



ROCKY

AND

ANSYS®

INTEGRATION

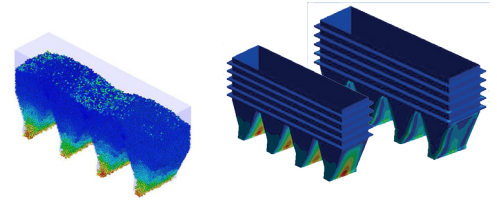
Rocky is fully integrated with the ANSYS Workbench suite of products, providing engineers with the ability to perform coupled analysis of particles simulation together with other physics such as structural and fluids. Such coupling can be performed using both 1-way and 2-way approaches, depending upon the nature of the problem to be solved.

Engineers can use loads for static structural and transient analysis, as well as modal and frequency analyses. Coupling DEM and CFD simulations are also possible.

Rocky is integrated as a new System into the Workbench schematic. As an example, any geometry from ANSYS Workbench is fed into Rocky, and Rocky then transfers back particle loads into the geometry from the DEM simulation.

Including Rocky into the Workbench schematic will also allow engineers to perform design optimization analysis, since Rocky input and output parameters are exposed to other ANSYS modules, such as DesignXplorer.

Structural Analysis of a Silo - Particle forces were obtained with ROCKY



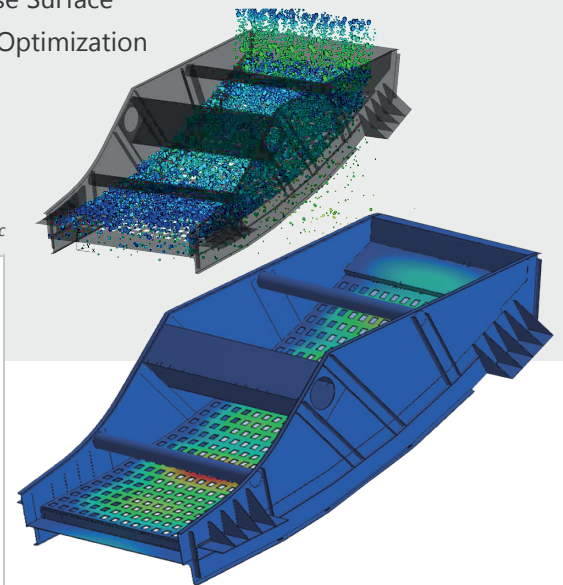
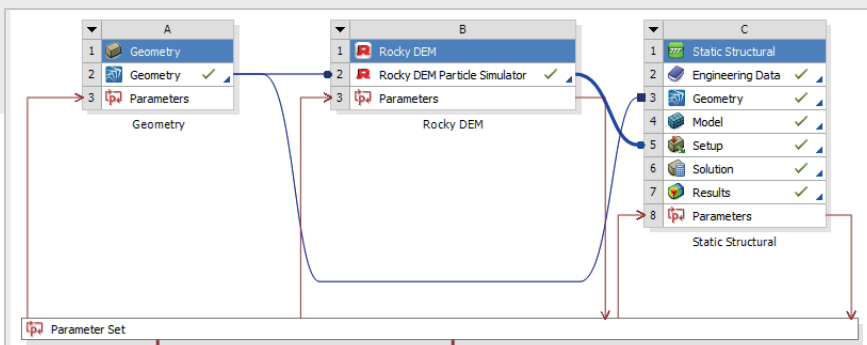
ROCKY'S DEEP INTEGRATION WITH ANSYS TOOLS INCLUDES:

- A new System in the ANSYS Workbench schematic
- A direct geometry link with SpaceClaim
- 1-way coupling with ANSYS Mechanical
- 1-way and 2-way coupling with ANSYS Fluent
- Full DesignXplorer Support
- True Multiple Physics Simulation abilities
- Pervasive Parametric Support for Input / Output variables

THE ENGINEERING BENEFITS OF ROCKY'S PARAMETRIC ANALYSIS FEATURES INCLUDE:

- Design Exploration
- Parameters Correlation
- Response Surface
- Design Optimization

ANSYS Workbench Project Schematic



ABOUT ROCKY

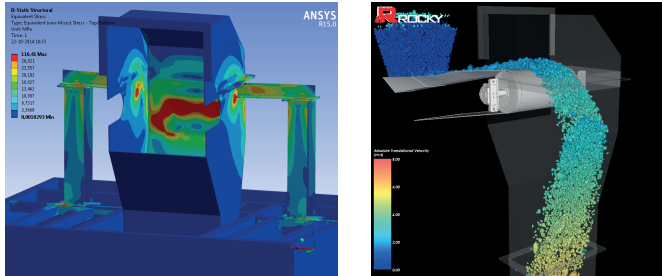
Rocky is a powerful 3D Discrete Element Modeling (DEM) program that quickly and accurately simulates the granular flow behavior of different shaped and sized particles using both GPU and CPU processing methods. Rocky is also tightly coupled with ANSYS Workbench, which enables engineers to conduct both structural and fluid Finite Element Analysis (FEA) using Rocky simulation data.

ABOUT ANSYS

ANSYS brings clarity and insight to customers' most complex design challenges through fast, accurate and reliable engineering simulations. ANSYS technology enables organizations to predict with confidence that their products will thrive in the real world. Customers trust ANSYS software to help ensure product integrity and drive business success through innovation.

ROCKY AND ANSYS MECHANICAL COUPLING

For the past several releases, Rocky has been coupled with Finite Element Analysis (FEA) ANSYS software, allowing engineers to evaluate the tensions stresses and forces generated by granular matter as it interacts with materials handling equipment, such as transfer chutes and conveyor belts.



Result of the analysis in ANSYS Mechanical with the equivalent stress

Rocky has a fast and accurate method of coupling the DEM and FEM simulations on non-coincidental meshes. The loads estimated by Rocky analyses are passed to ANSYS to predict the structural response, or one-way coupling.

This coupling enables the following analyses to be completed on Rocky data:

- Static Analysis
- Transient Analysis
- Harmonic Analysis
- Fatigue Analysis
- Dynamic Analysis

ROCKY AND ANSYS FLUENT COUPLING: A POWERFUL TOOL FOR SIMULATING GRANULAR-FLUID SYSTEMS

The coupled DEM - CFD approach is a promising alternative for modeling granular-fluid systems, enlarging the range of coupled particle-fluid processes that can be managed with numerical simulations. Complex phenomena such as pneumatic conveying, granular drying, slurry flow inside grinding mills, or even chemical reactions between particles and fluids can be simulated and analyzed with these powerful tools.

Considering the industry necessities for both single phase and multiphase simulations, both 1-way and 2-way coupling between Rocky and ANSYS Fluent are provided.

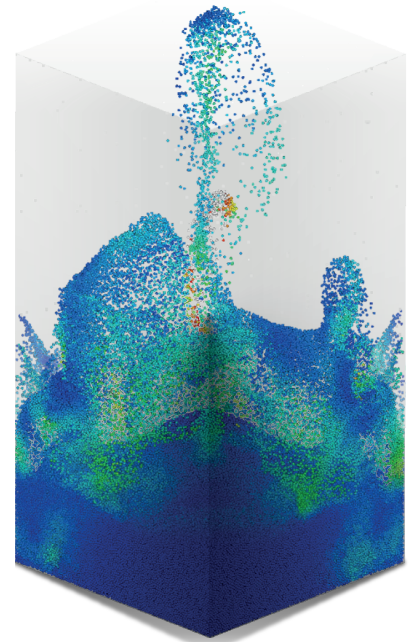
The coupling of the terms of the Navier-Stokes equation of fluid-to-particle interactions, which by taking into account drag, lift, buoyancy, virtual mass, angular momentum and other forces, provide physically consistent and very trustworthy results.

The work that Rocky DEM and ANSYS accomplished also allows ANSYS Fluent to work with Rocky in parallel CPU and GPU capabilities, which leads to very fast DEM-CFD simulation results.

This parallel processing methodology also opens up the possibility of simulating large multiphase flow problems, with many millions of different sized and shaped particles - historically a weak spot with DEM technology.

A plugin designed for ANSYS Fluent makes the combined DEM - CFD setup task easy, less error prone, and guarantees consistency between both physics.

Rocky - ANSYS Fluent coupling is a powerful tool for designing and troubleshooting particulate processes using simulation technology, enabling engineers to analyze a large range of processes in many different industries including oil and gas (e.g. drilling), agroindustry (e.g. grain transportation and drying), pharmaceuticals (e.g. transport), mining (e.g. mineral washing), and many others.



Fluidized bed simulation
(2-way coupling with ANSYS Fluent)