, minimum or total	Magnitude, X, Y, Z
Kinetic energy	Average, maximum, minimum or total	n/a
Mass	Average, maximum, minimum or total	n/a
Number of particles	Total-over-time checkbox	n/a
Potential energy	Average, maximum, minimum or total	n/a
Rotational kinetic energy	Average, maximum, minimum or total	n/a
Torque	Average, maximum, minimum or total	Magnitude, X, Y, Z
Total energy	n/a	n/a
Total force	Average, maximum, minimum or total	Magnitude, X, Y, Z
Velocity	Average, maximum, minimum or total	Magnitude, X, Y, Z
Voidage ¹	Total	n/a
Volume	Average, maximum, minimum or total	n/a
Custom property	Average, maximum, minimum or total	Depends on number of elements

¹ Not available with imported geometry bin groups.

Type Filter

The Type Filter section is used to determine which elements in your model are included. By default all particles (but nothing else) are included in the group. Elements are categorized by type. For any element, click the pulldown menu to select all, none, or selected elements:

Element	Filter Options
Particles	All Particles, No Particles, Selected Particles
Geometry	All Geometry, No Geometry, Selected Geometry
Surface - Geometry Contacts	All Surface-Geometry Contacts, No Surface-Geometry Contacts, Selected Surface-Geometry Contacts
Surface - Surface Contacts	All Surface-Surface Contacts, No Surface-Surface Contacts, Selected Surface-Surface Contacts
Surface - Geometry Bonds	All Surface-Surface Contacts, No Surface-Surface Contacts, Selected Surface-Surface Contacts
Surface - Surface Bonds	All Surface-Surface Bonds, No Surface-Surface Bonds, Selected Surface-Surface Bonds

For selected elements, tick the checkbox for all elements to include.

Representation

Display Mode

For bins and bin groups, the display mode determines how bins are rendered in the viewer.

- **Always** - All bins are always displayed.
- **If Populated** - Bins are only displayed when they contain elements enabled in the type filter.
- **Never** - Bins are not displayed. When several selections exist this can cause confusion, so you can choose this option to turn off specific selections.

Display Options

Display options let you specify which query is displayed in the viewer, as well as set highlighting and coloring options for individual bins (sub-selections). Click the Edit button to open the Display Options.

On Screen Display

For bins and bin groups, tick the On Screen Query checkbox then select which selection group query to display in the viewer.

For grid and geometry template selections, select options to highlight and color sub-selections. Bins can be colored in one of two ways: Highlighting individual bins, or coloring bins by a specific query.

Highlight Sub-Selection

Select which particular bin in the bin group to highlight. Only one bin can be highlighted at a time. Bins in grid bin groups are identified using an ID matrix; bins in imported template selections use the geometry names from the imported file (if available).

Sub-Selection Coloring

Bin groups can be colored according to any pre-defined query: For example, bins could be colored by the number of collisions taking place in them, or by the average speed of the particles they contain.

- Choose a query from the drop-down list.
- Set the value range. This is determined by setting the minimum and maximum query values. The values can be entered directly or can be read from the model by clicking the update buttons. This takes the current maximum and minimum values of the selected query. If the auto-update options are selected, these values will automatically update according to the time step being viewed. Bins will then be colored depending on whereabouts in the range they fall. The colors used are fixed from blue (minimum), through green, to red (maximum).
- Select the Legend option to display a color legend in the Viewer.
- Use the opacity option to vary the opacity of the bin coloring.

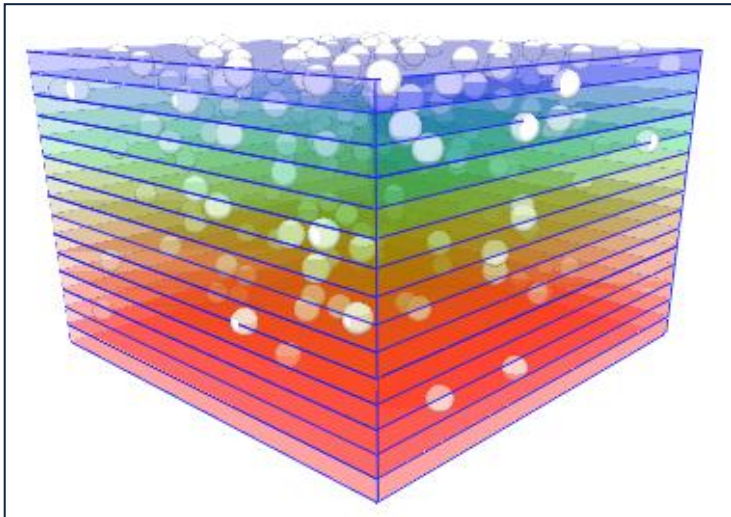


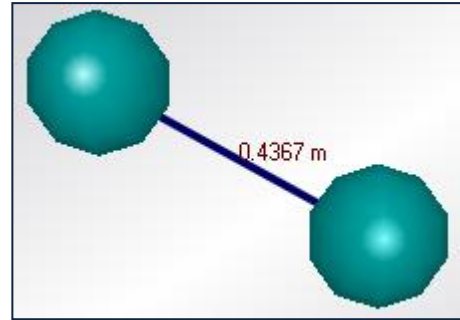
Figure 4-13: A bin group colored by average particle velocity. Bin cells have an opacity of 0.2.

Tools Tab

Ruler

The ruler measures the distance between two points. Use it to display the distance between elements in the Viewer.

1. From the Tools tab, click the Ruler tab.
2. Click the Enable checkbox.
3. For the start point, select what to link to. You can link to a particle, a geometry, or you can manually define a point in space. For example, to link to a particle:
 - a. Click the Link To Particle checkbox.
 - b. Click the Pick Particle button. The cursor changes to a cross-hair.
 - c. Click on a particle in the viewer to select the particle to link to.
4. Repeat the above step to set the end point.

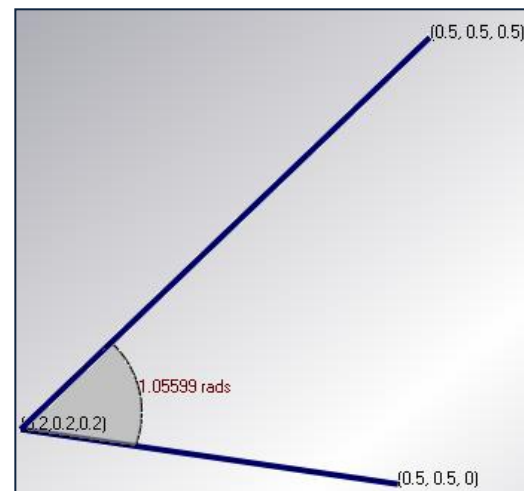


The Current Distance between the two points is displayed at the bottom of the Tools tab.

Protractor

The protractor measures the angle between elements in the viewer.

1. From the Tools tab, click the Protractor tab.
2. Click the Enable checkbox.
3. For the start point, select what to link to. For example, to link to a particle:
 - a. Click the Link To Particle checkbox.
 - b. Click the Pick Particle button. The cursor changes to a cross-hair.
 - c. Click on a particle in the viewer to select the particle to link to.
4. Repeat the above step to set the end point and axis.



The Current Angle between the two points is displayed at the bottom of the Tools tab.

Domain Grid

The domain grid is a visual aid used to indicate the scale of your model. A grid can be added to each side of the domain (+X, +Y, +Z, -X, -Y and -Z). The number of intervals in each grid is also specified. The coordinates of each vertex and the length of each edge are shown.

The figure below shows a model with 10 interval +X and +Y grids displayed:

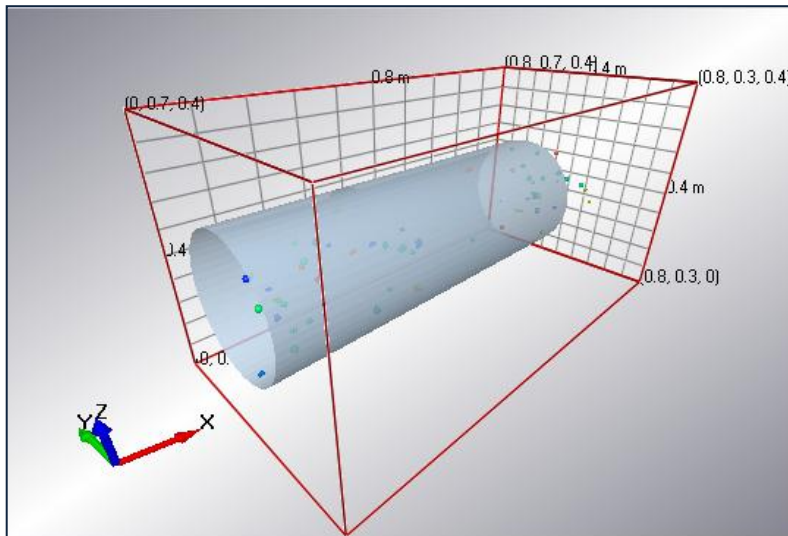


Figure 4-21: Example Model with Domain Grid

Clipping

Clipping is used to remove ('clip') areas of the model that are not currently of interest. A model can be clipped in three ways - using planes, slices or the domain boundaries. A clip group is a collection of one or more planes, slices or domain boundaries (though it can only contain clips of one type). A model may have any number of clip groups: For example, the model of a pipe may have two clip groups, one containing different slices of the pipe and another containing an area of pipe constrained by a number of planes.

Create a clip group by clicking the **+** button, then choose the clipping type from the Active Sets drop-down list. The planes, slices or boundaries can then be defined in the lower half of the Clipping Elements section. The top half of this section contains the list used to determine which elements within your model are affected by the clipping group: For example, you may wish to only clip the geometry and leave the particles intact. By default all elements are included in the group and hence are colored green. The elements are categorized by type - particles, geometry, contacts, and bonds. Double click on any element to remove it from the group. Its listing will turn black indicating its exclusion. An element can be added into the group by double clicking on it a second time.

The figure below shows a model of a pipe where geometry has been included in a slice and particles have been excluded.

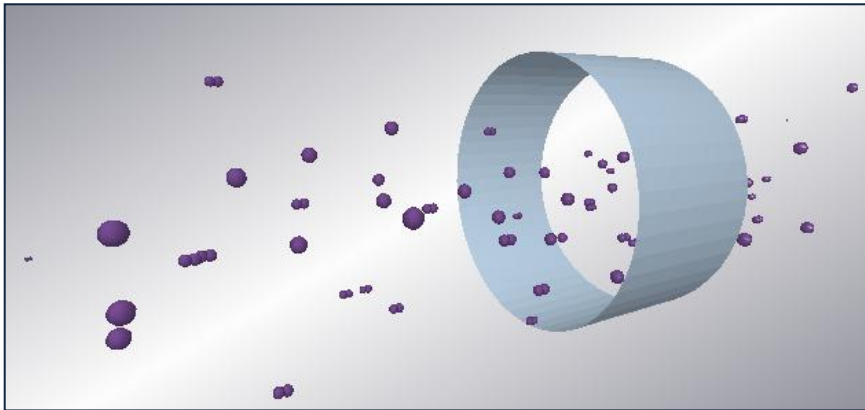


Figure 4-14: Sliced pipe

Planes

A planes clipping group can contain up to six individual planes. Choose a plane from the Elements drop-down list in the Clipping Elements section. The plane is then defined by its orientation and its distance from the center of the model. The section of the model that falls on one side of the plane will be clipped from the model. Simply negate the orientation of the plane to switch which side of the model is removed.

The Visual Representation option can be used to display a preview of the plane in the Viewer. Select the Enable Plane option to activate the plane and clip a section from the model.

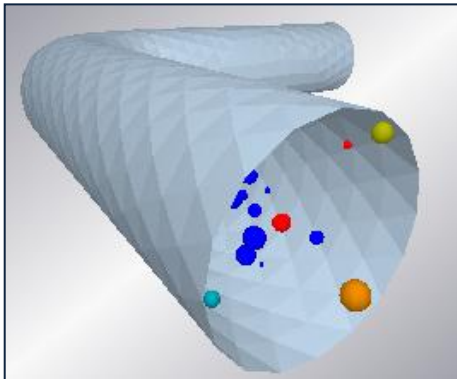


Figure 4-15: A Model Before Clipping

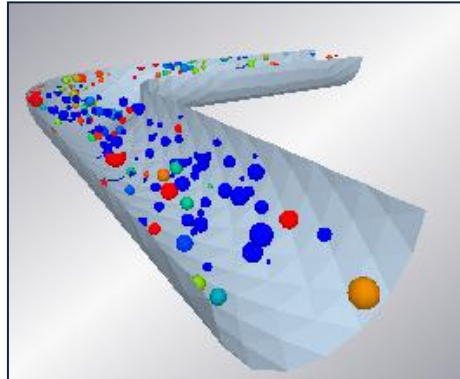


Figure 4-16: A Model After Plane Clipping

Slices

A slices clipping group can contain up to three individual slices. Choose a slice from the Elements drop-down list in the Clipping Elements section. The slice is then defined by its orientation, distance from the center of the model and depth. The section of the model that falls outside the slice will be clipped from the model. The Visual Representation option can be used to display a preview of the slice in the Viewer. Select the Enable Slice option to activate the slice and clip an area of the model.

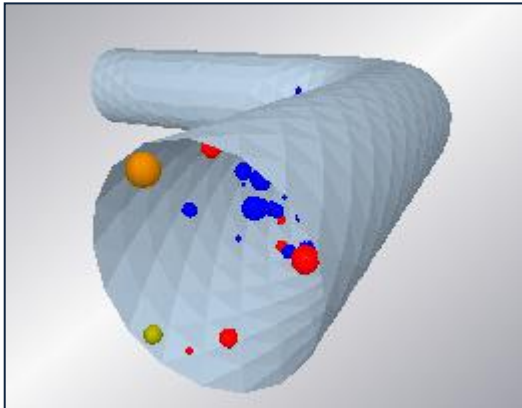


Figure 4-17: A Model Before Clipping

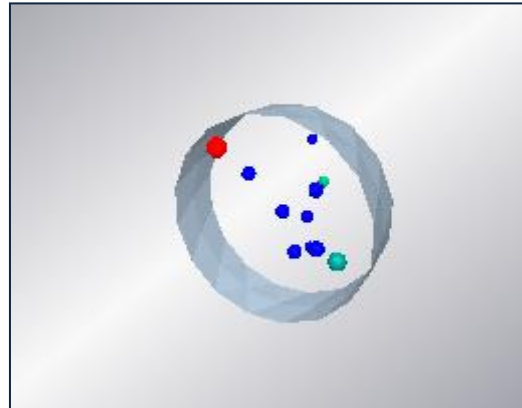


Figure 4-18: A Model After Slice Clipping

Domain Boundaries

Each side of your domain can be used to clip your model in a Domain Boundaries clipping group. Choose a side from the Elements drop-down list in the Clipping Elements section and select the Enable Slice option to clip the area of the model outside of the domain on that side.

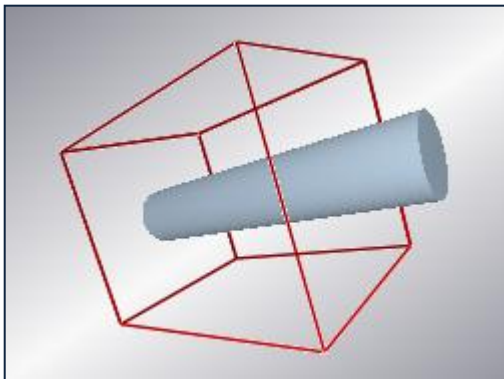


Figure 4-19: A Model Before Clipping

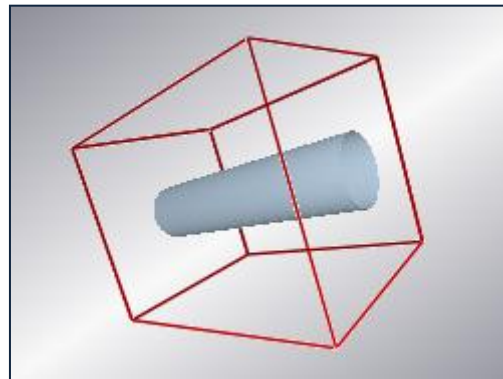


Figure 4-20: A Model After Clipping

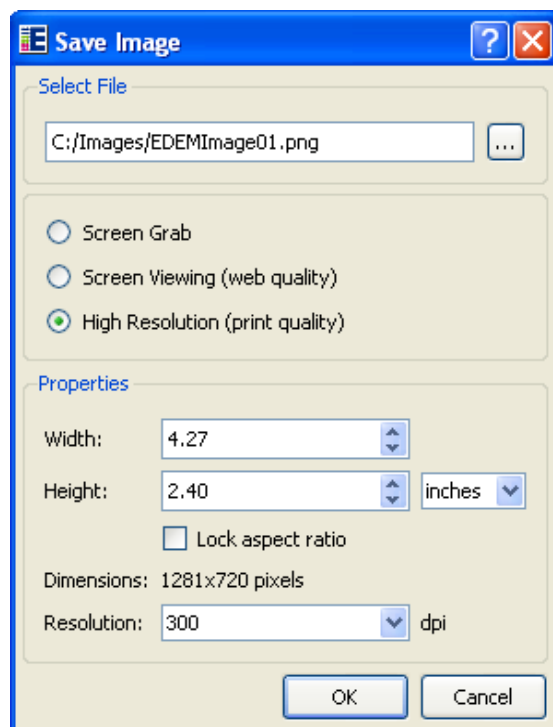
Saving Images

Full-color images can be exported at a single time step or over a range of time steps.

Saving a Single Image

The image displayed in the Viewer can be exported and saved as a .png (table network graphic) or .bmp (bitmap).

1. Use the animation controls to choose the image you wish to export.
2. Use the camera controls to set the angle.
3. Use the controls in the Tools menu to turn on or off the display of the axis key, bounding box or timestamp. Increase the particle detail to improve the appearance of particles.
4. Select File > Export > Image. The Save Image dialog will appear.
 - **Select Filename.** Set the file name and save location.
 - **Quality.** Choose one of the following:






1. **Screen Grab.** If this option is selected, the image exported will be the same dimensions as the Viewer.
2. **Screen Viewing.** If this option is selected, the image exported is web quality.

3. High Resolution. If this option is selected, the image exported is print quality, and width and height can be specified in inches, millimeters, or pixels. Also, the resolution can be specified in dpi.

- **Width.** Set the image width if the screen grab option is not selected.
- **Height.** Set the image height if the screen grab option is not selected.

Saving Multiple Images

To save a range of time steps:


1. Use the animation controls to choose the first image in the range and display it in the Viewer. Images will be exported from this point onward.
2. Set the step factor in the Viewer controls pane. This determines how often an image is exported: for example every fifth time step or every 0.0001s.
3. Use the controls in the Tools menu to set the axis key, boundary box and time stamp. Increase the particle detail to improve the appearance of particles.
4. Click the blue record animation button  in the viewer. The Export Images/Video dialog is displayed.
 - **Export Format.** Set the export format to Images.
 - **Filename.** Set the base file name and save location. Each image will be given the base name plus a numeric suffix.
 - **Frame Width.** Set the width of each image. For example, 640 pixels.
 - **Frame Height.** Set the height of each image. For example, 480 pixels.
 - **Set to Screen Size.** Click the screen grab button  to set the width and height to match the screen size. The image exported will be the same dimensions as the Viewer.
5. Click OK. The record animation button turns red: .
6. Use the animation controls to play or step through the simulation.
7. When finished, click the record animation button again to stop recording.

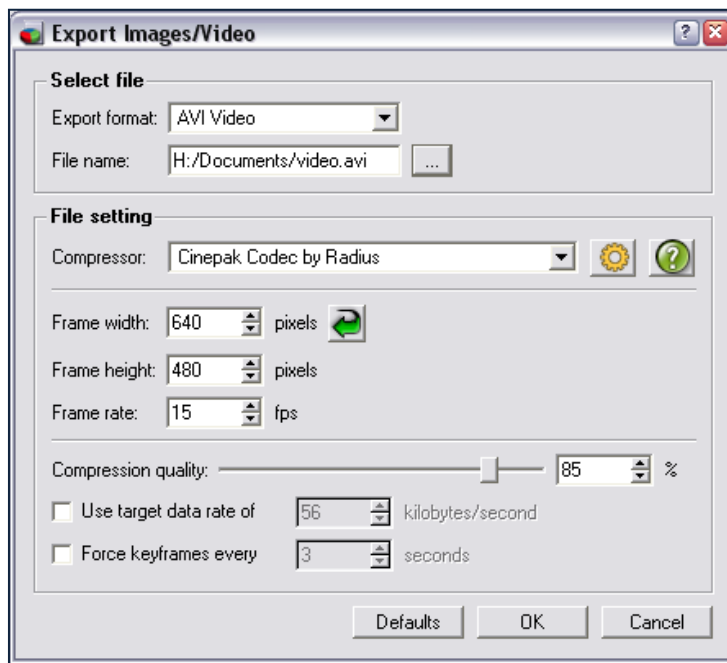
Creating Videos

EDEM can export videos as AVI, WMV (Windows Media Video), or EnSight Video files. You can specify encoding quality, frame rates and which compressor (codec) to use. A video can contain multiple display modes (for example, change the camera angle and particle representation halfway through the video) and even contain different simulations in the same video.

Prepare to Record a Video

1. Use the animation controls to select a start point. The video will be exported from this point onward.
2. If necessary, set the step factor in the Viewer controls pane.

3. Use the controls in the Tools menu to set display of the axis key, boundary box and time stamp. Increase the particle detail to improve the appearance of particles.
4. Click the blue record animation button  in the viewer. The Export Images/Video dialog is displayed:



Video Configuration


Export Format

Set the export format to either EnSight Video, AVI Video, or Windows Media Video.

- Select EnSight Video to save the video as a .evo file. The video can then be viewed using CEI's EnVideo, or edited using EnVe.
- Select AVI Video to save the video as a .avi file. Save your video as an AVI when you want to use a particular codec (compressor) or when you need a high degree of video configuration. You can use any codec installed on your system (for example, Cinepak, DivX, Xvid, or H.264).
- Select Windows Media Video (WMV) to use Microsoft's video codecs. A WMV file is better suited for sharing, since it should play on any PC without the need to install video codecs. You can also set a constant bitrate, useful for streaming the video from a website.

Select Codec


Select the codec to use or select uncompressed to not use any video compression. The list of codecs available depends on those installed on your system and on which operating system you are running EDEM. Be sure the system on which you intend to playback the video also has the same codec installed.

- For AVI Video, select a codec then (if available) click the configure button to set any additional codec options. 
- For EnSight Video, select Motion-RLE for lossless video files or select Motion-JPEG for compressed video.


NOTE: DEM Solutions does not provide support for external codecs developed by third parties. Refer to the codec's documentation or developer's website for assistance.

Video Options

Set the frame options and encoding quality as necessary.

- **Frame Width.** Set the width of the video. For example, 640 pixels.
- **Frame Height.** Set the height of the video. For example, 480 pixels.
- **Frame Rate.** Set the frame rate. For example, 25 fps. A lower frame rate decreases the size of the video.
- **Set to Screen Size.** Click the screen grab button  to set the width and height to match the screen size. The video exported will be the same dimensions as the Viewer.
- **Compression Quality.** Use the slider to set the encoding quality (where available). The higher the quality, the larger the size of the video.
- **Use Target Data Rate.** Tick this checkbox to specify a Constant Bitrate (CBR). You would typically use this when you want to stream the video from a website.
- **Force Keyframes.** Tick this checkbox then set how often to write a keyframe to the video. Because video compression only stores incremental changes between frames, forcing keyframes can result in a video that is easier to fast-forward or rewind.

Record the Video

1. Click OK. The record animation button turns red: 
2. Use the animation controls to play or step through the simulation. While recording, you can change whatever is displayed in the viewer. Any changes are recorded in the video. For example, you can:
 - Change the camera or move, rotate and zoom the display
 - Change the display mode, opacity, and particle representation
 - Modify attribute coloring and legends
 - Add different labels at different timesteps
 - Load a completely different simulation
3. When finished, click the record animation button again to stop recording.

Use a media player (such as Windows Media Player or VLC media player) to view the saved video.

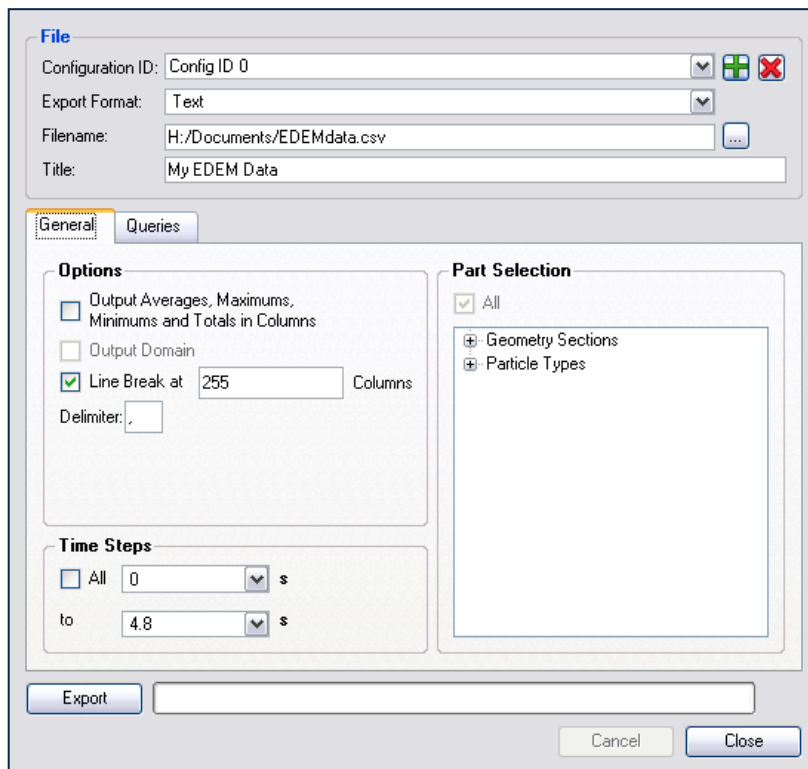
Exporting Data

Data collected during your simulation can be exported for analysis. Data can be exported from a single time step, a range of time steps, from all elements in your model or just from a selection or bin group.

Exporting data is carried out using queries. A query is used to define a single element attribute: For example, a query could be set up to extract the x-position of particles of type A in your model. Multiple queries can be grouped together under one Configuration ID and exported to a single file.

Creating a Configuration

Export data using the Export Data Dialog (File > Export > Results Data).



1. Click on the + button to create a new Configuration ID. Multiple queries can be created within a single Configuration. In addition, multiple configurations can be created for any model: For example, one configuration may contain a number of queries relating to particles whilst another might have queries solely relating to contacts. Configurations can be edited at any point: For example, new queries added or redundant ones removed.
2. Select the output file format: Enight (.case) or Text (.csv). For Enight files, be sure the total number of characters (filename and directory) is not more than 72.
3. Give the configuration a title and enter a file name and location.

General Settings

Options

Output Averages, Maximums, Minimums and Totals in Columns.

This determines how data is laid out in the exported file. If this option is selected, data will be separated and extracted in unlabelled columns. The figures below both show a sample of some exported data. The data is the same in each case. In the first figure, the data is presented with this option not selected. The second figure shows it with the option selected.

```
TIME: ,0.900039
Q02: Average Particle Velocity Magnitude: ,4.99865
Q03: Maximum Particle Velocity Magnitude: ,5.10404

TIME: ,0.910045
Q02: Average Particle Velocity Magnitude: ,4.99865
Q03: Maximum Particle Velocity Magnitude: ,5.10404
```

Figure 4-23: Column Option Not Selected

```
TIME,Q02: Average Particle Velocity Magnitude,Q03: Maximum Particle Velocity Magnitude
0.900039,4.99865,5.10404
0.910045,4.99865,5.10404
```

Figure 4-24: Column Option Selected

Output Domain

This option is only available when exporting data to an EnSight file. If selected, information on the domain will be exported with the data. If it is not, a new domain will be created automatically when the file is loaded in EnSight.

Delimiter and Line Break

If you are exporting to a text file, optionally change the delimiter to use and set whether to start a new line after X columns. By default, EDEM inserts a line break at 256 columns (the maximum allowable by Microsoft Excel).

Part Selection

This section is only active when exporting data to an EnSight file. Choose to export data on all elements in your model or just from specified ones. The elements are categorized by geometry sections and particle types. Information on contacts cannot be exported. Double click on any element to remove it from the group. Its listing will turn black indicating its exclusion. An element can be added into the list by double clicking on it a second time.

Time Steps

Data can be extracted from all time steps in your simulation or from a defined range. If a time step is from a partial save (indicated by the timestep being grayed-out), data for the selected attributes may not be available. This is indicated by n/a in the exported file.

Creating a Query

Queries define what information is exported. A configuration can contain multiple queries, each of which defines one data attribute.

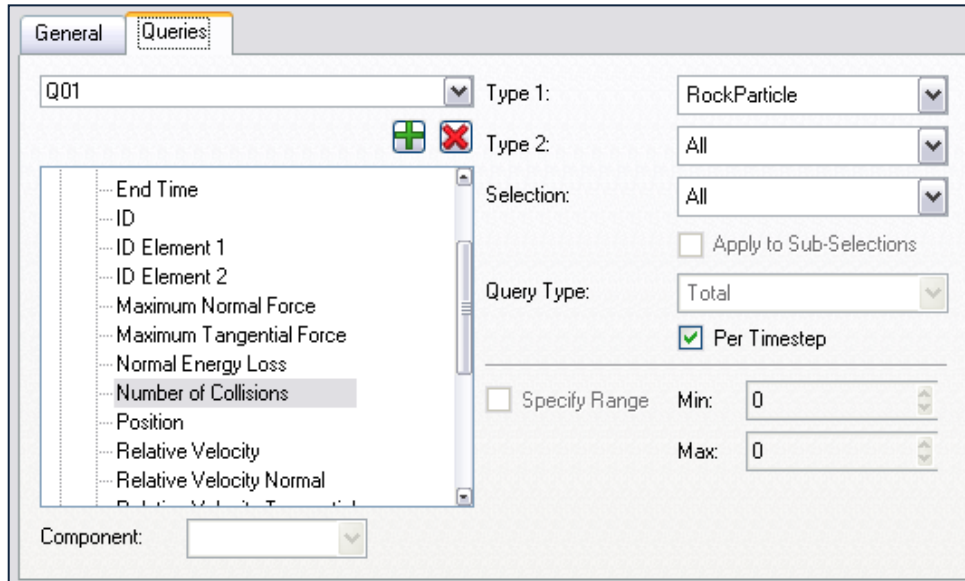


Figure 4-25: The Query Tab

Click on the Queries tab and click the **+** button to create a new query. The attributes that can be exported are listed under the element they relate to (collisions, contacts, particles or geometry). Click on any attribute to select it and then select the component of that attribute you are interested in: For example the magnitude of a contact vector. All available attributes and their components are listed in the tables below.

Next choose the geometry section or type of particle or contact you want to export data for: For example choose only collisions between certain particle types, or the force on a particular geometry section.

Data can be extracted on all elements of the selected type (or geometry section) or it can be limited to just those contained within a particular bin group or selection group. This can be selected using the Selection drop-down list where all bin and selection groups that have been created are listed. Enable the Apply to Sub-Selections checkbox to export a separate query for each bin in a group (if defined), rather than one combined query for the entire group.

The Query Type option is used to determine which data values will be exported for a particular attribute: For example, if the query has been set up to export the velocity of particles of type A you could choose to export the maximum, minimum or average velocity per time step. The following data is exported for each query type:

- **Standard.** The value of the attribute for every element of the selected type for each time step in the defined range: For example, the velocity of every particle at each time step.
- **Total.** The sum of the values of the attribute for every element of the selected type: For example, the total velocity of all particles per time step. The total can be calculated per time step or over the range specified in the General settings section.

- **Maximum.** The maximum value of the attribute across all elements of the selected type for each time step in the defined range: For example, the maximum particle velocity per time step. The maximum can be calculated and exported per time step or over the range specified in the General settings section.
- **Minimum.** The minimum value of the attribute across all elements of the selected type for each time step in the defined range: For example, the minimum particle velocity per time step. The minimum can be calculated and exported per time step or over a specified range.
- **Average.** The average value of the attribute across all elements of the selected type for each time step in the defined range: For example, the average particle velocity per time step. The average can be calculated per time step or over a specified range.

The final option is Specify Range. This is used to specify a value range for the data exported. In other words, limiting the export to data that falls between two specified values. For example, only exporting particle velocity data for particles traveling between 3m/s and 5m/s.

Further queries can be created in the same way. Once all queries have been set up, click the Export button to export your data to your specified file. Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Collision Elements

Affects all or any combination of specific particles types or geometry sections.

Attribute	Components	Query Type
Average normal force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Average tangential force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Duration	n/a	Standard, Avg., Min. or Max.
End time	n/a	n/a
ID	n/a	n/a
ID Element n	n/a	n/a
Maximum normal force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Maximum tangential force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Normal energy loss	n/a	Standard, Avg., Total, Min. or Max.
Number of collisions	n/a	n/a
Position	X, Y, Z	n/a
Relative velocity	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Relative velocity normal	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Relative velocity tangential	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Start time	n/a	n/a
Tangential energy loss	n/a	Standard, Avg., Total, Min. or Max.
Total energy loss	n/a	Standard, Avg., Total, Min. or Max.
Velocity of element A	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Velocity of element B	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.

Contacts Elements

Affects all or any combination of specific particles types or geometry sections.

Attribute	Components	Query Type
Contact vector 1	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Contact vector 2	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Normal force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Normal overlap	n/a	Standard, Avg., Total, Min. or Max.
Number of contacts	n/a	Avg., Total, Min. or Max.
Position	X, Y or Z	n/a
Tangential force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Tangential overlap	n/a	Standard, Avg., Total, Min. or Max.
Custom property	Depends on number of elements	Standard, Avg., Total, Min. or Max.

Bond Elements

Affects all or any combination of specific particles types.

Attribute	Components	Query Type
End time	n/a	n/a
Normal force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Normal moment	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
No. of broken bonds	n/a	n/a
No. of intact bonds	n/a	n/a
Tangential force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Tangential moment	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.

Geometry Elements

Affects all or any individual section (as specified in the Creator).

Attribute	Components	Query Type
Compressive force	n/a	Standard, Avg., Total, Min. or Max.
ID	n/a	n/a
Node n	X, Y, Z	n/a
Position	X, Y, Z	n/a
Surface Normal	Magnitude, X, Y, Z	n/a
Torque	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Total force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Velocity	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Custom property	Depends on number of elements	Standard, Avg., Total, Min. or Max.

Particle Elements

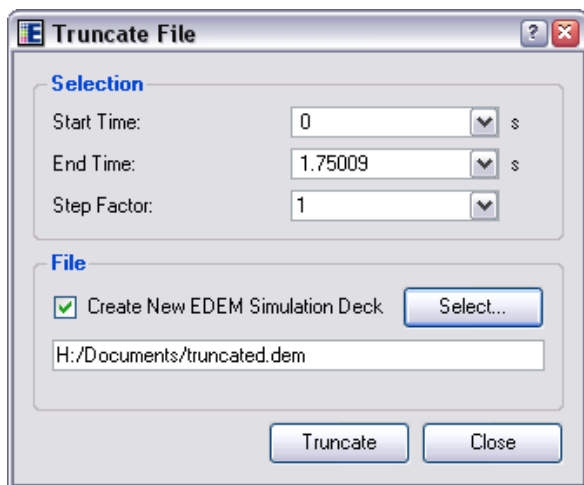
Affects all or any individual particle type (as specified in the Creator).

Attribute	Components	Query Type
Angular velocity	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Compressive force	n/a	Standard, Avg., Total, Min. or Max.
ID	n/a	n/a
Moment of Inertia	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Kinetic energy	n/a	Standard, Avg., Total, Min. or Max.
Mass	n/a	Standard, Avg., Total, Min. or Max.
Number of particles	n/a	Avg., Total, Total over time, Min. or Max.
Orientation	n/a	n/a
Position	X, Y or Z	n/a
Potential energy	n/a	Standard, Avg., Total, Min. or Max.
Rotational kinetic energy	n/a	Standard, Avg., Total, Min. or Max.
Torque	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Total energy	n/a	Standard, Avg., Total, Min. or Max.
Total force	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Velocity	Magnitude, X, Y, Z	Standard, Avg., Total, Min. or Max.
Volume	n/a	Standard, Avg., Total, Min. or Max.
Custom property	Depends on number of elements	Standard, Avg., Total, Min. or Max.

Truncating Results

The Truncate option is used to remove results data (time steps) from your simulation. Data can be removed entirely, or simply 'thinned out' by removing every n^{th} result.

Select File > Truncate File. The Truncate file dialog is displayed:



Set the Start and End Times. This defines the times steps between which you would like to remove data. Only full time steps can be truncated: you cannot truncate time steps from partial saves.

Set the Step Factor. A factor can be selected from the list or a new value entered directly. This determines how often a time step in the range is removed. If the step factor is set to one, every time step will be removed thus removing all data in the specified range. If it is set to greater than one, the data in the range will effectively be 'thinned out'. The greater the step factor, the more data is removed.

By default the selected data will be truncated from the current simulation deck. Tick *Create New EDEM Simulation Deck* to truncate to a new simulation deck. The original deck is unaffected.

Graphing

Creating Graphs

Data collected during your simulation can be graphed in a number of ways. Select the Analyst then switch between 3D Viewer and graph mode using the toolbar buttons:



Figure 4-26: The 3D Viewer and Graph Buttons

When you select graph mode, the Graph tab replaces the standard Tabs pane and the graph is displayed in the Viewer. The Graph pane has four individual tabs, one for each type of graph:

- Histogram
- Line Graph
- Scatter Plot
- Pie Chart

Note: Only one graph of each type can be created at any one time.

Creating a Histogram (Bar Chart)

A histogram is a graphical display of tabulated frequencies. For example, the figure below is a histogram which shows the number of particles traveling in particular velocity ranges. A tabular version of the data is also included.

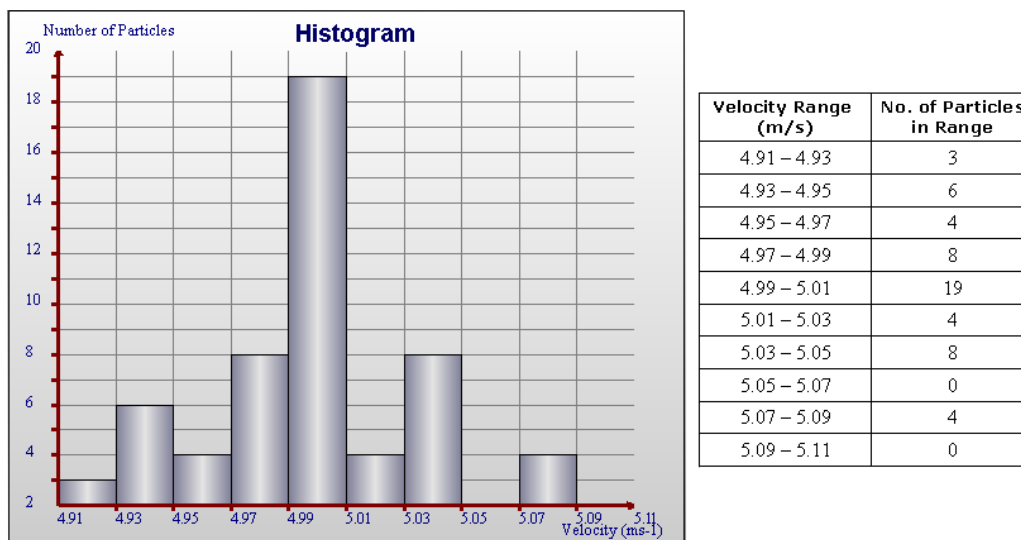


Figure 4-27: Data Showing the Distribution of Particle Velocities

Selecting the Elements to Graph

The Select Element section is used to determine which elements to collect data from. Choose the group or type of elements you are interested in.

Group	Type
Particle	All or any individual particle type (as specified in the Creator)
Geometry	All or any individual section (as specified in the Creator)
Contact	All or any combination of specific particle types or geometry sections
Collision	All or any combination of specific particle types or geometry sections
Bond	All or any combination of specific particle types

Configuring the X-Axis

Click on the X-axis tab then select the attribute and component to plot on the x-axis. The attributes available in the list will depend on the elements previously selected. The table below shows the attributes and components available for each element. Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Element	Attribute	Components
Particle	Angular velocity	Magnitude, X, Y, Z
	Compressive force	n/a
	Distance	Define reference object*
	Kinetic energy	n/a
	Mass	n/a
	Potential energy	n/a
	Rotational kinetic energy	n/a
	Time	n/a
	Torque	Magnitude, X, Y, Z
	Total energy	n/a
	Total force	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Volume	n/a
Custom property	Depends on number of elements	
Geometry	Time	n/a

Element	Attribute	Components
Contact	Contact vector 1	Magnitude, X, Y, Z
	Contact vector 2	Magnitude, X, Y, Z
	Distance	Define reference object*
	Normal overlap	n/a
	Normal force	Magnitude, X, Y, Z
	Tangential force	Magnitude, X, Y, Z
	Tangential overlap	n/a
	Time	n/a
	Custom property	Depends on number of elements
Collision	Average normal force	Magnitude, X, Y, Z
	Average tangential force	Magnitude, X, Y, Z
	Distance	n/a
	Duration	n/a
	Maximum normal force	Magnitude, X, Y, Z
	Maximum tangential force	Magnitude, X, Y, Z
	Normal energy loss	n/a
	Number of collisions	n/a
	Relative velocity	Magnitude, X, Y, Z
	Relative velocity normal	Magnitude, X, Y, Z
	Relative velocity tangential	Magnitude, X, Y, Z
	Tangential energy loss	n/a
	Time	n/a
	Total energy loss	n/a
	Velocity element A**	Magnitude, X, Y, Z
Velocity element B	Magnitude, X, Y, Z	
Bond	Normal force	Magnitude, X, Y, Z
	Normal moment	Magnitude, X, Y, Z
	Tangential force	Magnitude, X, Y, Z
	Tangential moment	Magnitude, X, Y, Z
	Time	n/a

* If the attribute is set to Distance you must define a point or plane from which the distance is measured. When distance is selected the **Define Reference Object** section of the pane will be activated. Choose Point or Plane and define its position and, for a plane, its distance from the origin.

** The velocity of the first element in the collisions. Similarly for Velocity element B.

The x and y axes are related. The attribute you choose to measure on the x-axis will limit those available to measure on the y-axis. Similarly if the y-axis is configured first the attributes available on the x-axis will be limited.

Configuring the Y-Axis

Click on the Y-axis tab and select the element attribute and component to be plotted on the y-axis. The attributes available in the list will depend on the elements previously selected and the attribute being measured on the x-axis. A component type can also be selected for certain attributes. This is used to determine which value will be graphed for a particular attribute component: For example, maximum, minimum or average particle velocity. The table below shows the range of attributes, components and component types available for each element. Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Element	Attribute	Components	Type
Contact	Contact vector 1	Magnitude, X, Y, Z	Average, Max or Min
	Contact vector 2	Magnitude, X, Y, Z	Average, Max or Min
	Distance	Define ref. object*	Average, Max or Min
	Normal force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Normal overlap	n/a	Average, Max or Min
	Number of contacts	n/a	Total or Total in Range**
	Tangential force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Tangential overlap	n/a	Average, Max or Min
	Custom property	Depends on no. of elements	Average, Max, Min, or Total
Collision	Average normal force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Average tangential force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Distance	Define ref. object*	Average, Max or Min
	Duration	n/a	Average, Max or Min
	Maximum normal force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Maximum tangential force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Normal energy loss	n/a	Average, Max, Min, or Total
	Number of collisions	n/a	Total or Total in Range**
	Relative velocity	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Relative velocity normal	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Relative velocity tangential	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Tangential energy loss	n/a	Average, Max, Min, or Total
	Total energy loss	n/a	Average, Max, Min, or Total
	Velocity of element A	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Velocity of element B	Magnitude, X, Y, Z	Average, Max, Min, or Total

Element	Attribute	Components	Type
Geometry	Compressive force	n/a	Average, Max, Min, or Total
	Distance	n/a	Average, Max or Min
	Torque	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Total force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Velocity	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Custom property	Depends on no. of elements	Average, Max, Min, or Total
Particle	Angular velocity	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Compressive force	n/a	Average, Max, Min, or Total
	Distance	Define ref. object*	Average, Max or Min
	Kinetic energy	n/a	Average, Max, Min, or Total
	Mass	n/a	Average, Max, Min, or Total
	Number of particles	n/a	Total or Total in Range**
	Potential energy	n/a	Average, Max, Min, or Total
	Rotational kinetic energy	n/a	Average, Max, Min, or Total
	Torque	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Total energy	n/a	Average, Max, Min, or Total
	Total force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Velocity	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Volume	n/a	Average, Max, Min, or Total
	Custom property	Depends on no. of elements	Average, Max, Min, or Total
Bond	Normal force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Normal moment	Magnitude, X, Y, Z	Average, Max, Min, or Total
	No. of broken bonds	n/a	Total, Total in range
	No. of intact bonds	n/a	Total, Total in range
	Tangential force	Magnitude, X, Y, Z	Average, Max, Min, or Total
	Tangential moment	Magnitude, X, Y, Z	Average, Max, Min, or Total

* If the attribute is set to Distance you must define a point or plane from which the distance is measured. When distance is selected the **Define Reference Object** section of the pane will be activated. Choose the point or plane from which the distance should be measured. A point is defined by its xyz position and a plane by its orientation and distance from the origin.

** When the axis is set to measure the number of particles or number of contacts, it is possible to graph either the total number of particles/contacts or the total number in a defined range: For example, counting only those particles with a mass between 0.1kg and 0.2kg or velocity between 2m/s and 5m/s. When the Total in Range option is selected the **Secondary Attribute** section of the pane is activated. Choose the attribute and component you wish to restrict the range by and enter the maximum and minimum values of that range.

Settings

Time Step

Graphs can be created for elements in a particular time step or over a range of time steps. By default the Current Time step option is selected. The current time step can be changed using the Current Time control in the Viewer control pane at the right of the screen. For example, the figure below shows two histograms each showing the number of particles traveling in particular velocity ranges. The graph on the left is for one time step, so each particle is only counted once. The graph on the right is for a range of ten time steps. The velocity of the particles may change between time steps and therefore be counted more than once: For example, a particle traveling at 4.92 m/s in the first time step and 5.0 m/s for all subsequent time steps will be counted once in the 4.91-4.93 column and nine times in the 4.99-5.01 column.

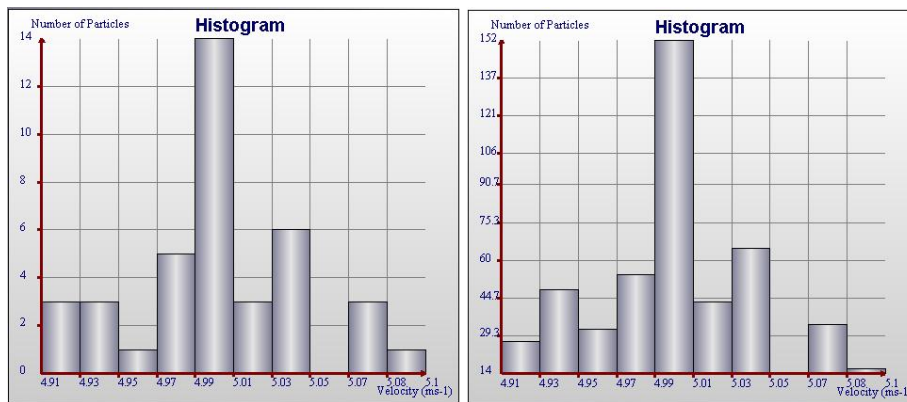


Figure 4-28: Histograms Showing the Distribution of Particle Velocities.

Unselect the Current Time Step to set a different start and end time. Grayed-out timesteps indicate partial saves and may not contain all the data you want to plot.

Axis Range

The x and y axis ranges can be determined automatically (from the values of the attributes selected) or entered manually. The number of intervals on each axis should also be defined. A greater number of intervals increases the accuracy of the chart.

Display Options

Click the Display Options button to open the dialog.

- **Title.** Set the graph title.
- **X-label, Y-label.** Can be determined automatically or entered manually.
- **Show grid.** Turn the interval grid on and off.
- **Show average value.** Turn the display of a horizontal line showing the average value on and off.
- **Show minimum value.** Turn the display of a horizontal line showing the minimum value on and off.
- **Show maximum value.** Turn the display of a horizontal line showing the maximum value on and off.

Create the Graph

Create the graph once the axes have been configured and the time step and display options set. Click on the Create Graph button to draw the graph. Use the Clear Graph button to clear it. If any setting is altered (including changing the units) click the Create Graph button to view the graph with the changes.

To save the graph data, right-click on the graph then select Export as Text.

Drawing a Line Graph

A line graph is used to measure any variable over time. That is, how the value of a particular variable changes over a specified time.

For example, the figure below is a line graph which shows the number of particles present in the model over a particular time. It clearly shows a large number of particles created in the first 0.0334s followed by a small increase over the next 0.0666s and finally a second large increase in the final 0.034s.

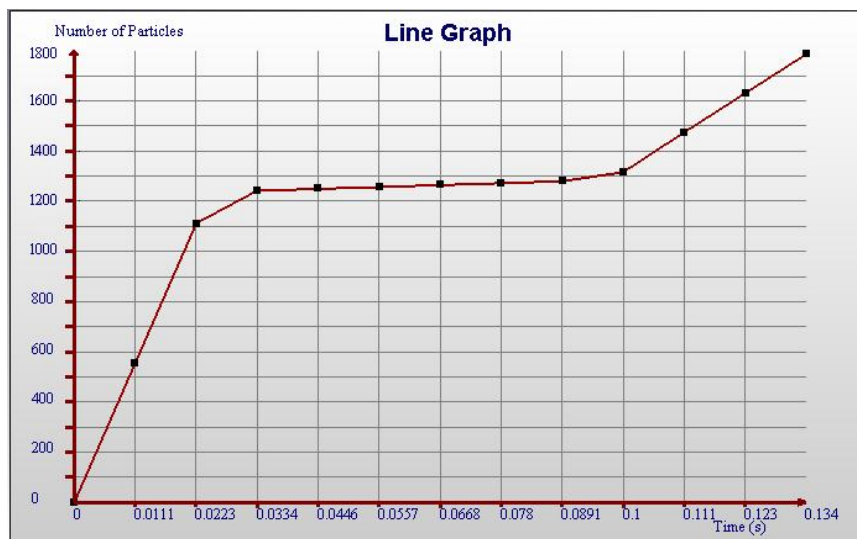


Figure 4-29: Line Graph Showing the Number of Particle in a Model

Selecting the Elements to Graph

The Select Element section is used to determine which elements in your model to collect data from. Choose the group of elements, the type of elements within that group and which selection of those elements you are interested in.

Configuring the X-Axis

Click on the X-axis tab. In a line graph the X-axis always measures time. Use the Start and End controls to set the range of time over which to plot your data. Grayed-out timesteps indicate partial saves and may not contain all the data you want to plot.

Set the number of intervals on the axis: a greater number of intervals increases the accuracy of the chart. Set the maximum number of points to display. Reducing the number of points may make the graph more readable.

Configuring the Y-Axis

Click on the Y-axis tab and select the element attribute and component to be plotted on the y-axis. The attributes available in the list will depend on the elements previously selected. A component type can also be selected for certain attributes. This is used to determine which value will be graphed for a particular attribute component: For example, maximum, minimum or average particle velocity.

Note that if you are graphing the number of elements (i.e. number of particles, contacts or collisions) then any interpolated data points (those constructed from the discrete set of known data points) are rounded down to the nearest whole number. Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Element attribute options are the same as for a Histogram (with the exception of collisions). See *Configuring the Y-Axis* on page 97 for details.

Settings

See *Settings* on page 99.

Drawing a Scatter Plot

A scatter plot is a way of displaying the distribution of two variables in relation to each other. The value of one variable is measured on the X axis and the values of the other on the Y axis. A wide scatter of the plots denotes poor correlation between the two variables. If the two variables are perfectly correlated, then all the plots will fall on the diagonal (regression line).

For example, the figure below is a scatter plot which shows the velocity of a set of particles as they travel away from a specified point. The plots are clearly scattered around the regression line indicating that the two variables are well correlated. The further a particle travels from the specified point, the greater its velocity.

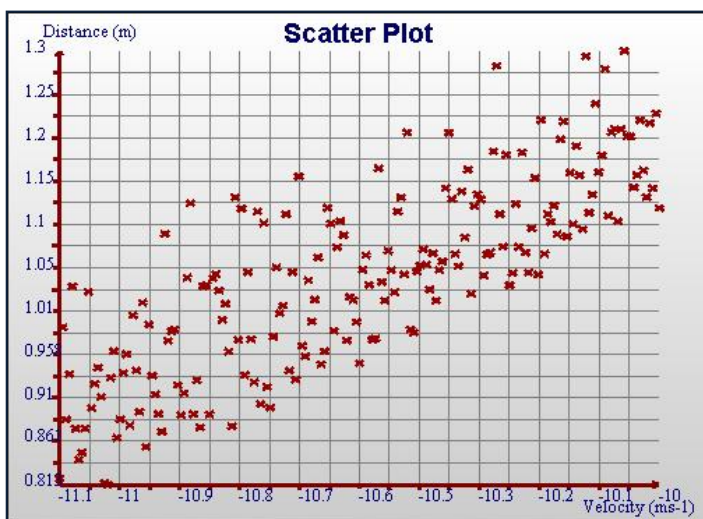


Figure 4-30: Scatter Plot Showing Two Well Correlated Variables

Selecting the Elements to Graph

The Select Element section is used to determine which elements in your model to collect data from. Choose the group of elements, the type of elements within that group and which selection of those elements you are interested in.

Configuring the X-Axis

Click on the X-axis tab and select the element attribute and component to be plotted on the x-axis. The attributes available in the list will depend on the elements previously selected. Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Element	Attribute	Components
Contacts	Contact vector 1	Magnitude, X, Y, Z
	Contact vector 2	Magnitude, X, Y, Z
	Distance	Define reference object*
	Normal force	Magnitude, X, Y, Z
	Normal overlap	n/a
	Position	X, Y, Z
	Tangential force	Magnitude, X, Y, Z
	Tangential overlap	n/a
	Custom property	Depends on number of elements
Collisions	Average normal force	Magnitude, X, Y, Z
	Average tangential force	Magnitude, X, Y, Z
	Distance	n/a
	Duration	n/a
	Maximum normal force	Magnitude, X, Y, Z
	Maximum tangential force	Magnitude, X, Y, Z
	Normal energy loss	n/a
	Position	X, Y, Z
	Relative velocity	Magnitude, X, Y, Z
	Relative velocity normal	Magnitude, X, Y, Z
	Relative velocity tangential	Magnitude, X, Y, Z
	Tangential energy loss	n/a
	Total energy loss	n/a
	Velocity element A**	Magnitude, X, Y, Z
Velocity element B	Magnitude, X, Y, Z	
Geometry	Compressive force	n/a
	Distance	n/a
	Position	X, Y, Z
	Torque	Magnitude, X, Y, Z
	Total force	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Custom property	Depends on number of elements

Element	Attribute	Components
Particle	Angular velocity	Magnitude, X, Y, Z
	Compressive force	n/a
	Distance	Define reference object*
	Kinetic energy	n/a
	Mass	n/a
	Position	X, Y, Z
	Potential energy	n/a
	Rotational kinetic energy	n/a
	Torque	Magnitude, X, Y, Z
	Total energy	n/a
	Total force	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Volume	n/a
	Custom property	Depends on number of elements
Bond	Normal force	Magnitude, X, Y, Z
	Normal moment	Magnitude, X, Y, Z
	Tangential force	Magnitude, X, Y, Z
	Tangential moment	Magnitude, X, Y, Z

* If the attribute is set to Distance you must define a point or plane from which the distance is measured. When distance is selected the **Define Reference Object** section of the pane will be activated. Choose Point or Plane and define its position and, for a plane, its distance from the origin.

The x and y axes are related. The attribute you select to measure on the x-axis will limit those available to measure on the y-axis. Similarly if the y-axis is configured first the attributes available on the x-axis will be limited.

Configuring the Y-Axis

Click the Y-axis tab then select the element attribute and component to be plotted on the y-axis. The attributes available in the list depend on the elements previously selected and the attribute to be measured on the x-axis. A component type can also be selected for certain attributes to determine which value will be graphed for a particular attribute component: For example, maximum, minimum or average particle velocity. Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Element	Attribute	Components
Contacts	Contact vector 1	Magnitude, X, Y, Z
	Contact vector 2	Magnitude, X, Y, Z
	Distance	Define reference object*
	Normal force	Magnitude, X, Y, Z
	Normal overlap	n/a
	Position	X, Y, Z
	Tangential force	Magnitude, X, Y, Z
	Tangential overlap	n/a

	Custom property	Depends on number of elements
Collisions	Average normal force	Magnitude, X, Y, Z
	Average tangential force	Magnitude, X, Y, Z
	Distance	n/a
	Duration	n/a
	Maximum normal force	Magnitude, X, Y, Z
	Maximum tangential force	Magnitude, X, Y, Z
	Normal energy loss	n/a
	Position	X, Y, Z
	Relative velocity	Magnitude, X, Y, Z
	Relative velocity normal	Magnitude, X, Y, Z
	Relative velocity tangential	Magnitude, X, Y, Z
	Tangential energy loss	n/a
	Total energy loss	n/a
	Velocity element A**	Magnitude, X, Y, Z
Velocity element B	Magnitude, X, Y, Z	
Geometry	Compressive force	n/a
	Distance	n/a
	Position	X, Y, Z
	Torque	Magnitude, X, Y, Z
	Total force	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Custom property	Depends on number of elements

Element	Attribute	Components
Particle	Angular velocity	Magnitude, X, Y, Z
	Compressive force	n/a
	Distance	Define reference object*
	Kinetic energy	n/a
	Mass	n/a
	Position	X, Y, Z
	Potential energy	n/a
	Rotational kinetic energy	n/a
	Torque	Magnitude, X, Y, Z
	Total energy	n/a
	Total force	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Volume	n/a
	Custom property	Depends on number of elements
Bond	Normal force	Magnitude, X, Y, Z
	Normal moment	Magnitude, X, Y, Z

	Tangential force	Magnitude, X, Y, Z
	Tangential moment	Magnitude, X, Y, Z

* If the attribute is set to Distance you must define a point or plane from which the distance is measured. When distance is selected the **Define Reference Object** section of the pane will be activated. Choose Point or Plane and define its position and, for a plane, its distance from the origin.

** When the axis is measuring the number of particles or number of contacts it is possible to graph either the total number of particles/contacts or the total number in a defined range: For example, counting only those particles with a mass between 0.1kg and 0.2kg or velocity between 2m/s and 5m/s. When the Total in Range option is selected the **Secondary Attribute** section of the pane is activated. Choose the attribute and component you wish to restrict the range by and enter the maximum and minimum values of that range.

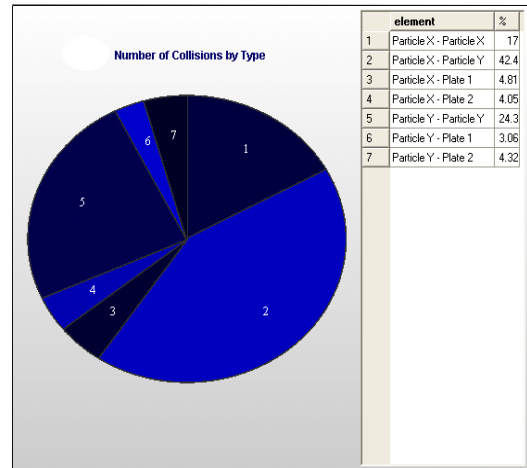
Settings

See *Settings* on page 99.

Creating a Pie Chart

A pie chart is a circular diagram divided into segments, each representing a category or subset of data (part of the whole). The amount for each category is proportional to the area of the sector. The total area of the circle is 100% and it represents the total population that is being shown.

For example, the figure below is a pie chart which shows the number of collisions, broken down by collision type. The largest slice is for particle X - particle Y collisions, which account for over 40% of the total.

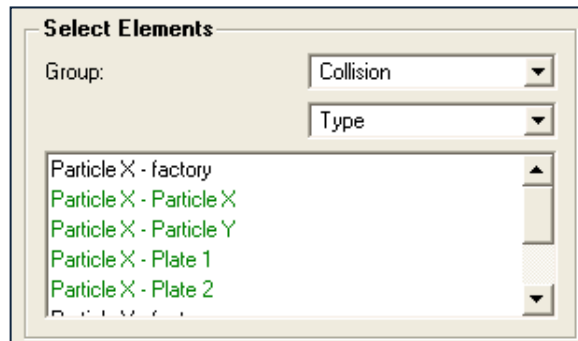


Selecting the Elements to Graph

The Select Element section is used to determine which elements in your model to collect data from.

Choose the group of elements you are interested in. For each element type, you can choose either to split the data by type or selection/bin group and these will form the segments (categories) of the pie chart.

When Type or Selection Group is selected, the categories within that selection are listed: For example if Contacts > Type are selected, all specific contacts are the categories listed: For example, particle x - particle x, particle x - particle y and particle y - particle y. Similarly if Contacts > Selection groups is selected, all selection and bin groups in your model are the categories listed. All categories within a



type or group are included in the chart by default. A category can be removed by double clicking on it and added back by double clicking on it a second time. Categories colored **green** are currently included in the pie chart, those colored **black** are not.

When the Selection group option is chosen, all selection and bin groups in your model are listed. Both the whole group and each individual bin are listed: For example 'bin group' (the group name) and 'bin group0', 'bin group1', 'bin group2' (the individual bins). When creating a pie chart, exclude the group name to compare the contents of individual bins (the bin group encompasses all of the individual bins and therefore accounts for 100% of the pie chart). For example, the figure below graphs the number of particles in a model. It is clear that all particles are contained within the bin group. However, if you want to compare how many are in each bin, exclude the whole bin group and just examine the individual bins.

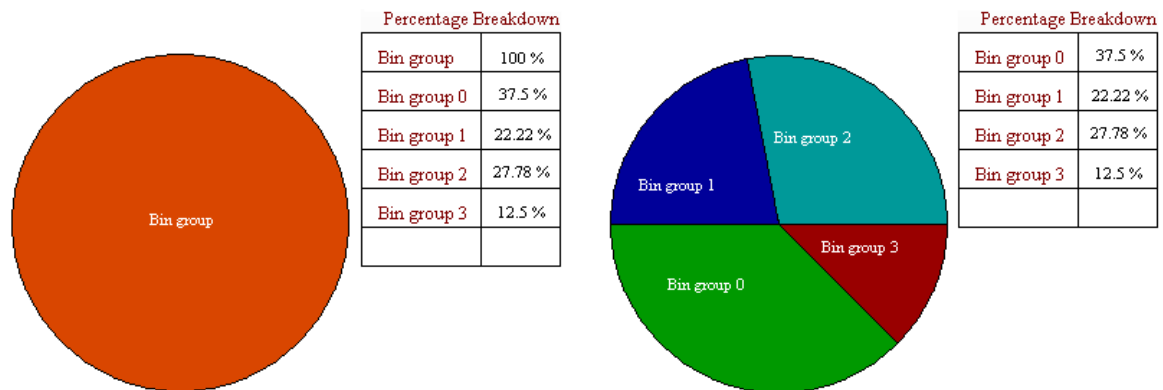


Figure 4-32: Alternative pie charts

Define Attribute

Click on the Select Attribute tab and select the element attribute and component you want to examine using the pie chart. The attributes available in the list will depend on the elements previously selected. The component type available for most attributes is Total as this is the only type that can be displayed using a pie chart: For example, the total mass of particles of Type A compared to particles of Type B. Refer to *Appendix B: Attribute Definitions* for more information on attributes.

Element	Attribute	Components
Contacts	Contact vector 1, 2	Magnitude, X, Y, Z
	Normal force	Magnitude, X, Y, Z
	Normal overlap	n/a
	Number of contacts**	n/a
	Tangential force	Magnitude, X, Y, Z
	Tangential overlap	n/a
	Custom property	Depends on number of elements
Collisions	Average normal force	Magnitude, X, Y, Z
	Average tangential force	Magnitude, X, Y, Z
	Maximum normal force	Magnitude, X, Y, Z
	Maximum tangential force	Magnitude, X, Y, Z

Element	Attribute	Components
	Normal energy loss	n/a
	Number of collisions**	n/a
	Relative velocity	Magnitude, X, Y, Z
	Relative velocity normal	Magnitude, X, Y, Z
	Relative velocity tangential	Magnitude, X, Y, Z
	Tangential energy loss	n/a
	Total energy loss	n/a
	Velocity of element A	Magnitude, X, Y, Z
	Velocity of element B	Magnitude, X, Y, Z

Element	Attribute	Components
Geometry	Compressive force	n/a
	Distance	n/a
	Torque	Magnitude, X, Y, Z
	Total force	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Custom property	Depends on number of elements
Particle	Angular velocity	Magnitude, X, Y, Z
	Compressive force	n/a
	Distance	Define reference object*
	Kinetic energy	n/a
	Mass	n/a
	Number of particles**	n/a
	Potential energy	n/a
	Rotational kinetic energy	n/a
	Torque	Magnitude, X, Y, Z
	Total energy	n/a
	Total force	Magnitude, X, Y, Z
	Velocity	Magnitude, X, Y, Z
	Volume	n/a
	Custom property	Depends on number of elements
Bond	Normal force	Magnitude, X, Y, Z
	Normal moment	Magnitude, X, Y, Z
	Number of broken bonds	n/a
	Number of intact bonds	n/a
	Tangential force	Magnitude, X, Y, Z
	Tangential moment	Magnitude, X, Y, Z

* If the attribute is set to Distance you must define a point or plane from which the distance is measured. When distance is selected the **Define Reference Object** section of the pane will be activated. Choose the point or plane from which the distance should be measured. A point is defined by its xyz position and a plane by its orientation and distance from the origin.

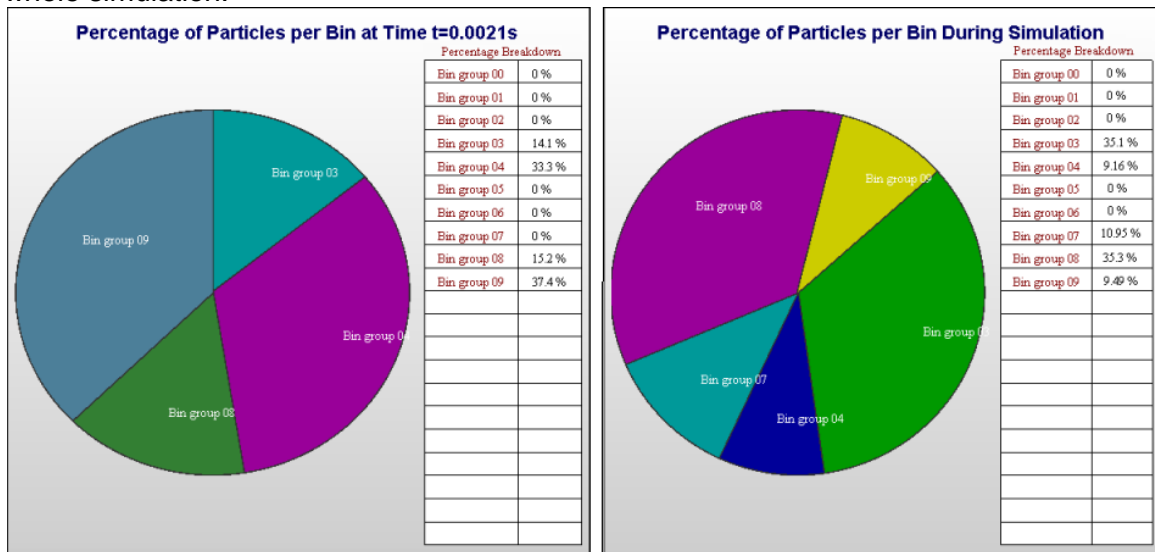
Settings

Two options are set in the Setting tab.

Time Step

Pie charts can be created for elements in a particular time step or over a range of time steps. By default the Current Time step option is selected. The current time step can be changed using the Current Time control in the Viewer control pane at the right of the screen.

For example, you could chart the total number of particles in each bin in a bin group at a particular time or the total number that have been in the bins over the course of the entire simulation. Comparing the charts we can see that at $t=0.0021s$ most particles are in bins 09 and 04, however this is not indicative of their location over the course of the whole simulation.



Unselect the Current Time Step to set a different start and end time. Grayed-out timesteps indicate partial saves and may not contain all the data you want to plot.

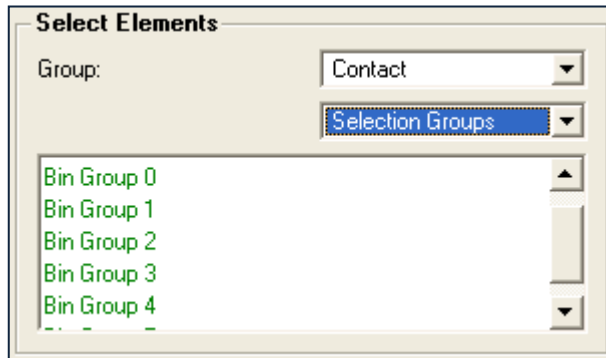
Graph Text

Set the pie chart title.

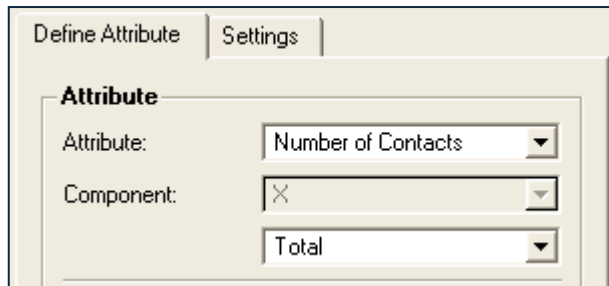
Example Pie Chart of Bin Groups

A model contains a bin group dividing the model domain into 10 bins. You want to show the number of contacts that are occurring in each bin at a given time.

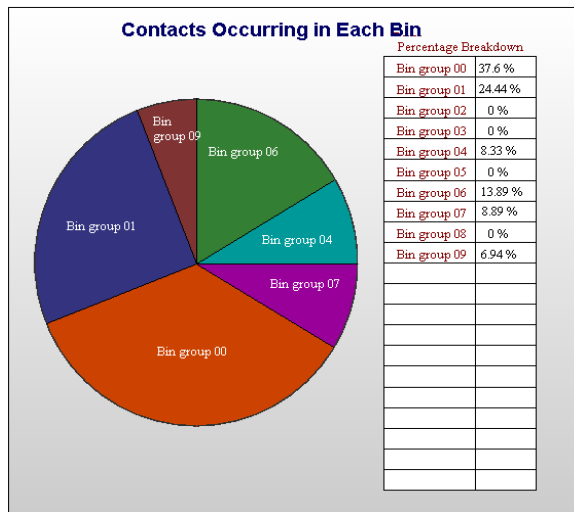
1. Select the element Contact > Selection Groups.



2. Choose the attribute Number of Contacts (Total).



3. Go to the Settings tab and give the chart a title.
4. Go to the Viewer Controls pane and choose the time step at which you want to examine the contacts. Ensure the Current time step option in the Settings tab is selected.
5. Click the Create Graph button.
6. The pie chart and the percentage breakdown of the number of contacts occurring in each bin will be displayed in the Viewer. For example:



Running EDEM in Batch Mode

EDEM's Command Line Interface (CLI)

With EDEM's command line interface, you can run EDEM in batch mode without using the GUI. You may want to do this when:

- You have a cluster with limited display capability and want to do grid processing
- You need to create batch jobs to run EDEM simulations
- You want to develop advanced scripting for parametric optimization.

The Command Line can also export data to a .csv file once simulation completes.

Using the Command Line

The EDEM Command Line has the same flags for both Windows and Linux:

```
edem.exe -console -i "input.dem" [processing flags][post-processing flags][-h]
```

For full details, use the `-h` flag to display the EDEM command line help.

Processing Flags

The processing flags set the simulator options such as simulation time, time step, and number of processors. You can also specify the license server to use from the command line. If you do not specify any processing flags, default values are used.

Post-Processing Flags

Use the post-processing flags to specify data export parameters. You can either specify the type of data to export (all, particle, geometry, etc.) or specify your simulation's configuration file.

If you want to export data from the command line, first select File > Export > Results Data from the Analyst to set the path and filename to export to.

Command Line Processing Output

While processing, the EDEM Command Line outputs a row of numbers at regular intervals. From left to right, the columns indicate:

- Number of time steps
- Simulation time (s)
- Processing time (s)
- Expected time to process one second of simulation time (s)
- Total linear kinetic energy (J)
- Total rotational kinetic energy (J)

Examples

Example 1: default processing options

1. Create and save a simulation deck using the EDEM Creator.
2. Open a console (cmd) window then change directory to the EDEM bin directory.
3. Type the following:

```
edem.exe -console -i "c:\My Documents\EDEM\mysim.dem" -rewind
```

The deck is rewound to the start then each time step is processed. A message is displayed once complete. You can then open the deck and use the EDEM Analyst to perform any post-processing.

Example 2: specifying processing options

1. Create and save a simulation deck using the EDEM Creator.
2. Open a console (cmd) window then change directory to the EDEM bin directory.
3. Type the following:

```
edem.exe -console -i "c:\My Documents\EDEM\mysim.dem" -p 4 -r 10
```

This command runs the simulation using 4 processors for 10 seconds total run time.

Example 3: run simulation and export data

1. Create and save a simulation deck using the EDEM Creator.
2. Open a console (cmd) window then change directory to the EDEM bin directory.
3. Type the following:

```
edem.exe -console -i "c:\My Documents\EDEM\mysim.dem" -rewind -ePC "c:\My Documents\EDEM\mysimdata.csv"
```

This command rewinds then runs the simulation using default processing flags. It then exports particle and contact data to a .csv file.

Example 4: run simulation and export data based on deck configuration

1. Create and save a simulation deck using the EDEM Creator.
2. Switch to the Analyst then select File > Export > Results Data.
3. Define the export queries then set the filename. Click Close.
4. Switch back to the Creator then save your deck. The configuration file is updated with details of the export queries.
5. Open a console (cmd) window then change directory to the EDEM bin directory.
6. Type the following:

```
edem.exe -console -i "c:\My Documents\EDEM\mysim.dem" -rewind -e "c:\My Documents\EDEM\mysim.dfg"
```

This command rewinds then runs the simulation using default processing flags. It then exports data based on the queries defined in the deck's configuration file.

Custom Properties

With custom properties you can dynamically define custom attributes to use in your simulation. Custom properties can be graphed and exported just like any other attribute. When loaded, contact models, particle body forces, and factories can all use and share data about these new custom properties.

Custom Property Category	Example
Contact	Contact duration, highest force during contact
Geometry	Wear, shot peening threshold, accumulation
Particle	Residence time, temperature
Simulation	Stickiness of a surface

Note: You must have an EDEM API license to use custom properties.

Creating a Custom Property

You can define any number of custom properties by developing plugins that use the EDEM API. Refer to the *EDEM Programming Guide* for details.

Viewing Custom Properties

Once a suitable plugin contact model, particle body force or factory is added to your simulation, select Tools > Custom Properties to view and edit details for each type of custom property.

Attribute	Description
Name	The unique name of the custom property.
State	Before simulation begins, custom properties are in a tentative state. Once simulation starts the properties become finalized.
Storage Type	The properties data type (currently always a double precision floating point number).
Number of Elements	Indicates the number of elements. Usually a property has one element. A property with a position in 3D space would have three elements (X, Y, and Z).
Units	The unit type, for example length, temperature, velocity etc. If the property has no unit, this is set to None.
Values	Indicates the property's initial values. Double-click to change the value (provided the custom property is either still tentative or is a simulation property). For custom simulation properties, the Values area also indicates the value for the current timestep.

When simulation starts, tentative properties are finalized. Once a property is finalized, you cannot modify or delete it (except for current values of simulation properties).

Appendix A: Estimating Simulation Time

Exact estimation of the real time required for a DEM simulation is almost impossible, as each simulation and each computer is different. However, see below for some general advice on estimating simulation time.

Time Step

One of the key numbers in DEM simulation is the Rayleigh time step. This is the time taken for a shear wave to propagate through a solid particle. It is therefore a theoretical maximum time step for a DEM simulation of a quasi-static particulate collection in which the coordination number (total number of contacts per particle) for each particle remains above 1. It is given by:

$$T_R = \frac{R}{c}$$

where R is the particle's radius, ρ its density, G the shear modulus and ν the Poisson's ratio. In practice some fraction of this maximum value is used, and for high coordination numbers (4 and above) a typical time step of $0.2T_R$ (20%) has been shown to be appropriate. For lower coordination numbers, $0.4T_R$ (40%) is more suitable.

Hertzian Contact

Whilst the Rayleigh time step is a suitable starting point for quasi-static simulations, for systems undergoing flow it may be that a shorter time step is required. Consider two elements approaching each other at a speed v . In one time step, t , the maximum possible overlap is:

In DEM, particles undergoing elastic (Hertzian) contact are treated as overlapping and this overlap is equated to a surface compression. t in the above equation must be such that the maximum overlap is lower than the theoretical maximum overlap for Hertzian contact. In practice, to get a good numerical integral to the contact graph at least six time points should occur (though then is more desirable) - three during approach and three during separation.

Elastic Impact

From Hertz theory of elastic collision the total time of contact is given by:

$$t_c = \frac{2.94}{\sqrt{g}}$$

Where:

$$g = \frac{2v}{t_c}$$

Where m_1 and m_2 are the mass of the two elements.

$$\frac{m_1}{m_2} = \frac{V_1}{V_2}$$

Where r_1 and r_2 are the radii of the two elements.

$$\frac{r_1}{r_2} = \frac{V_1}{V_2}$$

where ν_1 and ν_2 are the poisons ratio of the two materials.

Where V_{rel} is the relative velocity and V_1 and V_2 are the velocity of the two elements.

Appendix B: Attribute Definitions

Particle Attributes

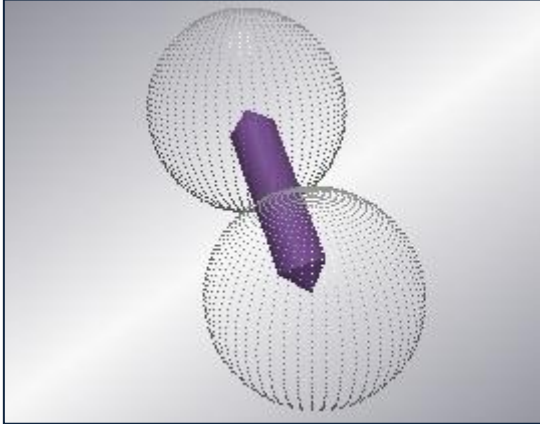
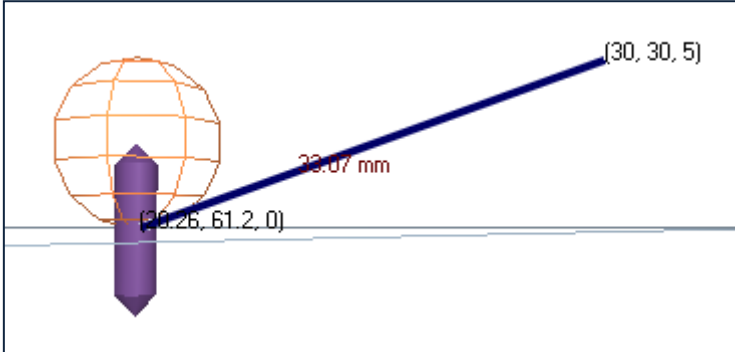
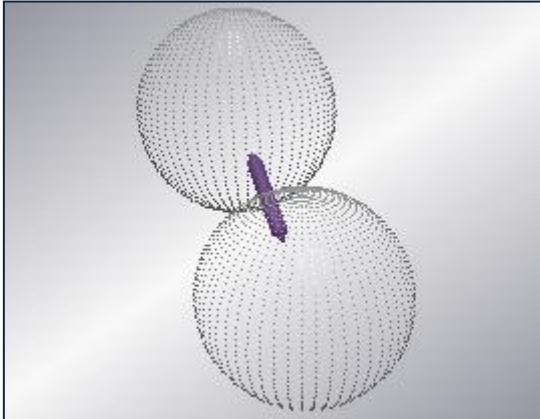
Attribute	Description
Angular velocity	Angular velocity is defined as the speed at which an object rotates. For a particle of radius r (m) rotating at velocity ω rad/s, $V = r \omega$ Angular velocity magnitude = $\sqrt{w_x^2 + w_y^2 + w_z^2}$
Charge	Charge is an electrical property of particles (such as electrons and protons) which causes them to attract and repel each other. A material with an excess of electrons has a negative charge; material with an absence of electrons (or an excess of protons) has a positive charge. Materials with a balanced number of electrons and protons are called 'neutral'. Positive and negative charges attract each other.
Compressive force	The compressive force is the sum of the surface normal force magnitudes.
Distance	Distance is defined as how far apart the center of a particle is from a reference point or plane.
Electrostatic force	Electrostatic force is a force of interaction between charged particles, measured in Newtons. In EDEM the force is calculated using the screened Coloumb force equation.
ID	A geometry element, particle surface, particle and collision all have unique ID numbers.
Moment of Inertia	Inertia is a measure of a particles resistance to a change of momentum. For a sphere $I = (2/5).m.r^2$
Kinetic energy	The kinetic energy is defined by $Ke = 0.5 * m * v^2$ where v is the velocity magnitude of the particle.
Mass	The mass of a particle is defined as the magnitude of the gravitational force on a particle. $Mass = volume * density$.
Number of particles	The number of particles is the total number of particles in a simulation (or bin group) at that point in time. In a bin group the option of "total over time" can be used. This will produce a cumulative particle count for any particles that pass through that bin group. Please note that the total over time option can only be used for bin groups with 1 bin.
Orientation	Orientation is the alignment of a particle in relation to Cartesian space. It is represented in EDEM as a 3-by-3 matrix that defines the rotation of the particle from its original orientation (as defined in the Creator and assigned an identity matrix).
Position	The X, Y, or Z position of the center of the particle.
Potential energy	The potential energy is defined by $Pe = m.g.h$ where m is the particle mass, g is gravity and h is the distance between the center of mass of the particle and the defined "ground" level. If gravity is acting in $-ve Z$ the ground level is $z=0$.

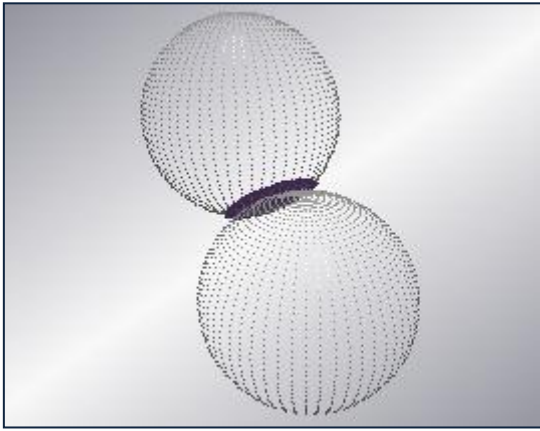
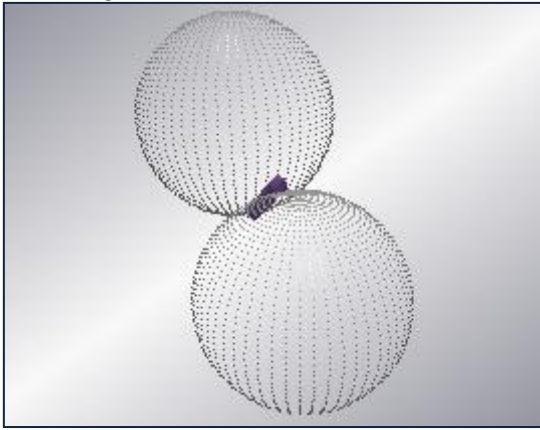
Attribute	Description
Rotational kinetic energy	The rotational kinetic energy is defined as $0.5 * \text{Inertia} * (\text{angular velocity magnitude})^2$.
Time	In EDEM time is the real time it takes for an event to occur.
Torque	The torque on the particles is defined as the tangential contact force * particle radius.
Total energy	The total energy of a particle is the sum of the kinetic energy, rotational kinetic energy and potential energy of a particle.
Total force	The total force is the resultant unbalanced forces on a particle or geometry element in x y and z.
Velocity	The velocity of the particles is defined as the rate of change of displacement (in X Y and Z).
Voidage	<p>Voidage is the percentage of space not taken up by particles.</p> <p>Voidage (%) = $100 * (\text{Bin Volume} - \text{Total particle volume in the bin}) / (\text{bin volume})$</p> <p>Voidage can only be exported from a bin group. Note that the total particle volume in the bin is calculated from the bin that contains the center of the particle. A particle which overlaps two or more bins will only be counted in the bin that contains the particle center of mass.</p>
Volume	The volume of a particle is defined as the amount of space that it takes up.

Geometry Attributes

Attribute	Description
Charge	See <i>Particle Attributes</i> .
Compressive force	See <i>Particle Attributes</i> .
Distance	Distance is defined as how far apart the center of a geometry node is from a reference point or plane.
Electrostatic Force	See <i>Particle Attributes</i> .
ID	A geometry element, particle surface, particle and collision all have unique ID numbers.
Node 1,2,3	The position (x, y and z) of the geometry nodes.
Number of geometry elements	Geometry in EDEM is split into triangular elements. The number of geometry elements is a summation of this.
Position	The X, Y, or Z position of the center of the geometry element.
Torque	<p>The torque on the geometry is defined as the normal contact force * distance where the distance is the distance from the center of mass to the contact point.</p> <p>The torque is calculated from the individual torque and moment about each individual geometry element. This is then converted to the torque on the whole geometry about the center of mass. The torque can be plotted/exported along the X, Y and Z axis.</p>
Total force	The total force is the resultant unbalanced forces on a particle or geometry element in x y and z.
Velocity	The velocity of the geometry is defined as the rate of change of displacement of the individual geometry nodes.

Contact Attributes

Attribute	Description
Contact vector 1,2	<p>When two particles collide the contact vector is the vector connecting the contact point to the center of the particle.</p> 
Distance	<p>The distance from the point of contact to a user-defined point or plane.</p> 
Normal force	<p>The normal force arises when two bodies are in direct contact with one another. It always acts perpendicular to the body that applies the force.</p> 

Attribute	Description
Normal overlap	<p>The normal overlap represents the normal deformation of a particle. The normal overlap N between two particles i and j at positions p_i and p_j with radii r_i and r_j is defined as:</p> 
Number of contacts	<p>Contacts are the impacts occurring between elements at data write-out points. In other words, the contact is in progress when the write-out takes place. The contact has an associated force, position and so on - these are discrete values. If two elements stay in contact with each other for some time e.g. over four write-out points, four contacts will be stored and each of these may have a different force, position and so on.</p>
Position	<p>The central point between two elements at the point in time when a contact starts.</p>
Tangential force	<p>The tangential force is the force from the tangential overlap.</p> 
Tangential overlap	<p>The tangential displacement of the contact point up to the point at which the contact ends or the particle begins to roll or slip. The tangential overlap represents the tangential deformation of a particle.</p>
Time	<p>In EDEM time is defined as the real time it takes for an event to occur.</p>

Collision Attributes

Attribute	Description
Average normal force	The average normal force over the duration of a single collision.
Average tangential force	The average tangential force over the duration of a single collision.
Distance	Defined as the distance from the collision position to a user defined point or plane. Distance from the collision position to the point (10,0,-12.05) mm is 10 mm. <div data-bbox="623 623 1195 951" data-label="Image"> <p>The diagram illustrates two overlapping spheres represented by a yellow grid. A vertical blue line segment connects the point (0,0,-12.05) at the top to the point (0,0,-22.05) at the bottom. A red double-headed arrow next to this line is labeled '10 mm', indicating the distance between these two points.</p> </div>
Duration	Collision duration is the total time that the two elements involved in a collision are in contact.
End time	The end time is the time at which the collision ends.
ID	Each collision has a unique number which can be used to track the unique collision and the elements involved.
ID Element n	Each element in a collision has a unique number.
Maximum normal force	The maximum normal force experienced during a single collision.
Maximum tangential force	The maximum tangential force experienced during a collision.
Normal energy loss	The energy lost during a collision due to the normal overlap.
Number of collisions	The number of completed collisions. Only completed collisions are counted: collisions in progress (contacts) are not counted.
Position	The central point between two elements at the point in time when a collision starts. The collision position is written out on completion of the collision.
Relative velocity	Relative velocity of two elements in a collision is $V_a - V_b$
Relative velocity normal	Relative normal velocity of two elements in a collision is $V_{n_a} - V_{n_b}$
Relative velocity tangential	Relative tangential velocity of two elements in a collision is $V_{t_a} - V_{t_b}$
Start time	Defined as the point in time when the first contact of a collision occurs.

Attribute	Description
Tangential energy loss	Energy lost during a collision due to the tangential overlap.
Time	In EDEM, time is defined as the real time it takes for an event to occur.
Total energy loss	The sum of the normal and tangential energy loss.
Velocity of element A/B	The velocity magnitude of an element involved in a collision, element A will have the lowest ID number and element B will have the highest.

Bonding Attributes

Attribute	Description
End Time	The point at which a bond breaks.
Normal Force	The resultant normal force experienced by the bond.
Normal Moment	The normal moment applied to the center of the cylindrical bond.
Number of Broken Bonds	The total number of broken bonds in the simulation.
Number of Intact Bonds	The total number of intact bonds in the simulation.
Start Time	The point in time at which the bond was created.
Tangential Force	The resultant tangential force experienced by the bond.
Tangential Moment	The tangential moment applied to the center of the cylindrical bond.

Appendix C: EDEM File Types

EDEM Files

File	Type	Description
.dem	Data file	Simulation data. One of three files created for every model. It is also created when you export a particle.
.dfg	Config file	Configuration information for the simulation. This is one of three files created for every model. It is also created when you export a particle. EDEM 2.4 automatically converts the .dem.cfg config file format to the .dfg format.
.dem.scn.n	Data file	Additional EDEM simulation data. A .dem.scn.n file (where n is a number) is created whenever the .dem file reaches approx. 1.5 GB in size.
.dll	Library	EDEM uses dll library files for contact models, particle body forces, and user-defined factories. Use Tools > Options > File Locations to change dll.
.ess	Data file	EDEM simulation settings. Binary file that stores per-simulation data for selections, custom properties, and field information (if applicable).
.idx	Index file	Index for the .dem file. Also created when you export a particle.
materialsDB.ddb	Materials database	The materials database is used to define any material used in a model. Use Tools > Options > File Locations to change its location.
.ptf	Particle template	The particle template file contains mesh information. It is created whenever you save a model containing a particle template.

EDEM Standard Supported Filetypes

File Extension	Type	Source Software Application
.case	Data	EnSight
.csv	Data	Any spreadsheet or other software application that can read text files
.igs, .iges	Geometry	IGES
.msh	Geometry ¹	FLUENT Mesh
.stl	Geometry	STL
.stp, .step	Geometry	STEP

¹ Enabled with EDEM-CFD Coupling for FLUENT

EDEM Optional Supported Filetypes

File	Type	Source Software Application
.cgns	Data ¹	CFD software applications that can read the generic CFD General Notation System format. For example, FLUENT, CFX, and STAR-CD
.CATPart, .CATProduct, .model	Geometry	Catia v4/v5
.dxf, .dwg	Geometry	AutoCAD
.prt, .asm, .g	Geometry ²	Pro/E
.sat	Geometry	ACIS
.vda	Geometry	Various CAD software applications
.x_t, .xmt_txt, .x_b	Geometry	Parasolid

¹ Enabled with the Field Data Coupling Module

² Pro/E import is Windows-only

References

General DEM

Cundall P. A. and Strack O. D. L., *A discrete numerical model for granular assemblies*, Geotechnique, 29, 517, (1979)

Werner Goldsmith, *Impact: The Theory and Physical Behavior of Colliding Solids*, (1960)

Rayleigh Timestep

Ning, Z; (1995). *Elasto-plastic impact of fine particles and fragmentation of small agglomerates*, Ph.D Thesis, Civil Eng; Aston University Birmingham

Ning, Z; Boerefijn, R and M. Ghadiri, *Effect of particle size and bond strength in impact breakage of weak agglomerates*, (1997)

Ning, Z et al; *Discrete element simulation of impact breakage of lactose agglomerates*, Advanced Powder Technology, Vol.1:15-37. (1997)

Damping Theory

A.O. Raji, *Discrete Element Modelling of the Deformation of Bulk Agricultural Particulates*, Department of Agricultural and Environmental Sciences at the University of Newcastle upon Tyne (1999)

Tsuji et al (1992) *Lagrangian numerical simulation of plug flow of cohesionless particles in a horizontal pipe*, Powder technology, 71:239-250

Zhang and Whitten, *The calculation of contact forces between particle using spring and damping models*, Powder Technology, 88:59-64 (1996)

Contact Models

Hertz-Mindlin (no slip) and Linear Spring

Y. Tsuji, T. Tanaka and T. Ishida, *Lagrangian numerical simulation of plug flow of cohesionless particles in a horizontal pipe*, Powder Technology, 71 239-250, (1992)

Alberto Di Renzo, Francesco Paolo Di Maio, *Comparison of contact-force models for the simulation of collisions in DEM-based granular flow codes*, Chemical Engineering Science, 59 525 – 541, (2004)

Hertz-Mindlin (no slip) with Rvd Rolling Friction

Y. C. Zhou, B. D. Wright, R. Y. Yang, B. H. Xu, A. B. Yu, *Rolling friction in the dynamic simulation of sandpile formation*, *Physica A* 269 (1999) 536–553.

Jun Ai, Jian-Fei Chen, J. Michael Rotter, Jin Y. Ooi, *Assessment of rolling resistance models in discrete element simulations*, *Powder Technology* 206 (2011) 269–282

Hertz-Mindlin with JKR Cohesion

K. L. Johnson, K. Kendal, and A. D. Roberts, Surface energy and the contact of elastic solids, *Proc. R. Soc. Lond. A.*, 324, 301-313 (1971)

Linear Cohesion

Namiko Mitarai, Franco Nori, *Wet granular materials*, *Advances in Physics*, 55, 1-45 (2006)

Hertz-Mindlin with Bonding

Potyondy D. O. and Cundall P. A, *A bonded-particle model for rock*, *International Journal of Rock Mechanics and Mining Sciences*, 41, 1329-1364 (2004)

Hertz-Mindlin with Heat Conduction

B. Chaudhuri et al, *Modeling of heat transfer in granular flow in rotating vessels*, *Chemical Engineering Science* 61 6348 – 6360 (2006)

Hysteretic Spring

Walton, O. R. and Braun, R. L., “Viscosity and Temperature Calculations for Assemblies of Inelastic Frictional Disks”, *J. Rheology*, 30(5), 949-980 (1986)

Walton, O. R. and Braun, R. L., “Stress Calculations for Assemblies of Inelastic Spheres in Uniform Shear”, *Acta Mechanica*, 63, 73-86 (1986)

Particle Body Forces**Temperature Update**

B. Chaudhuri et al, *Modeling of heat transfer in granular flow in rotating vessels*, *Chemical Engineering Science* 61 6348 – 6360 (2006)

Technical Support

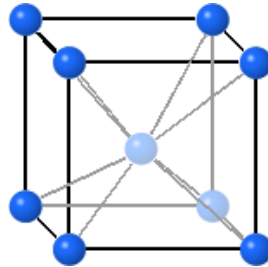
DEM Solutions is committed to providing you with quality software. If at any time you experience problems with EDEM, please contact our support group:

E-mail: support@dem-solutions.com

Glossary

B

BCC Lattice: In a body-centered cubic (BCC) lattice there is one particle at each corner of a cubic unit cell and one particle in the cell center. The packing efficiency of a BCC lattice is considerably higher than that of a simple cubic lattice. See also *Cubic Lattice*.



C

CAD: Computer-aided design.

CAE: Computer-aided engineering.

CFD: Computational fluid dynamics.

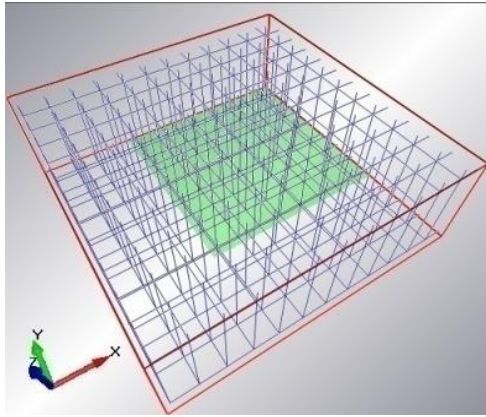
Coefficient of Restitution: The coefficient of restitution is the ratio of speed of separation to speed of approach in a collision.

Coefficient of Rolling Friction: When a body rolls on a surface, both the body and the surface can deform to oppose this rolling motion. This is usually represented as a torque being applied to each of the bodies. The coefficient of rolling friction is a scalar value used to determine how much torque is needed to be applied to an object of a given material at rest on a flat surface to put it into motion.

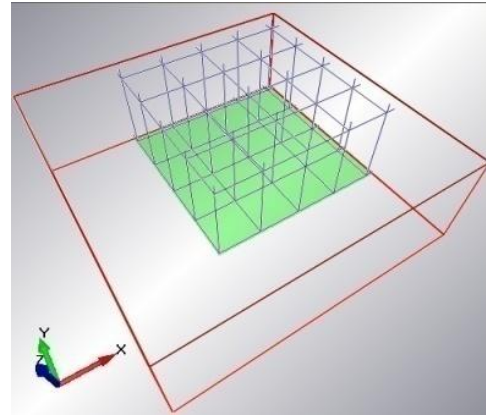
Coefficient of Static Friction: The coefficient of static friction is a scalar value used to determine how much force is required before an inert object, of a given material, at rest on another known substance, can be put into motion.

Coordination Number: The total number of contacts in a simulation.

Cubic Lattice: The cubic lattice is defined by setting the start point, separation and rotation. The start point is the point in the domain where the lattice will be drawn from. The separation is the distance between each corner node in the lattice. This can be defined for each dimension. The rotation is the angle at which the entire lattice is tilted. The simulator will attempt to place a particle at every node, side center and cell center on or in the chosen section. The center of each cell and side is not marked when the lattice is displayed in the Viewer. For example, the figures below are the same model each with a different lattice.



The lattice has been given a start point to just within the domain boundaries. The separation distance in each direction is quite small leading to a dense lattice.



The lattice has been given a start point to coincide with the geometry. The separation distance in each direction is larger and will result in less particles being placed on the surface.

D

DEM: (1) Discrete element method; (2) Discrete element modeling; (3) Distinct element modeling.

Density: The density of a body is the weight per unit volume.

Dynamic Section: Sections can be static or dynamic. Dynamic sections move during the course of a simulation.

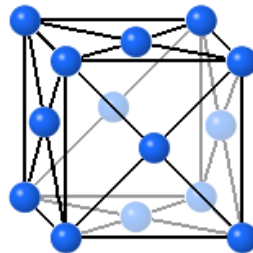
Dynamic Factory: A dynamic factory produces particles over the course of a simulation. You can define the total number of particles to be created, the creation rate and the creation start time.

F

Facet Length: The facet length is the maximum length of a facet edge.

Facet Sag: The facet sag is the distance between the curved body surface and the facet.

FCC Lattice: In a face-centered cubic (FCC) lattice there is one particle at each corner of a cubic unit cell and one particle in the center of each face. The packing efficiency of a FCC lattice is considerably higher than that of both a simple cubic lattice and a body centered cubic lattice. See also *Cubic Lattice*.



FEA: Finite element analysis.

Fixed Time-step: A fixed time step remains constant through the simulation.

H

Hertz-Mindlin (no slip) with RVD Rolling Friction: A model that provides relative velocity-dependent rolling friction and ensures the same torque being applied to each of the particles in contact.

Hertz-Mindlin with JKR Cohesion: A cohesion contact model that allows users to edit the cohesive quality of sticky materials, especially fine dry powders with particles sized 100 microns or less. The model can also be successful in modeling wet grains.

Hysteretic Model: A model that allows plastic deformation behaviors to be included in the contact mechanics equations, resulting in elastic-plastic deformation behavior.

I

Simulation Deck: An EDEM simulation usually consists of four files: a *.dem, *.dfg, *.idx and *.ptf file. All simulation data is held within the .dem file. The .dfg and .idx files contain configuration and index information. If the .idx and .dfg files are not available, they will automatically be created by EDEM when the .dem file is loaded. The *.ptf file contains details of any particle templates imported into your model.

L

Log-normal Distribution: The log-normal distribution is the probability distribution of any random variable whose logarithm is normally distributed. If x is a random variable with a normal distribution, then $\exp(x)$ has a log-normal distribution.

N

Normal Distribution: Normal distribution is a theoretical frequency distribution with a finite mean and variance. A probability density function is used to represent the distribution in terms of integrals. It is usually represented by a bell-shaped curved, symmetrical about the mean. This is called the *normal curve*.

O

Orientation Matrix: Non-spherical particles have their own three-dimensional coordinate system. This is necessary to define the position and orientation of the spheres making up the particle surface. The orientation matrix describes the transformation of a co-ordinate to a new position. Refer to the online help for more detail.

P

Particle Factory: Used to define where, when and how particles appear in a simulation.

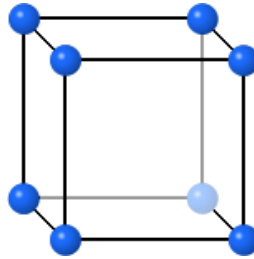
Physical Section: A physical section of geometry is an actual surface or volume that particles can interact with.

Poisson's Ratio: Poisson's ratio is the ratio of transverse contraction strain to longitudinal extension strain in the direction of stretching force.

S

Shear Modulus: Shear modulus (G) is defined as the ratio of shear stress (F) to the shear strain (A) i.e. $G = F / A$, where shear stress is the components of stress at a point that act parallel to the plane in which they lie, and shear strain is the components of a strain at a point that produce changes in shape of a body (distortion) without a volumetric change.

Simple Cubic Lattice: In a simple cubic lattice there is one particle at each corner of a cubic unit cell. See also *Cubic Lattice*.



Simulation Time: The simulation time is the amount of real time your simulation represents. The number of iterations required to complete a simulation of a defined time is displayed at the bottom of the Simulation Time section.

Static Factories: A static factory produces particles at a specified time. Simulation is paused whilst this takes place.

T

Time Step: The amount of time between iterations (calculations).

V

Virtual Section: A virtual section is a surface or volume that does not actually exist and does not interact with anything in a simulation. Virtual sections are usually used to create particle factories.

W

Work Function: The work function is the minimum energy (measured in electron volts) needed to remove an electron from a solid to a point immediately outside the solid surface.

Index

2Rmin.....	55	Coloring Bin Groups.....	79
3D.....	16, 47, 58	Coloring Selection Groups.....	79
ACIS v1.1.....	29	Columns.....	88
Active Sets.....	81	Command Line Interface.....	111
Alpha value.....	24	Commuter Licensing.....	14
Alternative Representation.....	62	Compressive Force.....	116
Analyst.....	7, 57, 60	Compressor.....	85
Angular Velocity.....	116	Cones.....	62
Animation.....	85	Configuration ID.....	88
Animation Controls.....	62	Console.....	111
Attribute Coloring.....	68	Constant Bitrate Encoding.....	85
Auto Update.....	47, 68	Contact.....	62, 68
AVI.....	85	Contact Model.....	21, 125
Axis Range.....	100, 101, 102	Contact Radius.....	29
Background Color.....	19, 61	Contact Vector.....	62, 119
Batch Mode.....	111	Coordination Number.....	128
BCC Lattice.....	128	Copy Geometry.....	40
Bin Elements.....	70	Creation Rate.....	40
Bin Group.....	70	Creator.....	7, 15
Bin Group Coloring.....	79	Critical Normal Stress.....	22
Binary.....	40	Cubic Lattice.....	40, 128
Bitmap.....	84	Custom Particle Properties.....	113
Bond Formation Time.....	23	Custom Properties.....	113
Bond State.....	67	Custom Properties.....	44
Bonded Disk Radius.....	22	Damping Theory.....	125
Bonding.....	67	Data.....	88, 94
CAD.....	128	Data Browser.....	7, 8, 9, 18, 48, 59
CAD Template Selection.....	73	Data Point.....	50, 55
CAE.....	128	Data Save Configuration.....	51
Calculate Properties.....	29	Delimiter.....	89
Camera Angles.....	16, 47, 58	DEM.....	6, 55, 127, 129
Camera Position.....	16	DEM Solutions.....	127
Catia.....	29	Density.....	129
Center of Mass.....	28	Display Bounding Box.....	39
CFD.....	128	Display Grid.....	29
Changing Units.....	12	Display Mode.....	78
Charge.....	116	Display Options.....	62, 68, 78
Check In License.....	14	Display Threshold.....	67
Check Out License.....	14	Distance.....	116, 121
Clipping Group.....	81	DivX.....	85
Closing Panes.....	8, 9	Docked Panes.....	7, 8, 9
Codec.....	85	Domain.....	31
Coefficient of Restitution.....	128	Domain Boundaries.....	81
Coefficient of Rolling Friction.....	128	Domain Grid.....	81
Coefficient of Static Friction.....	128	Dynamic Factory.....	40, 129
Collision Detection.....	53	Dynamic Section.....	129
Coloring.....	68	Electrostatic Force.....	116

Electrostatics.....	21	ID	116
Encoding Quality	85	IGES	29
End Time.....	122	Images	84
Energy.....	18, 48	Imported Geometry Bin Group	73
EnSight	88, 89	Importing Geometry.....	38
EnSight Video	85	Importing Materials.....	21, 28
Estimating Simulation Time	114	Importing Particle Factories.....	45
Export Custom Properties	61	Importing Particles	29
Export Simulation Deck	61	Inertia	116
Export Video.....	85	Interactions	28
Exporting a Particle	29	Iteration	50
Exporting Data	88	Keyframes.....	85
Facet Length	129	Kinetic Energy.....	18, 48, 116
Facet Sag.....	129	Lattice	16, 40
Factories	40	Legend.....	68
FCC Lattice	129	License Manager.....	14
FEA.....	129	License Server	13
File Locations	19, 61	Licensing.....	13, 21
Fixed Time Step	50, 55, 129	Line Graph	101
Fluent Mesh	29	Linear Cohesion	21, 24, 126
Frame Rate	85	Linear Distribution	40
Full Data Save.....	51	Linear Spring.....	21, 24
Full Save	51	Log-normal distribution.....	40, 130
Generation Rate.....	42	Manual Selections	70
Geometry Bin	72	Mass	29, 116
Geometry Transformations.....	40	Mass Flow Analysis.....	73
Globals.....	21	Mass Generation Rate	40
Graph.....	94, 95, 101, 102, 106	Materials	21, 28
Gravity.....	21	Materials Database	21, 28
Grid	29, 55	Materials Editor	21, 28
Grid Bin Group	71	Merge Geometry	38, 40
Grid Selection.....	71	Model Domain.....	31
Group Area	70	Modifier Keys	19
Group Query	70	Moment of Inertia	29
Groups	70, 81	Mouse Configuration	19, 61
GUI	7, 8, 9	Moving Plane	21, 26
Guide Lines.....	95, 101, 102	Multiple Images.....	84
H.264	85	Multiple Selection Queries.....	90
Headlight.....	17	Node	118
Heat Conduction	22, 126	Normal Distribution.....	40, 130
Hertz-Mindlin.....	21, 125, 126	Normal Force	62, 119, 122
Hertz-Mindlin (no slip)	22	Normal Moment.....	122
Hertz-Mindlin (no slip) with RVD Rolling Friction.....	22, 130	Normal Overlap.....	62, 119
Hertz-Mindlin with JKR Cohesion.....	23, 130	Normal Stiffness.....	22
Highlight Bin	70	Number of Broken Bonds	122
Highlight Sub-Selection	79	Number of Geometry Elements	118
Histogram.....	95	Number of Intact Bonds.....	122
Html	18, 48, 59	Number of Particles.....	116
Hysteretic Model	130	Number of Processors.....	54
Hysteretic Spring.....	25	On Screen Display	78
		Opacity.....	16, 47, 62

Orientation	116	Selective Save	51
Orientation Matrix.....	44, 130	Set Start Time	61
Orthographic Projection.....	16	Shear Modulus	131
Parameters	40	Shear Stiffness.....	22
Parasolid	29	Simple Cubic Lattice.....	131
Partial Save.....	51	Simulation Deck	130
Particle Body Force	21	Simulation Time	50, 55, 131
Particle Body Forces	26	Simulator	46, 55
Particle Detail.....	19, 61	Simulator Grid	53
Particle Display Options	20	Slices	81
Particle Export.....	29	Smooth Colors	69
Particle Factory	40, 45, 130	Solve Report	48
Particle Limits.....	31, 50	Spray	40
Particle Template	29	Start Time	121, 122
Particles	29, 40	Static Coloring.....	68
Periodic Boundaries	32	Static Factory	40, 131
Perspective Projection.....	16	STEP	29
Physical Section.....	130	Step Factor	58
Pie Chart.....	106	STL	29
Planes	81	Streams.....	62
Point of Action	37	Sub-Selection Coloring.....	79
Poisson's Ratio.....	131	Surfaces.....	29
Portable Network Graphic	84	Tabs Pane.....	7, 8, 9, 16, 46, 58
Position	116, 121	Tangential Energy Loss.....	121
Potential Energy.....	18, 48, 116	Tangential Force	62, 119, 122
Processing Time	50, 55	Tangential Moment	122
Processors	54	Tangential Overlap.....	119
ProE	29	Target Data Rate.....	85
Protractor	80	Target Position	17
Queries	70, 88	Technical Support	127
Radius.....	29	Template	29, 62
Range	95, 101, 102, 106	Time.....	116
Rayleigh Time Step.....	50, 55, 125	Time Settings	50
Record Animation.....	85	Time Step.....	50, 55, 94, 131
Reference Object	95, 101, 102, 106	Tools	80, 81
Relative Velocity.....	121	Torque.....	116
Relative Velocity Normal	121	Total Energy.....	116
Relative Velocity Tangential	121	Total Energy Loss	121
Removing Materials.....	28	Total Force.....	116
Representation.....	62	Tribocharging	21
Rotational Kinetic Energy	116	Truncate Results	61, 94
Ruler	80	Type Filter	78
Scatter Plot	102	Units.....	12
Screen Grab.....	84	Up Vector	17
Secondary Attribute....	95, 101, 102, 106	User Defined Particle Attributes	113
Selected IDs.....	70	User Defined Particle Factories.....	45
Selection Group	62	User Interface.....	7, 8, 9
Selection Group Coloring	79	Velocity	116
Selection Group Representation.....	78	Velocity of Element	121
Selection Query Editor	74	Video.....	85
Selections	70	Video Codec	85

Video Compressor.....	85	WMV	85
Viewer	7, 8, 9, 16, 47, 58	Work Function.....	27, 131
Viewer Controls.....	7, 16, 47, 58	X-axis.....	94, 95, 101, 102
Virtual Section.....	131	Xvid.....	85
Voidage.....	116	Y-axis.....	94, 95, 101, 102
Volume.....	29, 116	Zoom.....	8, 19
Windows Media Video.....	85		