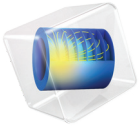

INTRODUCTION TO

Granular Flow Module

6.4

COMSOL

Introduction to the Granular Flow Module

Version: COMSOL 6.4

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at www.comsol.com/support/case. Useful links:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: www.comsol.com/support/learning-center

Part number: CM025402

Contents

The Granular Flow Module

The Granular Flow Module is used to model the bulk behavior or flow of granular materials. Granular flow simulations are used to model the behavior of macroscopic particles, such as sand, grains, powders, or pellets, when subjected to external forces like gravity, vibration, or shear. Unlike fluids or solids, granular materials can exhibit both solid-like and fluid-like behavior depending on their packing, confinement, and stress conditions. Granular flow is important in a variety of applications including pharmaceuticals, geotechnics, additive manufacturing, and food processing.

Most of the physics interfaces in COMSOL Multiphysics use the Finite Element Method (FEM) to compute fields such as temperature, fluid velocity, electric potential, concentration, or displacement. In contrast, granular flow provides a Lagrangian description of a problem, in which the particles are treated as discrete entities that can interact with external fields, with boundaries in the surrounding geometry, and with each other. Granular flow is based on the Discrete Element Method (DEM) which treats the particles (referred to as grains) as spheres (3D) or cylinders (2D) occupying space in the computational domain.

Grains are allowed to collide with each other and with the boundaries (walls). Contact force models are utilized to determine the resultant grain deformations and contact forces. Their trajectories are computed in the time domain by solving a set of equations based on the Newton's laws of motion. Both translational and rotational degrees of freedom are accounted for and are updated every time step based on the forces and torques resulting from contact and external fields.

Heat transfer effects on the grains can also be included by tracking the temperature of each grain. A grain's temperature can change due to an external heat source, convective heat transfer with the surroundings and conductive heat transfer due to grain-grain and grain-wall contacts.

The Granular Flow Module also offers a wide variety of release mechanisms to control how the grains are inserted into the geometry. Grains can be released directly into the domain or along the boundaries. By default, grains are released such that they do not overlap with other grains or walls. After the simulation is complete, many dedicated tools for visualizing grain motion and reporting grain statistics are available.

The Granular Flow Module supports simulations in two and three space dimensions.

The Granular Flow Module Physics Interface Guide

Each COMSOL Multiphysics physics interface (for example, the Granular Flow interface) expresses the relevant physical phenomena in the form of sets of partial or ordinary differential equations, together with appropriate boundary and initial conditions. Each feature added to the physics interface represents a term or condition in the underlying equation set. These features are usually associated with a geometric entity within the model, such as a domain, boundary, edge, point. The Granular Flow interface also includes features that are associated with the grains and global features that are not associated with any geometric entities in the model.

The Granular Flow Module offers the Granular Flow interface ( ), found under the Fluid Flow branch ( ) in the Model Wizard. This interface models the bulk behavior of granular materials by computing the individual grain trajectories under the influence of external forces and contact forces.

Figure 1 uses the application library example *Mixing Grains in a Ribbon Mixer* to show the Model Builder tree structure and the Settings window for the selected physics interface node. This node acts as the starting point for the Granular Flow simulations and has options to choose the contact force models, rolling resistance models and additional options to include the thermal effects. A granular flow model must contain at least one node each to specify the material and contact properties of the grains and walls. Additionally at least one feature that releases the grains must be included in the model.

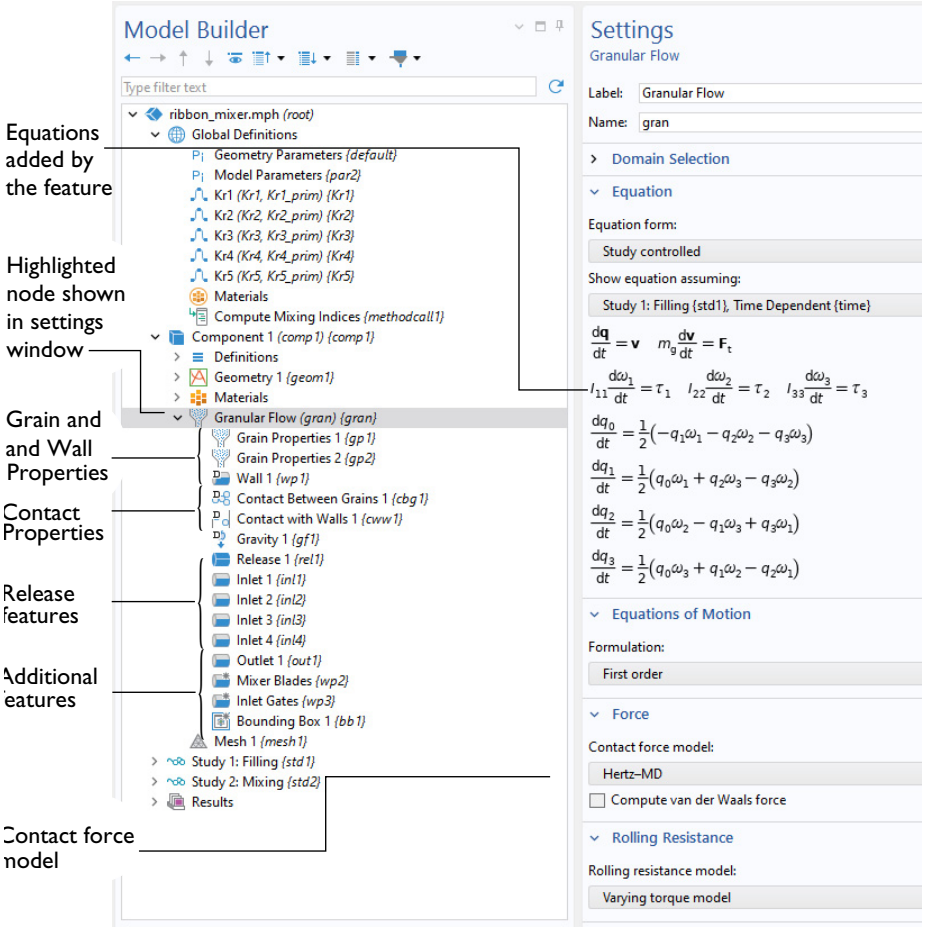

Figure 1: The Model Builder (to the left), and the Settings window for the selected feature node (to the right). The Equation section in the Settings window shows the model equations.