



2025/R2

ANSYS ROCKY MODULES & SCRIPTS

Installation Guide

/Table of Contents

/ Ansys Rocky Download Links (Windows/Linux)	3
/ Ansys Rocky Documentation	3
/ SDK-Modules: ROCKY_SDK_MODULES_2025R2	3
/ Ready-to-use Modules	4
/ Source Codes	13
/ Functional Modules	13
/ Current Functionalities – Code Examples	19
/ Simple Modules - Code Examples	21
/ Setting up the environment for Rocky SDK in Windows	23
/ Setting up the environment for Rocky Solver SDK in Linux	25
/ Additional Package: ROCKY_ADDITIONAL_PACKAGE_2025R2	27
/ Ansys Rocky Integrations	29
/ Installation: Ansys Motion	29
/ Installation: Adams	37
/ Installation: Ansys Minerva	41
/ Installation: Ansys optiSLang	66
/ Installation: Ansys EnSight	70
/ Installation: Ansys System Coupling	72
/ PrePost Scripts Repository	73
/ Material Wizard	73
/ Material Storage	73
/ Calibration Suite	74
/ Coating Visibility Wizard	74
/ Job Schedulers Scripts	74
/ System Requirements	75
/ Batch Mode Execution	76

/ Ansys Rocky Download Links (Windows/Linux)

Please use the following link to download the latest **Ansys Rocky** software and related packages: [Link](#)

/ Ansys Rocky Documentation

Rocky **User** and **Technical Documentation** are on **Ansys Help**, the Ansys documentation portal. as well as **Rocky Tutorials**, **Verification Manual** and all the **Modules and Scripts Documentation**. [Click here](#) to access the portal.

Note: The **PrePost Scripting** and **Solver SDK Manual** can be accessed in the [Ansys Developer Portal](#).

/ SDK-Modules: ROCKY_SDK_MODULES_2025R2

This package contains:

- **Ansys Rocky SDK files (SDK Folder):** The software development kit that is required to build custom modules. Using the Solver SDK, you can develop custom modules for Rocky.
- **Compiled version of Ready-to-use Rocky Modules (Ready-To-Use Modules Folder):** These modules are specialty models to expand the core solver capabilities. These additional modules are ready to be used.
- **Source Codes (Source Codes folder):**
 - **Functional Modules (Functional Folder):** The modules that have been developed to extend Rocky capabilities in a real-case scenario, allowing module users to add external forces, remove particles, modify particle properties, gather data and more.
 - **Current Functionalities – Code Examples (Current Models Folder):** The modules that have the exact same behavior as some of the existing functionality in the current Rocky UI (user interface). The source code is provided here for you to use as a starting point for your own customization needs.
 - **Simple Modules - Code Examples (Simple Modules Folder):** The modules that have been created as **simple examples** of certain

Solver SDK functionalities, but currently have **no physical or functional meaning**. Therefore, these modules should not be considered for usage in a real case. The source code can instead be reviewed as a good example of Solver SDK usage.

/ Ready-to-use Modules

To install a module, download and unzip the file into the "Ready-To-Use Modules" folder (OS-specific location provided below) and then restart Rocky:

Windows: %USERPROFILE%\Documents\Rocky\Modules

Linux: ~/.Rocky/Modules

You can find documentation for all available modules in the **Rocky Module Manual** on the [Ansys Help Portal](#).

Type	Name	Description	Version
Adhesion Model	Bond Model	This module is designed to simulate the behavior of agglomerate materials by using a model which bonds particles together using cylindrical, massless, spring-dashpot connections. These bonds can be broken at any time if the tensile or shear stresses exceed prescribed values. Also, the module creates bonds for particles recently unfrozen by the "Particle Freeze Outside Region" module.	1.3.0
	Velocity-Dependent Adhesive Force	Enables you to use a parametric adhesive force model that takes into account the particle's impact velocity. This model is best suited for spherical particles. Application examples include snow accretion.	1.1.0
Boundary Condition	Region Outlet	Enables you to select a Region of Interest as an outlet for particles; any particles entering this region will be removed from the simulation, similarly to what happens when a particle crosses the limits of the simulation domain. Application examples include blast furnaces.	2.0.0
	Boundary Outlet	Enables you to define an imported geometry component and/or default	1.0.1

Type	Name	Description	Version
		conveyor as an outlet for particles. This module then removes from the simulation any particle that becomes in contact with that geometry. Application examples include blast furnaces.	
Breakage Model	Overlap-Based Breakage Model	This module enables you to use a custom, instantaneous breakage model and a custom fragments size distribution model to account for the physical effects of compression upon particle breakage, where the contact overlap is used as a measurement of particle deformation. Application examples include jaw, gyratory, or roll crushers.	1.0.1
	TBM Breakage Model	This module re-implements the same TBM (instantaneous) Breakage Model and the same Uniform Size Fragment Distribution model that comes by default with Rocky. Its calculations will be applied according to the model developed by TAKRAF, which uses statistical data from experimental breakage tests. It can be used as an example of Solver SDK usage.	1.0.0
CFD Coupling	Parametric Gidaspow, Bezburuah & Ding	Enables you to use a parametric version of the Gidaspow, Bezburuah & Ding Drag Law for computing the drag coefficient for the drag force between particles and fluids when CFD Coupling 2-Way with Ansys Fluent is enabled.	1.0.0
	Parametric Ranz & Marshall Law	Enables you to use a parametric version of the Ranz & Marshall Convective Heat Transfer Law included by default in Rocky for interactions between spherical particles and fluids when CFD Coupling is enabled. This model is made parametric by adding two free parameters to the calculation of the Nusselt number.	1.0.3
	Dhole, Chhabra & Eswaran Drag Law	Enables you to use a power-law drag correlation for your CFD Coupling simulations that is specifically designed to model non-Newtonian fluids that obey the Ostwald-de Waele (power-law) relationship. This module is designed to work with Rocky's 1-Way Constant, 1-	1.0.0

Type	Name	Description	Version
		Way Fluent Steady State, or 2-Way Fluent (unresolved) coupling methods. This module is a drag factor correction that accounts for the fluid non-newtonian properties in the drag correlation, it does not account for the particle shape nor the local volume fraction. Therefore it is suited for spherical particles in dilute flows.	
	CFD-Coupled Particle Combustion (Beta)	Enables you to consider the reaction of coal particles during combustion. Two reactions happen in parallel: $C + O_2 \rightarrow CO_2$, and $C + CO_2 \rightarrow 2CO$. Note that this is a Beta version and have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve reported defects. However, your feedback will help us improve the overall quality of the module. We will not guarantee that the projects using this version of the module will run successfully when a stable version is released.	2.0.2
Contact Model	Leeds Contact Model	Enables you to use the Leeds Contact Model for both the Normal Force and Adhesive Force used in the project. Application examples include cohesive frictional powders.	2.0.0
	Stress-Based Multi-Contact Normal Force	Enables you to use the Stress-Based Multi-Contact Model proposed by K. Giannis et al. for the Normal Forces used in the project.	1.0.0
	Thornton-Ning Normal Force	This module implements the Thornton Ning Normal Force Model. Application examples include elasto-plastic materials.	1.0.0
	[New] Edinburgh Elasto-Plastic-Adhesive (EEPA)	The Edinburgh Elasto-Plastic-Adhesive (EEPA) module implements an elasto-plastic contact force model combined with an adhesive force model, in which the adhesion effect is increased due to contact area enlargement caused by plastic deformation.	1.0.0
Data Gathering	Coordination Number	Informs the Coordination Number (number of contacts) for each particle. Can be especially useful for mixing	1.0.0

Type	Name	Description	Version
		simulations to evaluate the homogeneity and for powder compression simulations to evaluate the packing efficiency.	
	Normal and Tangential Energy Dissipation	This module re-implements and then splits into its normal and tangential components the same Particle Curve calculations you get in Rocky when the Energy Dissipation option is enabled on the Inter-group Collision Statistics module; and similar Particle Property calculations you get in Rocky when the Power option is enabled on the Inter-particle Collision Statistics module, only instead of power, it calculates energy.	1.0.1
	Particles Stress Tensor Property Components	This module re-implements and then displays as nine individual Properties the same calculations that generate the components for Stress Tensor, which is a property for Particles that comes by default with Rocky. This module provides data for all particles but the calculations are valid only for spherical particle shapes. It can be used as an example of Solver SDK usage.	1.0.0
Dynamic Domain	Particle Freeze when Settled	Reduces the computational cost of simulating projects involving large amounts of non-moving particles. It does this by freezing and then disabling from contact calculations the particles that are not actively involved in the simulation. Can be especially useful for simulations with static beds--even ones that grow over time.	1.0.0
	Disable Collision Outside Region	Reduces the computational cost of simulating projects by disabling from contact calculations the particles that are outside a defined Region of Interest (ROI).	1.0.0
	Particle Freeze Outside Region	Reduces the computational cost of simulating projects by freezing and disabling from contact calculations the particles that are outside the desired regions.	2.0.0
External Force	1-Way CFD Field Coupling	This module enables you to couple 1-way with Rocky a fluid velocity field that has	1.0.2

Type	Name	Description	Version
		been imported into Rocky as a Point Cloud. It can be used as an alternative to the 1-Way Fluent Steady State coupling method included with Rocky, as it enables you make use of the fluid results from various external CFD solvers. This module also re-implements the Schiller & Naumann Drag Law that comes by default with Rocky, and is therefore best suited for spherical particles. It can be used as an example of Solver SDK usage.	
	Electromagnetic Coupling (Beta)	The Electromagnetic Coupling (Beta) module enables you to add electromagnetic physics to the simulation, such as Electrostatic forces for particle-particle and particle-boundary interactions, charging and discharging effects on particles due contact with walls, or additional forces on charged or dipole particles immersed on electric/magnetic fields. Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	1.1.0
	[New] Gravity Variable	The Gravity Variable Module allows defining a time-dependent gravitational field. Users can specify the magnitude and direction of gravity acceleration as a function of time, enabling simulations in dynamic environments.	1.0.0
External Heat	Heat Generation Region	Enables you to select a Region of Interest in which a specific heat source per unit area will be applied to particles within the region, in order to simulate radiation effects. It can be used as an example of Solver SDK usage.	2.0.0

Type	Name	Description	Version
	Frictional Heating	Enables you to consider the heat generation due to friction in particle-particle and particle-boundary collisions. Application examples include grinding processes and sanding.	1.1.0
	Thermal Radiation	Enables you to add radiation heat exchange between particle-particle interactions that occur during the simulation. Also enables you to set geometries as radiation heat sources for the particles.	2.0.1
Joint Model	Leblicq Joint Model	Enables you to use the Leblicq Joint Model when defining flexible Straight Fiber particle shapes.	1.0.1
	Heterogeneous Custom Fiber (Beta)	This module enables you to set different elastic, plastic and breakage parameters for each joint Disclaimer: Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	1.0.0
Liquid Film	Liquid Injection Sphere Clouds	Enables you to define one or more spherical regions in which any particle that enters will receive a liquid injection.	1.0.0
	Liquid Drying Region	Enables you to select one or more Regions of Interest in which liquid film that particles may contain is evaporated from all particles inside it. Application examples include tablet/food coating, food drying, and mixing processes.	2.0.0
	Liquid Drying by Heat Flux	Enables you to model the liquid film evaporation due to latent heat and the addition of the dried solute to the mass and volume of the particles. Application examples include tablet/food coating, food drying, and mixing processes.	1.0.2
	Liquid Bridge Model	Enables you to include the effects of liquid films around particles in your	2.0.0

Type	Name	Description	Version
		simulation. This type of adhesion model can be useful for studying wet particular processes.	
	Liquid Injection Region	Enables you to select one or more Regions of Interest in which liquid film is added to all particles inside it. Application examples include tablet/food coating, food drying, and mixing processes.	2.0.0
	CFD-Coupled Particle Drying	Enables you to model how the particle's moisture content influences the rate of evaporation considering either a constant or transient humidity field, and by using the local pressure, temperature, and velocity fields from a coupled 1-Way Constant, 1-Way Fluent Steady State, or 2-Way Fluent simulation.	2.1.0
Particle Modification	Liquid Absorption	Enables you to use a Liquid Absorption model to consider the increase of particle density due to liquid absorption.	1.0.1
	[New] DEM Liquid Spray	The DEM Liquid Spray module enables you to model spray droplets using the DEM method. On top of that, the module gives users the ability to create a simple spray effect on spray droplets, spreading particles apart in a cone-like fashion, to represent a real spray pattern.	1.0.0
SPH	SPH Air Drag (Beta)	This module estimates and applies the drag force caused by surrounding air flowing at constant velocity onto a liquid flow simulated with SPH. Disclaimer: Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	1.0.0

Type	Name	Description	Version
	[New] SPH Disable Wall Corrections	Utility module for disabling corrections for kernel deficiency near walls, either on all walls or on specific ones.	1.0.0
	SPH Non-Newtonian Models (Beta)	This module implements three non-Newtonian models for SPH: - Power law model - Bingham plastic model - Herschel-Bulkley model Disclaimer: Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	1.0.0
	SPH Point Cloud Air Drag (Beta)	This module uses a given point cloud file as the air velocity fields for drag calculations. It expects the point cloud .txt file to have the following fields to describe air velocity: x-velocity y-velocity z-velocity Disclaimer: Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	1.0.0
	SPH Freeze Outside Region (Beta)	Reduces the computational cost of simulating projects by freezing and disabling from calculations the SPH elements that are outside a defined Region of Interest (ROI). Disclaimer: Note that beta features have not been fully tested and validated.	1.0.0

Type	Name	Description	Version
		Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	
	[New] SPH Temperature Dependent Viscosity (Beta)	This module uses a viscosity X temperature table to update viscosity of SPH elements depending on their temperature. The table is provided by the user through a .csv file. Disclaimer: Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	1.0.0
Thermal Contact Model	Nonlinear Thermal Contact Model	Enables you to use a nonlinear extension of the internal heat conduction model that comes by default in Rocky. In this implementation, both the thermal conductivity and the specific heat are considered functions of the temperature.	1.0.1
Variable Property	Velocity Dependent Restitution Model	The constant values of the coefficient of restitution specified per material interaction are overridden by this module, which sets values dependent on the normal component of the impact velocity at a collision.	1.0.1
Wear Model	Eroded Mass Analyses	Enables you to use one or more empirical erosion models to calculate the eroded mass due to particle-boundary collisions. Models include the McLaury model, the Oka model, and the Finnie model. Application examples include oil & gas pipeline erosion, and bulk material handling.	1.2.1

Modules Changes Released on July, 16 2025 for SDK version 2025 R2.0

New Modules

- A new DEM Liquid Spray module was created;
- A new Variable Gravity module was created;
- A new Disable Wall Corrections module was created;
- A new Edinburgh-Elasto-Plastic-Adhesion contact model was created;
- A new Temperature Variable Viscosity (Beta) was created.

Improvements

- The Particle Freeze when Settled module was updated to include a delay time for the freezing.
- The Electromagnetic Coupling module was updated to include field modulations.
- In the 1-Way CFD Field Coupling module, the velocities in the point cloud header were changed from [u, v, w] to [x-velocity, y-velocity, z-velocity].

Bug Fixes

- In the 1-Way CFD Field Coupling module, the possibility of division by zero was fixed when the relative velocity was equal to zero;
- In the CFD Particle Combustion module, an error that could prevent users from using the module on Linux distros was fixed.

/ Source Codes

/ Functional Modules

Type	Name	Description	Version
Adhesion Model	Velocity-Dependent Adhesive Force	Enables you to use a parametric adhesive force model that takes into account the particle's impact velocity. This model is best suited for spherical particles. Application examples include snow accretion.	1.1.0

Type	Name	Description	Version
Boundary Condition	Region Outlet	Enables you to select a Region of Interest as an outlet for particles; any particles entering this region will be removed from the simulation, similarly to what happens when a particle crosses the limits of the simulation domain. Application examples include blast furnaces.	2.0.0
	Boundary Outlet	Enables you to define an imported geometry component and/or default conveyor as an outlet for particles. This module then removes from the simulation any particle that becomes in contact with that geometry. Application examples include blast furnaces.	1.0.1
Breakage Model	TBM Breakage Model	This module re-implements the same TBM (instantaneous) Breakage Model and the same Uniform Size Fragment Distribution model that comes by default with Rocky. Its calculations will be applied according to the model developed by TAKRAF, which uses statistical data from experimental breakage tests. It can be used as an example of Solver SDK usage.	1.0.0
	Overlap-Based Breakage Model	This module enables you to use a custom, instantaneous breakage model and a custom fragments size distribution model to account for the physical effects of compression upon particle breakage, where the contact overlap is used as a measurement of particle deformation. Application examples include jaw, gyratory, or roll crushers.	1.0.1
CFD Coupling	Parametric Ranz & Marshall Law	Enables you to use a parametric version of the Ranz & Marshall Convective Heat Transfer Law included by default in Rocky for interactions between spherical particles and fluids when CFD Coupling is enabled. This model is	1.0.3

Type	Name	Description	Version
		made parametric by adding two free parameters to the calculation of the Nusselt number.	
	Parametric Gidaspow, Bezburuah & Ding	Enables you to use a parametric version of the Gidaspow, Bezburuah & Ding Drag Law for computing the drag coefficient for the drag force between particles and fluids when CFD Coupling 2-Way with Ansys Fluent is enabled.	1.0.0
	Dhole, Chhabra & Eswaran Drag Law	Enables you to use a power-law drag correlation for your CFD Coupling simulations that is specifically designed to model non-Newtonian fluids that obey the Ostwald–de Waele (power-law) relationship. This module is designed to work with Rocky's 1-Way Constant, 1-Way Fluent Steady State, or 2-Way Fluent (unresolved) coupling methods. This module is a drag factor correction that accounts for the fluid non-newtonian properties in the drag correlation, it does not account for the particle shape nor the local volume fraction. Therefore it is suited for spherical particles in dilute flows.	1.0.0
Contact Model	Leeds Contact Model	Enables you to use the Leeds Contact Model for both the Normal Force and Adhesive Force used in the project. Application examples include cohesive frictional powders.	2.0.0
	Stress-Based Multi-Contact Normal Force	Enables you to use the Stress-Based Multi-Contact Model proposed by K. Giannis et al. for the Normal Forces used in the project.	1.0.0
	Thornton-Ning Normal Force	This module implements the Thornton Ning Normal Force Model. Application examples include elasto-plastic materials.	1.0.0

Type	Name	Description	Version
Data Gathering	Normal and Tangential Energy Dissipation	This module re-implements and then splits into its normal and tangential components the same Particle Curve calculations you get in Rocky when the Energy Dissipation option is enabled on the Inter-group Collision Statistics module; and similar Particle Property calculations you get in Rocky when the Power option is enabled on the Inter-particle Collision Statistics module, only instead of power, it calculates energy.	1.0.1
	Coordination Number	Informs the Coordination Number (number of contacts) for each particle. Can be especially useful for mixing simulations to evaluate the homogeneity and for powder compression simulations to evaluate the packing efficiency.	1.0.0
Dynamic Domain	Particle Freeze when Settled	Reduces the computational cost of simulating projects involving large amounts of non-moving particles. It does this by freezing and then disabling from contact calculations the particles that are not actively involved in the simulation. Can be especially useful for simulations with static beds—even ones that grow over time.	1.0.0
	Disable Collision Outside Region	Reduces the computational cost of simulating projects by disabling from contact calculations the particles that are outside a defined Region of Interest (ROI).	1.0.0
	Particle Freeze Outside Region	Reduces the computational cost of simulating projects by freezing and disabling from contact calculations the particles that are outside the desired regions.	1.0.0
	SPH Freeze Outside Region (Beta)	Reduces the computational cost of simulating projects by freezing and disabling from calculations the SPH elements that are outside a defined Region of Interest (ROI).	1.0.0

Type	Name	Description	Version
		Disclaimer: Note that beta features have not been fully tested and validated. Ansys, Inc. makes no commitment to resolve defects reported against these prototype features. However, your feedback will help us improve the overall quality of the product. We will not guarantee that the projects using this beta feature will run successfully when the feature is finally released so you may, therefore, need to modify the projects.	
External Force	1-Way CFD Field Coupling	This module enables you to couple 1-way with Rocky a fluid velocity field that has been imported into Rocky as a Point Cloud. It can be used as an alternative to the 1-Way Fluent Steady State coupling method included with Rocky, as it enables you make use of the fluid results from various external CFD solvers. This module also re-implements the Schiller & Naumann Drag Law that comes by default with Rocky, and is therefore best suited for spherical particles. It can be used as an example of Solver SDK usage.	1.0.2
External Heat	Thermal Radiation	Enables you to add radiation heat exchange between particle-particle interactions that occur during the simulation. Also enables you to set geometries as radiation heat sources for the particles.	2.0.1
	Heat Generation Region	Enables you to select a Region of Interest in which a specific heat source per unit area will be applied to particles within the region, in order to simulate radiation effects. It can be used as an example of Solver SDK usage.	2.0.0
	Frictional Heating	Enables you to consider the heat generation due to friction in particle-particle and particle-boundary	1.1.0

Type	Name	Description	Version
		collisions. Application examples include grinding processes and sanding.	
Joint Model	Leblicq Joint Model	Enables you to use the Leblicq Joint Model when defining flexible Straight Fiber particle shapes.	1.0.1
Liquid Film	CFD-Coupled Particle Drying	Enables you to model how the particle's moisture content influences the rate of evaporation considering either a constant or transient humidity field, and by using the local pressure, temperature, and velocity fields from a coupled 1-Way Constant, 1-Way Fluent Steady State, or 2-Way Fluent simulation.	2.1.0
	Liquid Injection Region	Enables you to select one or more Regions of Interest in which liquid film is added to all particles inside it. Application examples include tablet/food coating, food drying, and mixing processes.	2.0.0
	Liquid Injection Sphere Clouds	Enables you to define one or more spherical regions in which any particle that enters will receive a liquid injection.	1.0.0
	Liquid Drying Region	Enables you to select one or more Regions of Interest in which liquid film that particles may contain is evaporated from all particles inside it. Application examples include tablet/food coating, food drying, and mixing processes.	2.0.0
	Liquid Drying by Heat Flux	Enables you to model the liquid film evaporation due to latent heat and the addition of the dried solute to the mass and volume of the particles. Application examples include tablet/food coating, food drying, and mixing processes.	1.0.2
Particle Modification	Liquid Absorption	Enables you to use a Liquid Absorption model to consider the increase of particle density due to liquid absorption.	1.0.1

Type	Name	Description	Version
Thermal Contact Model	Nonlinear Thermal Contact Model	Enables you to use a nonlinear extension of the internal heat conduction model that comes by default in Rocky. In this implementation, both the thermal conductivity and the specific heat are considered functions of the temperature.	1.0.1
Variable Property	Velocity Dependent Restitution Model	The constant values of the coefficient of restitution specified per material interaction are overridden by this module, which sets values dependent on the normal component of the impact velocity at a collision.	1.0.1
Wear Model	Eroded Mass Analyses	Enables you to use one or more empirical erosion models to calculate the eroded mass due to particle-boundary collisions. Models include the McLaury model, the Oka model, and the Finnie model. Application examples include oil & gas pipeline erosion, and bulk material handling.	1.2.1

/ Current Functionalities – Code Examples

Type	Name	Description	Version
Breakage Model	AbT10 Breakage Model	This module re-implements the same Ab-T10 (instantaneous) Breakage Model that comes by default with Rocky. It can be used as an example of Solver SDK usage.	1.0.0
	Griffith Energy Criterion Breakage Model	This module re-implements a reduced version of the same Griffith Energy Criterion (discrete) Breakage Model that comes by default with Rocky, only without the contribution of rotational deformation. It works with flexible solids, and also works, when a correction of the bending deformation is applied, with flexible Fiber and flexible Shell particle shapes. It can be used as an example of Solver SDK usage.	2.0.0

Type	Name	Description	Version
CFD Coupling	CFD Coupling Example	This module re-implements some of the same CFD Coupling laws and coefficients that comes by default with Rocky, including drag (Schiller & Naumann), lift (Saffman), torque (Denis), and virtual mass (constant). It can be used as an example of Solver SDK usage.	1.0.0
Collision Statistics	Boundary Collision Statistics API	This module re-implements the built-in Boundary Collision Statistics that comes by default with Rocky	1.0.0
	Inter-group Collision Statistics API	This module re-implements the built-in Inter-group Collision Statistics that comes by default with Rocky	1.0.0
	Inter-particle Collision Statistics API	This module re-implements the built-in Inter-particle Collision Statistics that comes by default in Rocky	1.0.0
Contact Model	Coulomb Limit Tangential Force	This module re-implements the same Coulomb Limit Tangential Force contact model that comes by default with Rocky. It can be used as an example of Solver SDK usage.	1.0.2
	Hysteretic Linear Spring Normal Force	This module re-implements the same Hysteretic Linear Spring Normal Force Model that comes by default with Rocky. It can be used as an example of Solver SDK usage. Application examples include elasto-plastic materials.	1.0.0
Data Gathering	Particles Stress Tensor Property Components	This module re-implements and then displays as nine individual Properties the same calculations that generate the components for Stress Tensor, which is a property for Particles that comes by default with Rocky. This module provides data for all particles but the calculations are valid only for spherical particle shapes. It can be used as an example of Solver SDK usage.	1.0.0
Impact Energy Model	Elastic Deformation Impact Energy	This module re-implements the same internal Impact Energy model that comes by default with Rocky. It can be used as an example of Solver SDK usage.	1.0.0

Type	Name	Description	Version
Joint Model	Bilinear Elastoplastic Joint Model	This module implements a reduced version of the same Bilinear Elastoplastic Joint Model that comes by default with Rocky, but without the ability to define the Torsion Angle Limit and joint Failure definitions. In this module, both the torsion and the tension/compression at the joints are modeled using the standard linear elastic-viscous model in Rocky. It can be used as an example of Solver SDK usage. Application examples include agricultural equipment.	1.0.0
Wear Model	Archard Wear Model	This module re-implements the same Shear Work Proportionality (Archard's Law) surface wear model that comes by default with Rocky. It can be used as an example of Solver SDK usage.	1.0.0

/ Simple Modules - Code Examples

Type	Name	Description	Version
Adhesion Model	Transient Adhesive Force	Enables you to use a Transient Adhesive Force model that will increase over time the adhesive forces between particles and between particles and boundaries. It can be used as an example of Solver SDK usage.	2.0.0
Data Gathering	Contact Curves	Enables Rocky to create additional Contact Frequency and Contact Number curves for both Geometries and Particles. It can be used as an example of Solver SDK usage.	1.0.0
Particle Identification	Spherical Region	Enables you to define a spherical region and then mark particles inside of it for later analysis. It can be used as an example of Solver SDK usage.	1.0.0
Particle Modification	Transient Size Scale	Enables you to use a Transient Size Scale to increase the particle volume and mass over the predefined time interval. It can be used as an example of Solver SDK usage.	1.1.0

Type	Name	Description	Version
Field Force	Field Force	Enables you to specify the name of the Point Cloud that was already defined with an imported force field .txt file, which will then be used as an additional body force acting on particles. It can be used as an example of Solver SDK usage.	1.0.0
SPH Motion	SPH LES Turbulence Model	This module re-implements the SPH turbulence model (LES), the viscosity model Morris and the Free Slip boundary type that come by default with Rocky. It can be used as an example of Solver SDK usage.	1.0.0
	SPH Spray Nozzle	Applies an acceleration to SPH Elements Inside an ROI to generate some scattering in order to mimic a spray nozzle.	1.0.0
	Air Knife Actuator Sensor	This module enables you to define a sensor through a region of interest and link it with a 1-way CFD coupling using the fluid velocity field that has been imported into Rocky as a Point Cloud. The flow field activation/deactivation will be controlled by the sensor.	1.0.0
	Motion Frame	Enables you to define a translation or rotation motion to be applied to the gravity center of geometry. It can be used as an example of Solver SDK usage.	1.0.0
	Geometry Actuator Sensor	This module enables you to implement a Sensor through a Region of Interest and an actuator motion. The motion of the Actuator geometry will be controlled by the sensor.	1.0.0
	Compression Motion	This module enables you to implement a compression motion. The motion will be reverted when the maximum force is reached.	1.0.0

/ Setting up the environment for Rocky SDK in Windows

To prepare your system for building custom modules for Rocky using the Solver SDK, you need to install and set up a few components. Following are the steps to set up the environment,

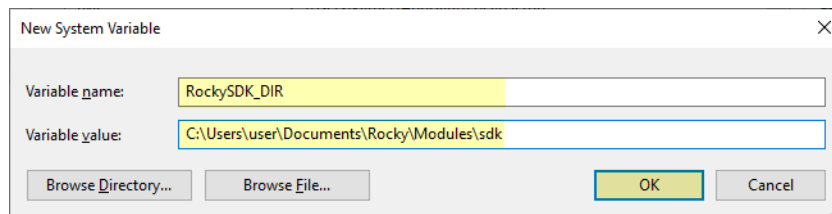
/ Download and Setup Rocky Solver SDK

Download the SDK Package corresponding to your OS from the **Rocky SDK Package Download** page and unzip the file at your preferred location.

1. Download the SDK and Modules package for Windows.

Reminder: The Solver SDK version you install allows you to make modules only for the same version of the Rocky product. This means that modules made from the 2025 R2 Solver SDK, for example, will work only in Rocky 2025 R2

2. Extract the files, and copy (or cut) and paste the **sdk** folder into a folder of your choice. (For example, within your **%HOMEPATH%/Documents/Rocky/Modules** folder.)
3. From the **Windows Search bar**, type **Edit the system environment variables**, and then open the **Control Panel** result that appears.
4. From the **System Properties** dialog, on the **Advanced** tab, click the **Environment Variables...** button.
5. From the **Environment Variables** dialog, from the **System variables** box, click the **New...** button.
6. From the **New System Variable** dialog, define the **Variable name** as **RockySDK_DIR** and the **Variable value** with the folder path of the **SDK subdirectory** (as shown), and then click **OK**.



/ Cuda Toolkit Setup (Windows)

1. Install and configure the **CUDA Toolkit 12.8**

Important: The 2025 R2 (and later) versions of the Rocky Solver SDK are compatible **only with version 12.8** of the CUDA toolkit.

2. Open the **CUDA Toolkit 12.8 Archive** page from the NVIDIA official website and download the installer for your operational system.
3. Open the downloaded *.exe, and then extract the installation files as instructed.
4. Follow the NVIDIA Installer using the **Custom (Advanced)** options.
5. On the **Custom installation options** step, under the **CUDA** component, ensure that the **Runtime, Development** and **Visual Studio Integration** checkboxes are selected.
6. Follow the installer instructions to complete the installation.
7. Ensure that an environment variable called **CUDA_PATH_V12_8** was created.
8. Confirm the location of the installed CUDA Toolkit on your system.

Tip: This is typically **C:/Program Files/NVIDIA GPU Computing Toolkit/CUDA/v12.8**

/ Build Tools Setup (Windows)

Download and install **Visual Studio Build Tools**.

1. Go to Microsoft | Visual Studio official website and download **Build Tools for Visual Studio 2022 LTSC 17.6.17**.
 - a. From the Microsoft official website, access the **Visual Studio 2022** page.
 - b. On the page that opens, roll down until you find Visual Studio downloads list, and access the **Tools for Visual Studio** drop-down list and click the **long term servicing baselines (LTSCs)** link.
 - c. From the page that opens, access the **Release History** for **version 17.6**.
 - d. From the **Fixed version bootstrappers** list, find the **17.6.17 Version** (click **Ctrl + F** and search for **17.6.17**) and click the link to download the .exe installer for the **Build Tools**.
2. Open the downloaded .exe.
3. From the **Installing** screen:

- Select **Desktop development with C++** (under **Desktop & Mobile** on the **Workloads** tab)
 - Ensure that **build tools**, **Windows SDK** and **C++ CMake tools for Windows** (under **Optional** on the **Installation details** panel) .
4. Click **Install** and finish installation accordingly.
 5. Go to the **C:\Program Files\NVIDIA GPU Computing Toolkit\CUDA\v12.8\extras\visual_studio_integration\MSBuildExtensions** folder and copy its files.
 6. Go to the **C:\Program Files (x86)\Microsoft Visual Studio\2022\BuildTools\MSBuild\Microsoft\VC\v170\BuildCustomizations** folder and paste the files you just copied.

More information on **setting up the environment** and creating and building your own modules can be found in Tutorial 23 - SDK and Solver SDK and in the Solver SDK Manual.

/ Setting up the environment for Rocky Solver SDK in Linux

To prepare your system for building custom modules for Rocky using the Solver SDK, you need to install and set up a few components. Following are the steps to set up the environment,

/ Download and Setup Rocky Solver SDK

Download the SDK Package corresponding to your OS from the **Rocky SDK Package Download** page and unzip the file at your preferred location.

Create an environment variable called **RockySDK_DIR** that points to the location you extracted the SDK (in this case, we extracted the zip file in **~/.Rocky/Modules**):

```
$ echo "export RockySDK_DIR=$HOME/.Rocky/Modules/sdk" >> $HOME/.bashrc
```

/ Install CUDA Toolkit 12.8

To download and install CUDA Toolkit 12.8, run the following commands:

```
$ wget https://developer.download.nvidia.com/compute/cuda/12.8.0/local_installers/cuda_12.8.0_515.43.04_linux.run
```

```
$ bash cuda_12.8.0_515.43.04_linux.run --silent --nox11 --override --no-man-page --toolkit --toolkitpath={INSTALLATIONPATH}/cuda/v12.8
```

```
$ echo "export CUDA_PATH_V12_8={INSTALLATIONPATH}/cuda/v12.8" >> $HOME/.bashrc
```

```
$ source $HOME/.bashrc
```

Where **{INSTALLATIONPATH}** is the desired installation path.

/ Build Tools Setup (Linux)

Download and install **Miniconda 3** or **Miniforge 3** and enable the required **make**, **cmake**, and **gxx_linux-64** packages.

1. Download Miniconda 3 or Miniforge 3.
2. Navigate to the folder where you downloaded the installer, open a terminal and execute it as in the example below, and continue the installation.

```
bash -Miniforge3-Linux-x86_64.sh
```

3. Activate the environment, as in the example below.

```
source ~/miniforge3/bin/activate
```

4. Install the three required packages by entering the following commands:

```
conda install -c anaconda make
```

```
conda install -c anaconda cmake
```

```
conda install -c anaconda "gxx_linux-64>=11,<12".
```

The conda prompt now contains a minimum set of tools required to build Rocky modules.

/ (Optional) Custom path for Solver Modules

You can create the environment variable **ROCKY_PLUGINS_PATH** if you want Rocky to find modules in a folder different than the default one (default path: **\$HOME/.Rocky/Modules**).

/ Additional Package: ROCKY_ADDITIONAL_PACKAGE_2025R2

- **Ansys Rocky Integrations (Integrations Folder):** This folder contains installation modules for integrating Rocky with other products.
 - **Ansys Motion (Ansys Motion Folder):** Ansys Motion installation module.
 - **Adams (Adams Folder):** Adams installation module.
 - **Ansys Minerva (Minerva Folder):** Minerva installation module.
- **PrePost Scripts Repository (Scripts Folder):** This folder contains some examples of Pre- and post-processing scripts, implemented by the Ansys Rocky team, which can be fully automated with a Python-based PrePost Scripting.
 - **Script: Chain Courtain Generation (chain_generator.zip):** This script creates a chain courtain using Rocky's custom fiber model.
 - **Script: Equivalent Network Circuit (equivalent_network_circuit.zip):** This script computes a steady-state flow on a contact network between a potential difference, which is specified using custom geometries bounded by a user process shape.
 - **Script: Set up a Basic Rocky Project (setup_basic_rocky_project.zip):** This example script provides an introduction to PrePost Scripting from the point of view of setting up a basic Rocky project.
 - **Script: Export Boundary Properties (export_boundary_properties.zip):** This script will, for the selected time range, export the selected properties for the selected boundary. The user can select between writing one single file containing all outputs or one file per output.
 - **Script: Export Particle Time Selection Properties (export_particle_time_selection_properties.zip):** This script will

export the selected properties of all particles within the selected Particle Time Selection process.

- **Script: Export Particle Beds and Geometries as Binary STL Files (export_particle_and_boundary_to_stl.zip):** This script exports several binary stl files representing the Particle Bed and Geometries that are active in the simulation at a particular output time.
 - **Script: Export Eulerian Statistics Properties (export_eulerian_statistics_properties.zip):** For cases that have been post-processed to include a User Process Cube or Cylinder that has been divided into Eulerian Statistics, this script will, for the desired time range, export the selected variables of a selected Eulerian Statistics process.
 - **Script: Export Particle Properties (export_particle_properties.zip):** This script will, for the selected time range, export the selected variables of all particles within the computational domain. The user can select between writing one single file containing all outputs or one file per output.
- **Material Wizard and Calibration Suite (Material Calibration Folder):** The Material Wizard is a complete application that runs inside Rocky and enables the user to set the material properties faster and easier. On the other hand, Calibration Suite makes it easy to determine real-life input parameters. It is a collection of simulation projects that incorporate an automatic post-processing script that uses advanced techniques to perform a high-level analysis of data.
 - **Material Storage (Material Calibration Folder):** Material Storage is a complete application that runs within Rocky and enables you to access and interact with a material repository.
 - **Coating Visibility Wizard (Coating Visibility Wizard Folder):** The Coating Visibility Wizard is a complete application that runs inside Rocky and enables you to quickly and easily define the nozzle parameters for your coating applications.
 - **Job Schedulers Scripts (Job Schedulers Scripts Folder):** Job schedulers provide additional benefits, including workload management, load balancing, and resource monitoring.

/ Ansys Rocky Integrations

Ansys Rocky 2025 R2 can be coupled with several **Ansys 2025 R2** products:

- Ansys Fluent
- Ansys Mechanical
- Ansys Motion
- Ansys optiSLang
- Ansys Minerva
- Ansys DesignXplorer
- Ansys Workbench
- Ansys EnSight
- Ansys System Coupling
- Ansys Discovery
- Ansys SpaceClaim

The **Ansys Rocky 2025 R2** can also be coupled with Adams.

/ Installation: Ansys Motion

Ansys Motion Coupling enables users to prescribe specific, complex motions for their equipment in Ansys Motion software and then have those same motions be used in a DEM simulation that is carried out by Rocky software.

Introduction

The Ansys Motion Coupling installation process is designed for the following scenarios:

- *Ansys Motion* users who are creating FMU files to share with Rocky users.
- *Rocky* users who are receiving FMU files from Ansys Motion users.
- Users who have both Ansys Motion and Rocky software will therefore do both parts of the coupling process on the same machine.

Setup Requirements

The setup requirements for Ansys Motion Coupling are similar to the requirements for the Rocky and Ansys Motion programs, with the following caveats:

- Coupling is currently supported only on Windows machines.
- The coupling is currently supported in Ansys **2025 R2** version for both Ansys Rocky and Ansys Motion software.

Important: The Ansys Motion program must be installed in the same installation folder as your other Ansys products.

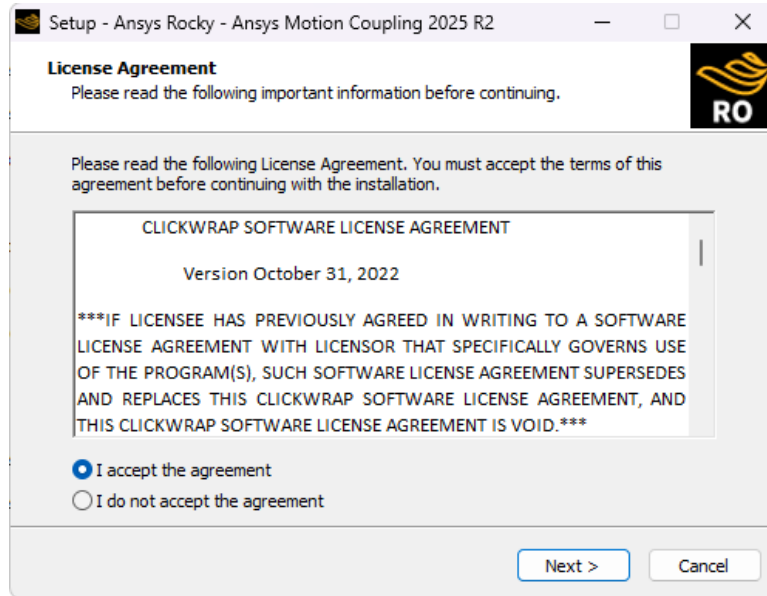
Installation Instructions

There are **two ways** of proceeding with this installation: automatically (via automated installer) or manually (via external Ansys Motion Coupling installer).

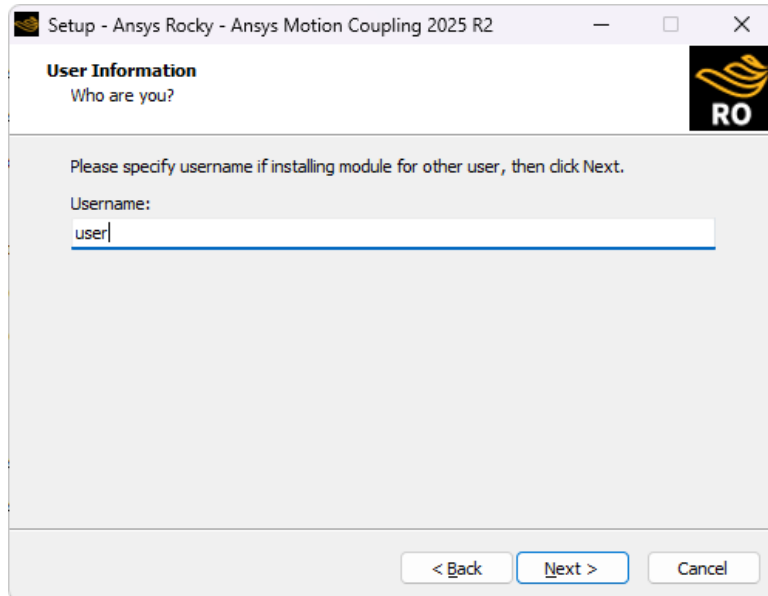
- **Automatic installation:** when installing Ansys Rocky *together with* or *after* Ansys Motion via automated installer, the Ansys Motion Coupling will be automatically installed (this method does not work for installing Ansys Motion *after* Ansys Rocky).
- **Manual installation:** when installing Ansys Motion *after* Ansys Rocky, the Ansys Motion Coupling needs to be installed via a separate setup wizard.

To install the Ansys Motion Coupling manually (in case you installed Ansys Motion *after* Ansys Rocky), open the **Ansys Motion Coupling Installer (Ansys Motion Folder)**, and then follow the installation instructions provided in the setup wizard. Specifically:

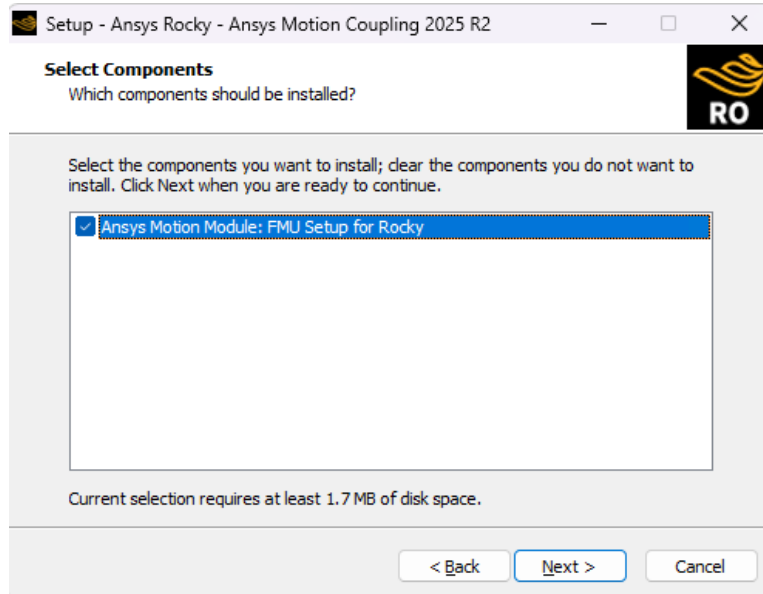
1.
 - a. From the **License Agreement screen**, read the Ansys End User License Agreement, click the **I accept the agreement** option, and then click **Next**.



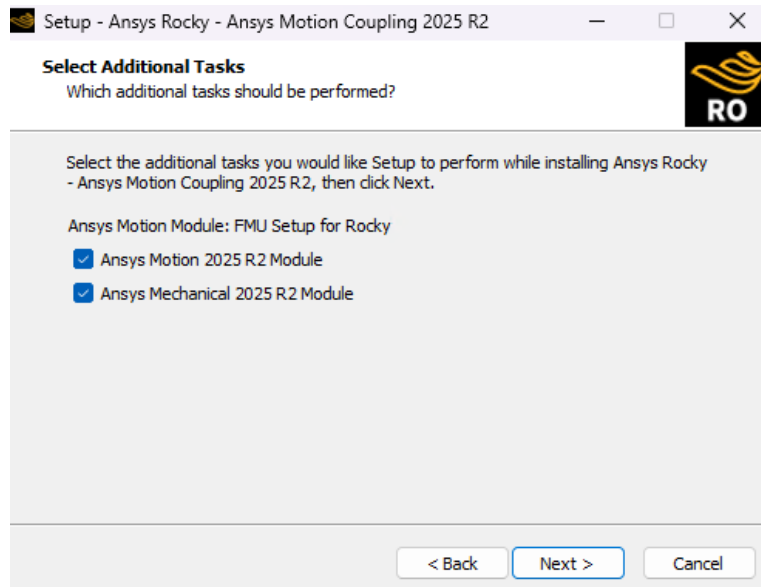
- b. From the **User Information** screen, do one of the following:
 - a. If you are installing this module for someone else, enter the name of the Windows User for which Ansys Motion Coupling will be installed.
Tip: This step should only be necessary if you are logged in as an administrator and the user for which you are installing the module does not have the proper installation permissions.
 - b. If you are installing this module for yourself, you can leave this field blank.



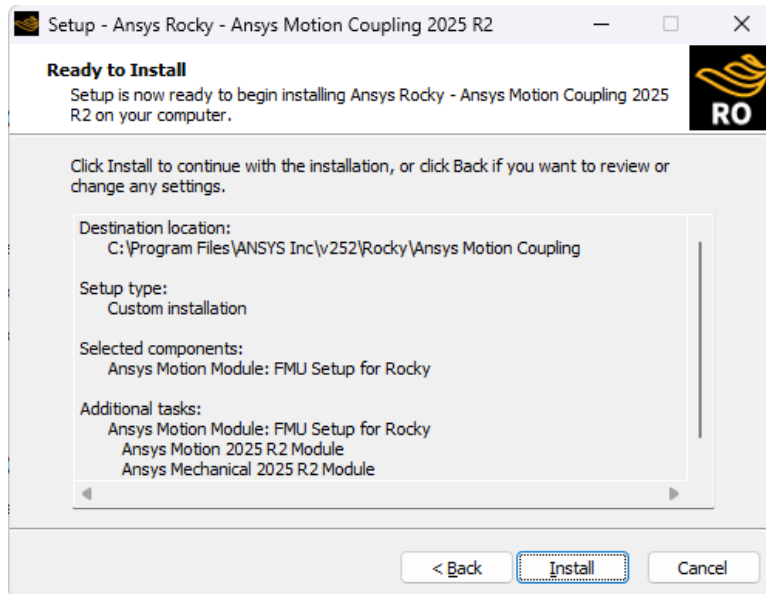
- a. From the **Select Components** screen, select the following and then click **Next**.
 - If you plan to use Ansys Motion, ensure the second option for **Ansys Motion Module: FMU Setup for Rocky** is selected.



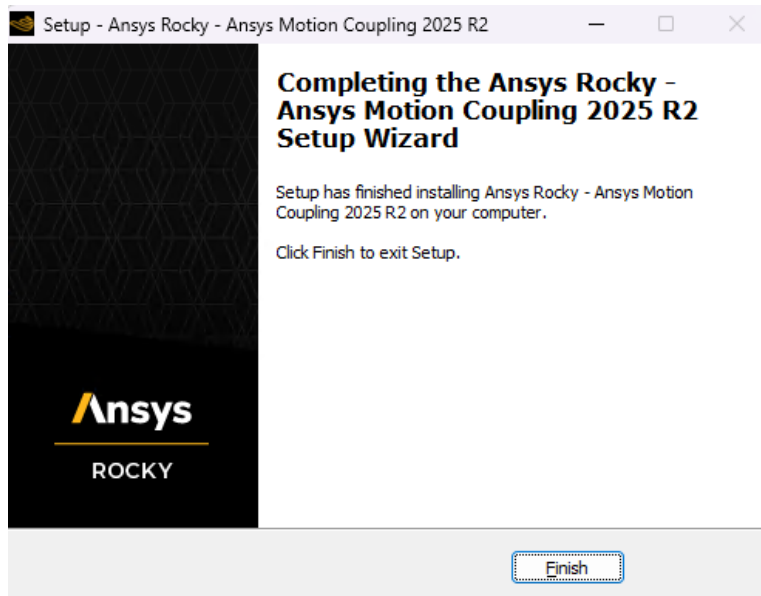
2. From the **Select Additional Tasks** screen, both Ansys Motion and Ansys Mechanical modules are selected to install. If you want to install only one of them, clear the checkbox from the other. If you want to install both modules, just click **Next**.



3. From the **Ready to Install** screen, review the installation tasks shown, and then click **Install**.



4. From the **Completing the Ansys Motion Coupling 2025 R2 Setup Wizard** screen, click **Finish** to close the wizard.

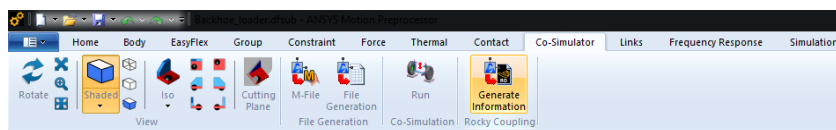


Your system should now be ready to create Ansys Motion files coupled with Ansys Rocky.

Verifying Proper Installation

You will know that Ansys Motion Coupling was installed correctly if you see the following indicator:

- In Ansys Motion, you have a **Generate Information Rocky Coupling** button on the **Co-Simulator** tab. (From the **Ansys Motion Preprocessor** program, view a **Subsystem**, and then select the **Co-Simulator** tab.)



Troubleshooting

If you do not see the indicator listed above, try one or more of the following solutions:

- **Verify** that your **Ansys Motion** product is installed in the same location as your other Ansys products.
- If it isn't installed in this location, uninstall Ansys Motion and re-install it in the correct location.
- **Tip:** The Ansys Motion installer automatically installs to your C: drive, thus if your other Ansys products are installed elsewhere, you can use the ANSYSMotion.msi directly to indicate a custom install option.

/Installation: Adams

Adams Coupling enables users to prescribe specific, complex motions for their equipment in Adams and then have those same motions be used in a DEM simulation that is carried out by Rocky.

Introduction

The Adams Coupling installation process is designed for the following scenarios:

- *Adams* users who are creating FMU files to share with Rocky users.
- *Rocky* users who are receiving FMU files from Adams users.
Important: To run the coupled simulation the user is required to have a valid license of Adams and have it installed on the same computer as Rocky.
- Users who have both Adams and Rocky will therefore do both parts of the coupling process on the same machine.

Setup Requirements

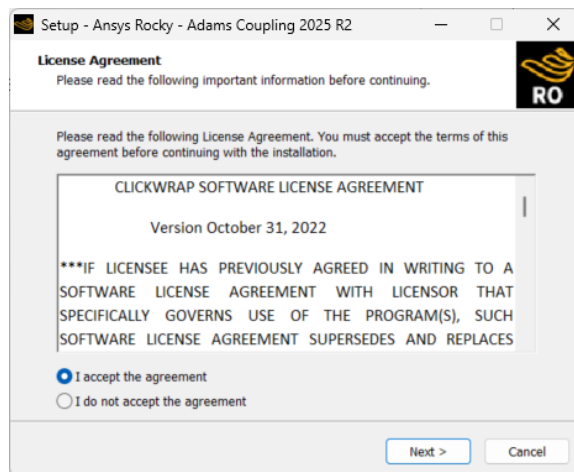
The setup requirements for Adams Coupling are similar to the requirements for the Rocky and Adams programs, with the following caveats:

- Coupling is currently supported only on Windows machines.

- For Adams users, the coupling is currently supported in Adams **2023.4.1** and **2024.1.1**.
- For Adams users to complete the FMU exportation process, it is necessary to have installed **Build Tools for Visual Studio 2022 LTSC 17.6.17** (a Microsoft account is required).

Installation Instructions

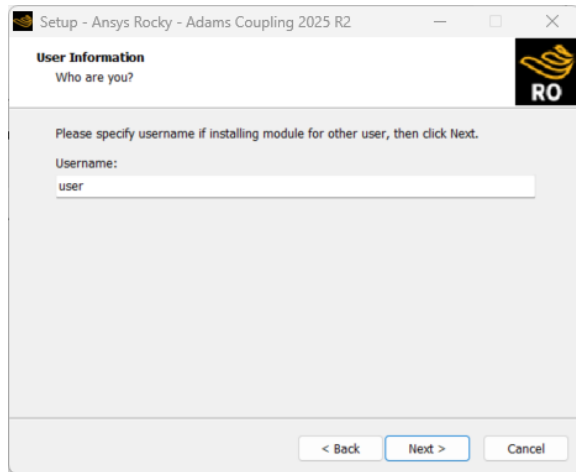
1. If you are using Rocky, ensure you have downloaded and installed **Rocky 2025 R2**.
2. Ensure you have downloaded and installed a supported version of Adams.
3. Open the **Adams Coupling Installer (Adams Folder)** and then follow the installation instructions provided in the setup wizard. Specifically:
 - a. From the **License Agreement screen**, read the Ansys End User License Agreement, click the **I accept the agreement** option, and then click **Next**.



- b. From the **User Information** screen, do one of the following:
 - If you are installing this module for someone else, enter the **Username** of the Windows user for which Adams Coupling will be installed.

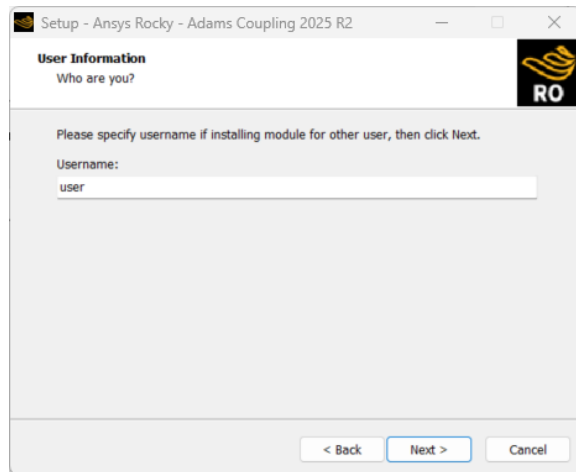
Tip: This step should only be necessary if you are logged in as an administrator and the user for which you are installing the module does not have the proper installation permissions.

- If you are installing this module for yourself, you can leave this field blank.



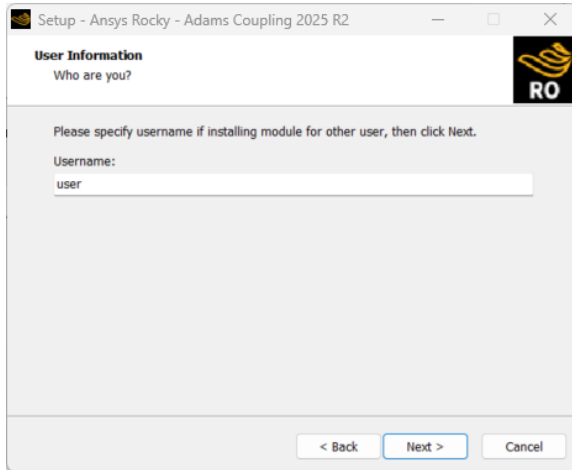
From the **Select Components** screen, do the following, and then click **Next**.

- If you plan to use Adams, ensure the second option for **Adams Module: FMU Setup for Rocky** is selected.

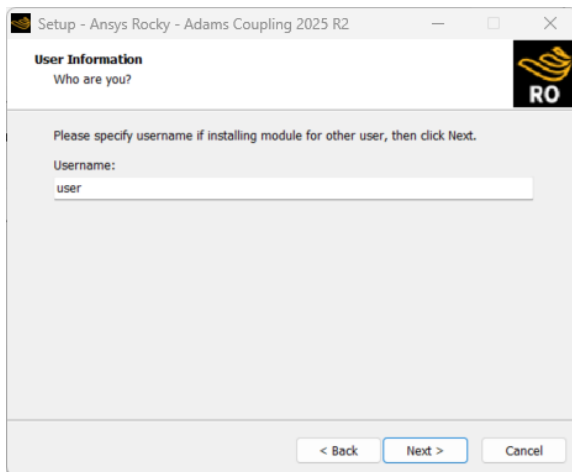


- c. From the **Ready to Install** screen, review the installation tasks shown,

and then click **Install**.



- d. From the **Completing the Ansys Adams Coupling 2025 R2 Setup Wizard** screen, click **Finish** to close the wizard.

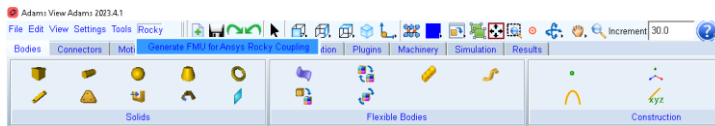


Your system should now be ready to create Adams files coupled with Ansys Rocky.

Verifying Proper Installation

You will know that Adams Coupling was installed correctly if you see the following indicator:

- In Adams View, you have a **Rocky Coupling Plugin** option in **Tools > Plugin Manager** tab (you can choose to activate it for the open project only or at every startup). After enabling the plugin, a **Rocky** tab should be available to proceed with the coupling process.



Troubleshooting

If after installation Adams is unable to export the FMU files, try one or more of the following solutions:

- Verify if you have installed **Build Tools for Visual Studio 2022 LTSC 17.6.17**.
- Make sure that the following **Environment Variables** exist in Windows. If not, create them:
 - **VS160COMNTOOLS** - C:\Program Files (x86)\Microsoft Visual Studio\2019\BuildTools\Common7\Tools
 - **VS2019INSTALLDIR** - C:\Program Files (x86)\Microsoft Visual Studio\2019\BuildTools**Important:** The path can change if the user changed the default install location.

/Installation: Ansys Minerva

Why Integrate Ansys Minerva and Rocky

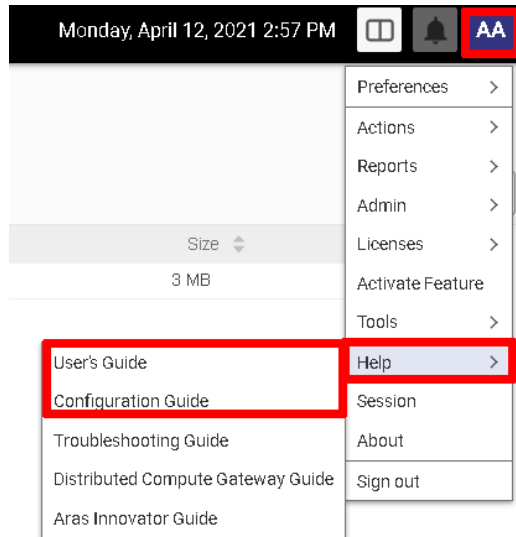
The capabilities of *Ansys Minerva* include the ability to create tasks, simulation projects and simulation programs. Because Ansys Minerva is available for both on-premise and cloud deployment, it connects team members regardless of geography and functional silos. Integrating **Rocky** with Minerva makes Rocky available inside this environment.

Purpose of this guide

In this Guide, we will show you how to:

1. Configure **Ansys Minerva** to extract Metadata from a project.
2. Configure a local launcher for **Rocky** inside **Ansys Minerva** to edit **Rocky projects** directly from the database.
3. Configure a batch launcher to submit simulation jobs to HPC Clusters.

Important: This guide covers only how to configure the **batch launcher** for HPC clusters, and not the cluster itself. Ensure that Distributed Compute Gateway (DCG) is configured, and that Rocky is already installed on your cluster. Refer to Ansys Minerva documentation section 7.3. *Integrating ANSYS Minerva with Distributed Compute Gateway (DCG)* of the **Distributed Compute Gateway** for configuration details.



Prerequisites

This guide was written assuming that you have one or more of the following machine types supported by the the **Ansys Minerva** workflow:

1. Server machine: a **Windows Server 2019** machine that has **Ansys Minerva 2021 R1** installed. This machine also must have installed so that **Ansys Minerva** can launch **Rocky** to extract metadata from projects.

2. User machine: **Windows 10** or **Windows Server 2019** machine that has **Rocky 2021 R1** installed. From this machine, the end user will launch **Rocky** to edit projects and/or run simulations.
3. HPC cluster: a **Linux Centos 7**, **Windows 10** or **Windows Server 2019** machine that has **Rocky 2021 R1** installed. This machine has high processing capabilities used for large simulations.

Commented [SV1]: @Janaina Oliveira check the version

Commented [SV2]: @Janaina Oliveira check the version

Important: It is critical that you refer to the **Ansys Minerva Configuration Guide** and the **Ansys Minerva User's Guide** when reading this integration guide as several steps require specific **Ansys Minerva** configuration steps. (From the **Ansys Minerva** program, click the user icon in the top right corner of the application header, and then select **Help > User's Guide** or **Help > Configuration Guide**.)

Limitations

1. This guide covers the integration of **Rocky 2021 R1** and **Ansys Minerva 2021 R1** only.
2. All the **Rocky** projects used inside **Ansys Minerva** must be **.rocky_archive** projects. (Refer to the *Archive a Simulation Project* topic in the *Rocky User Manual* for details on creating this kind of file.)

Commented [SV3]: @Janaina Oliveira check the version

Commented [SV4]: @Janaina Oliveira check the version

Setting up Rocky for integration with Minerva

For Ansys Minerva to be able to launch Rocky, we need to set up two environment variables. These variables need to be set for all machines in the Minerva workflow (described in Prerequisites) that contain the **Rocky** program.

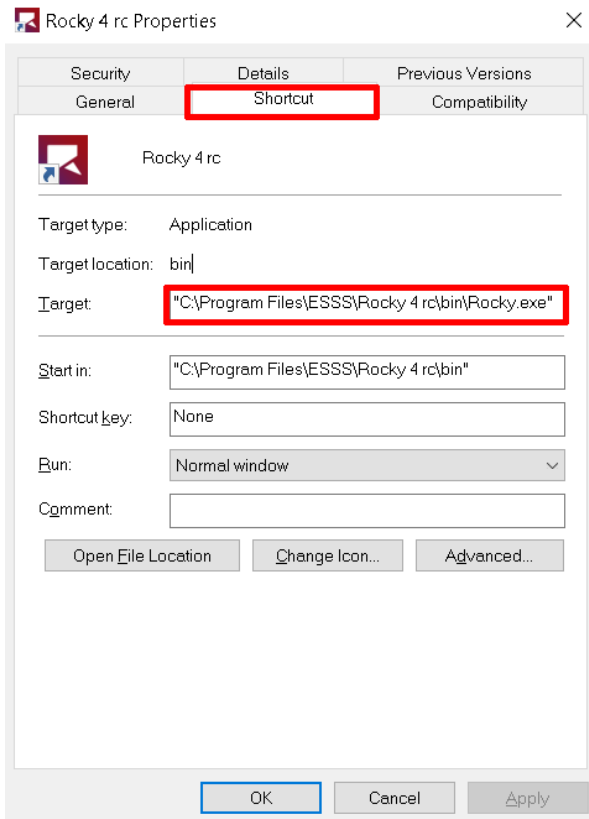
ROCKY_DIR

This variable is used by the integration scripts to find the Rocky executable file.

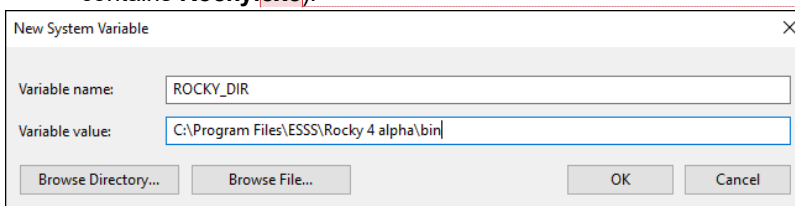
To find Rocky's installation path:

1. From your machine's desktop, right-click the **Rocky** icon, and then click **Properties**.

2. From the **Rocky Properties** dialog, select the **Shortcut** tab, and then copy the folder path (which ends in **bin**) from the **Target** field:2. From the **Rocky Properties** dialog, select the **Shortcut** tab and check the **Target**:



3. Create a new environment variable called **ROCKY_DIR** and then define the value as the **bin** folder path you copied earlier. (This is the folder that contains **Rocky.exe**):



Commented [SV5]: @Janaina Oliveira Rocky 4 alpha???

Setting up Minerva for integration with Rocky

Configure metadata extraction

In this section, we will show you how to configure **Ansys Minerva** for **Metadata Extraction** of **.rocky_archive** files.

Set the application settings file and Integration Scripts

There are three files that are needed by the **Metadata Extraction Service** within **Ansys Minerva** to execute metadata extraction of **.rocky_archive** files. These three files are inside the **minerva_integration_files** folder you previously extracted:

- **MDSERVICE.config.json** This is the application settings file.
- **python-extract-metadata.py** This is the python script for extracting metadata.
- **rocky-API-extract-metadata.py** This is the PrePost Scripting script for extracting metadata.

First, we will set the **application settings file**.

Note: For the purposes of this guide, we provided the **MDSERVICE.config.json** file already configured for **Rocky**.

Tip: To learn how to set up your own application settings file for **Custom Metadata Extraction**, refer to sections 3.6. *Specifying Custom Metadata Extraction Settings* and 3.7. *Extracting Metadata from Custom File Types* of the **Configuration Guide**.

1. Navigate to your **%ANSYSMINERVAROOT%** folder (typically **C:/Program Files/Ansys Inc/Minerva**), create the **config** folder (if it does not already exist), and then copy the **MDSERVICE.config.json** file to that folder.

Now, we will place the scripts that handle the metadata extraction and generate the **report.xml** and the **metadata.xml** that are used by **Ansys Minerva** to display the extracted information about the project.

2. Navigate to **%ANSYSMINERVAROOT% / tools** folder and create a new folder called **Rocky**.
3. Place the **python-extract-metadata.py** and the **rocky-API-extract-metadata.py** files inside the **Rocky** folder you just created.

After you complete these steps, you should have the following files:

- **%ANSYSMINERVAROOT% / tools / Rocky / python-extract-metadata.py**
- **%ANSYSMINERVAROOT% / tools / Rocky / rocky-API-extract-**

- metadata.py
- %ANSYSMINERVAROOT% / config / MDService.config.json

Ansyes Minerva Configuration for Metadata Extraction

By default, **Ansyes Minerva** will not recognize the **.rocky_archive** file type and so we must configure Minerva to support these files. To do that, we have to define the **Custom File Type** for **.rocky_archive** files, add this Custom File Type to the **MDConverter Rule** and then configure how **Ansyes Minerva** handles the **Extracted Metadata Mapping**.

1. Use section 4.6. *Adding a Custom File Type* of the **Configuration Guide** to add a new **Custom File Type**.
2. From the **FileType** tab, define the **Name**, **Ext.** and **Mime Type (Content Type)** fields as shown in the table:

Field	Value
Name	Rocky Archive
Extension	rocky_archive
Mime Type (Content Type)	application/rocky-custom

The screenshot shows the 'FileType 1' configuration window. At the top, there are buttons for 'Save', 'Done', and 'Delete'. Below this is a tab labeled 'FileType'. The configuration fields are as follows:

Name	Ext.	Mime Type (Content Type)
Rocky Archive	rocky_archive	application/rocky-custom

Below these fields, there is a 'Description' text area, a 'Rule Type' dropdown menu (set to 'MDConverter'), and a 'Priority' input field (set to '1').

3. Use section 3.7 - *Extracting Metadata from Custom File Types* of the **Ansyes Minerva Configuration Guide** to add the **Rocky Archive** File Type to the **MDConverter Rule**.
4. Use section 3.8 - *Mapping Extracted Metadata to Properties* of the **Configuration Guide** to configure how Ansys Minerva displays the extracted metadata. During this step, **Ansyes Minerva** will ask for an

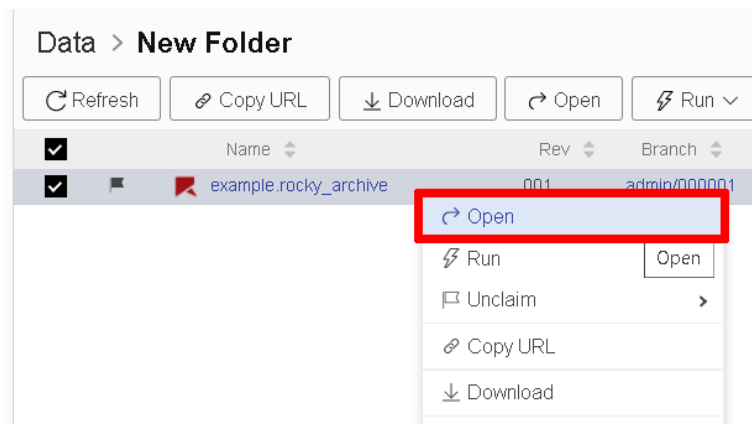
image; it is suggested that you select the %ROCKY_DIR% / .. / all / rocky_logo.png image for this step.

5. Restart the **Ansys Metadata Service**.

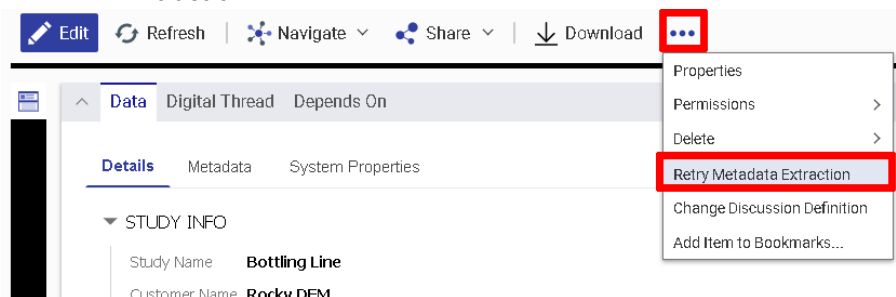
Metadata Extraction of an Example

For demonstration purposes, the **example.rocky_archive** file was provided inside the **minerva_integration_files** folder you previously extracted. We will use this file to illustrate the Rocky + Minerva integration in action.

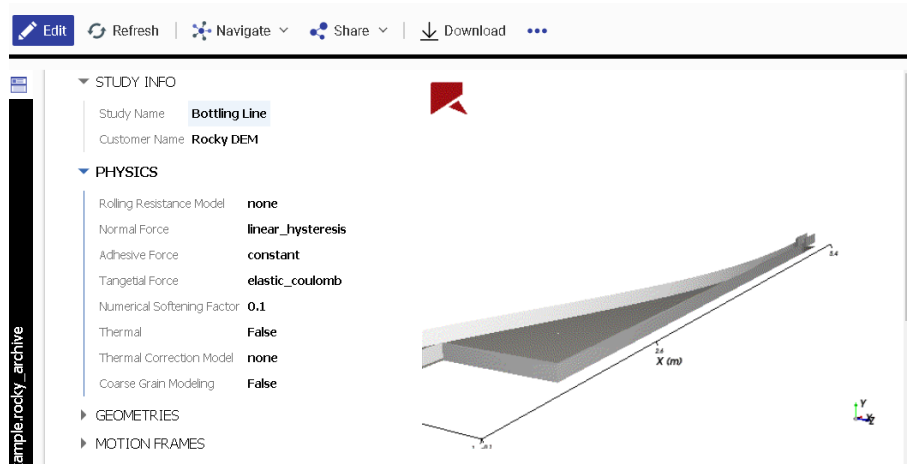
1. Upload the file to **Ansys Minerva**.
2. Right click **example.rocky_archive** and click **Open** as shown below:



3. **Ansys Minerva** will automatically extract the metadata from the **example.rocky_archive** file. If you do not see the extracted metadata, right-click the **More** button, and then click **Retry Metadata Extraction**:



After the extraction, the metadata is displayed:



Configure local launcher

Ansys Minerva can launch local applications for custom file types like the **.rocky_archive** you just configured. By doing that, the end user can edit files/projects directly from the **Ansys Minerva** database and **Ansys Minerva** handles the download and upload steps of the file/project from and to the database.

In this section, we are going to configure **Ansys Minerva** to launch **Rocky** to edit the **example.rocky_archive** we uploaded in the previous section.

Install the Application Launcher

For **Ansys Minerva** to be able to launch **External Applications**, the **Application Launcher** must be installed:

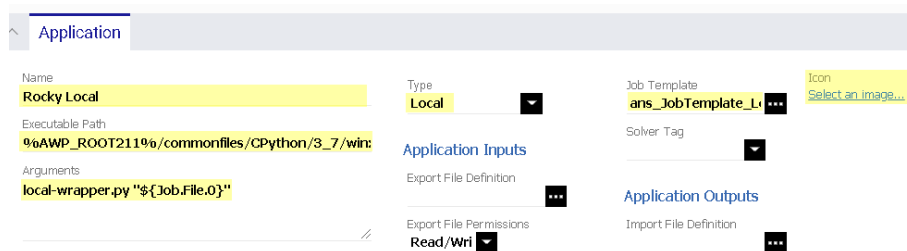
1. Use section 6.1. *Installing the ANSYS Application Launcher* of Minerva's **User's Guide** to install the **Application Launcher**.

Configure a new Application

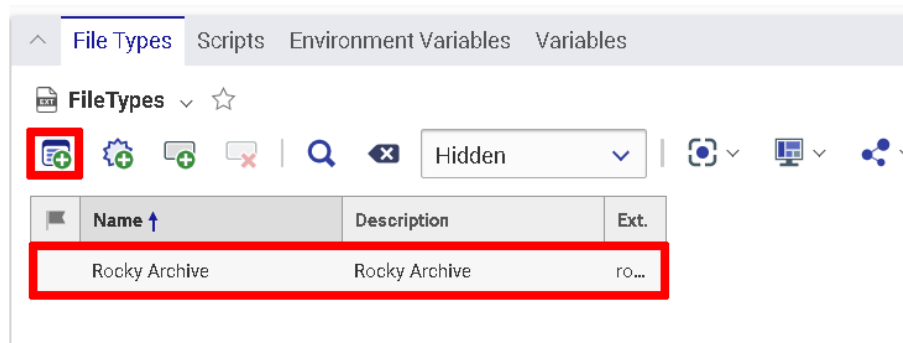
1. Use section 9.1. *Defining Applications* of the **Configuration Guide** to configure a new **Application** definition in **Ansys Minerva**.
2. From the **Application** tab, click **Edit** and then configure the fields as

shown in the table:

Name	Rocky Local
Executable Path	%AWP_ROOT211%/commonfiles/CPython/3_7/win64/Release/python/pythonw
Type	Local
Job Template	ans_JobTemplate_Local
Icon	<i>Click in Select an image... and upload %ROCKY_DIR% / .. / all / rocky_logo.png</i>
Arguments	job-wrapper.py "\${Job.File.0}" "local"



3. From the the **FileTypes** sub-tab, add a new entry and then select the **Rocky Archive** file type previously set in the Configure metadata extraction section:

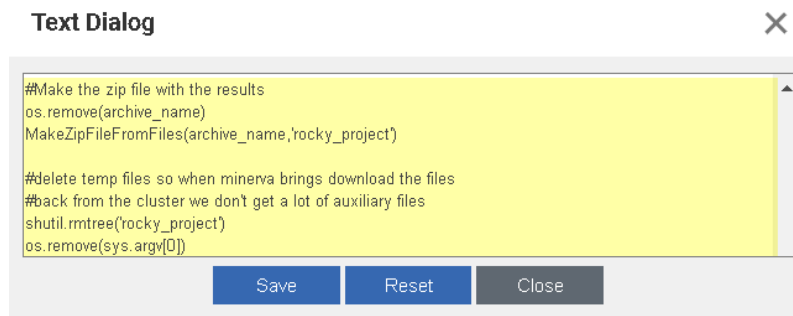
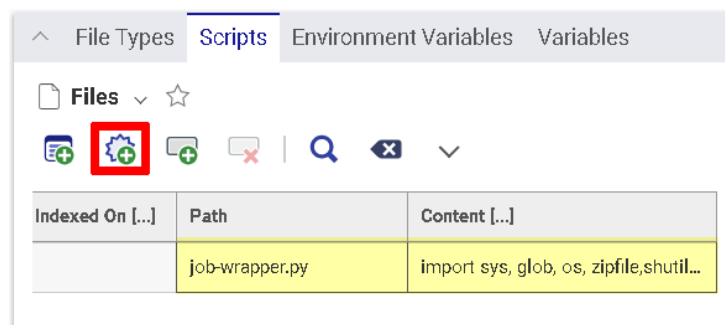


4. Open the **job-wrapper.py** file provided inside the **minerva_integration_files** folder you previously extracted, and then

copy its content to the clipboard.

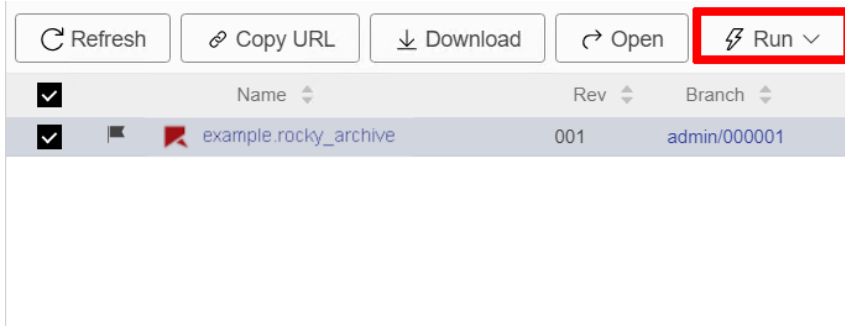
5. From the **Scripts** sub-tab, add a new entry and then set the **Path** and the **Content [...]** fields as shown in the table and images below:

Field	Value
Path	job-wrapper.py
Content [...]	<i>Paste the clipboard content from job-wrapper.py here</i>



Run the Application

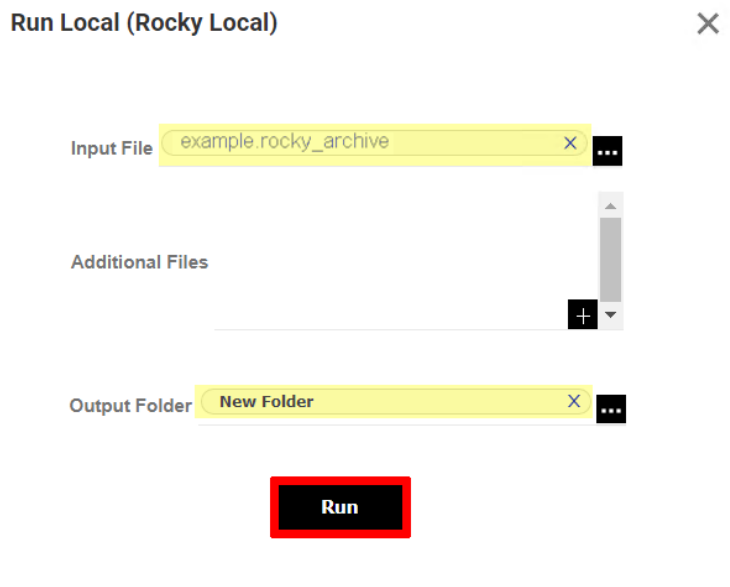
1. From the **Data** screen, select the **example.rocky_archive** file, click **Run**, and then click **Rocky Local**.



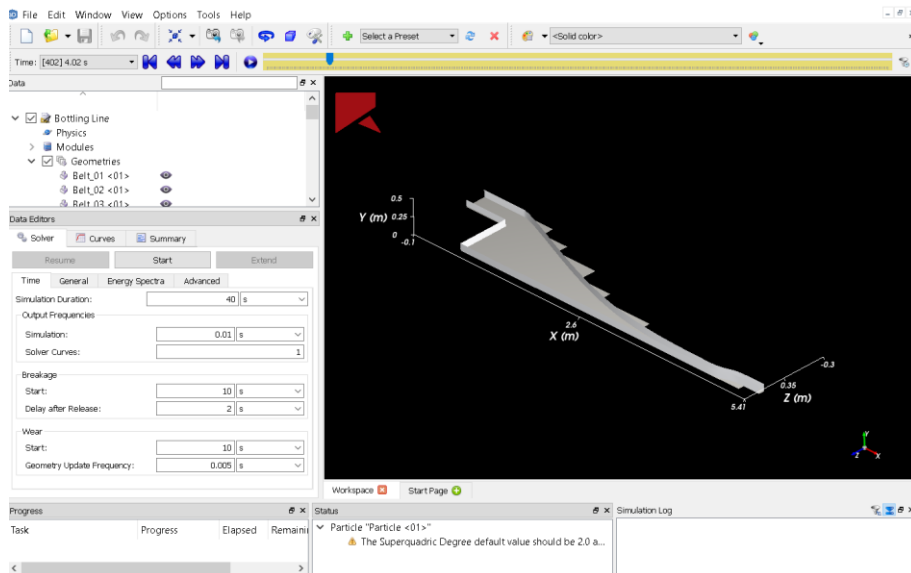
The local launcher dialog of **Rocky** will appear.

By default, when a project is modified and saved, **Ansys Minerva** replaces the database file when it uploads the modified project, since the default values of the **Input File** and **Output Folder** are the same as the original file name and folder, respectively.

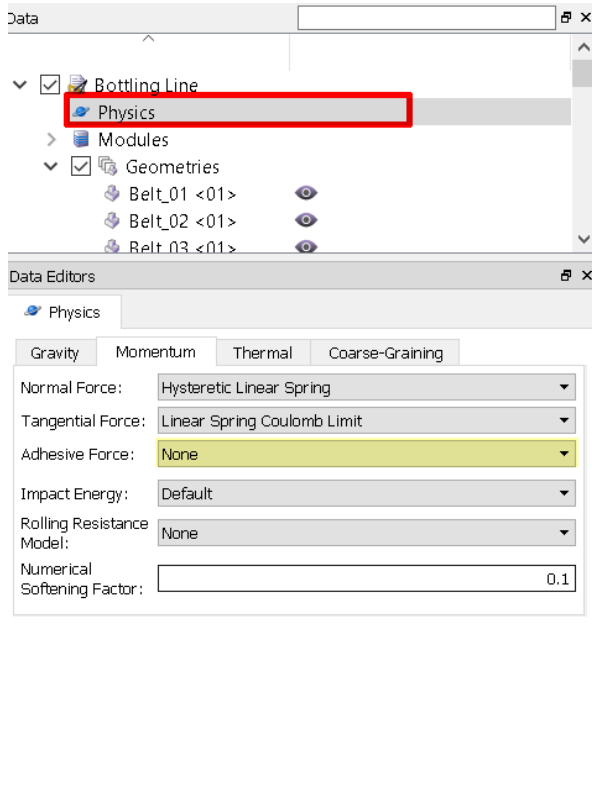
2. To open the project, click **Run**:



The **Rocky** program will open:



3. Edit the **Adhesive Force**, as shown:



1. Save your project by pressing **Ctrl + S** or clicking the save button.

Note: In this step you don't need to save the **Rocky** project as **.rocky_archive**; the script you configured previously in this section handles the conversion for you.

6. Close **Rocky**.

Once you close it, **Ansys Minerva** will upload your changes back to the database.

Configure batch launcher (for HPC cluster)

Ansys Minerva can also submit batch simulations in HPC (High Performance Computing) clusters. By doing that, the user can run a simulation of a previously set up Rocky project and choose the desired number of resources to use for the simulation.

In this section, we are going to configure **Ansys Minerva** to launch a batch simulation using the **example.rocky_archive** we edited in the previous section.

Create a Job Template

First, a **Job Template** will be created for the batch application. For the local application (created earlier in the section Local Launcher) it was not necessary since the job template for local applications of **Ansys Minerva** was used. Now, a custom job template containing the **Number of Cores** and the option of using **GPU** will be created:

1. Use section 8.2. *Create a Job Template Item Type* of the **Configuration Guide** to create an **Item Type**, and then set the **Name** as **ans_JobTemplate_Rocky**:

The screenshot shows the 'Item Type' configuration window. The 'Name' field is set to 'ans_JobTemplate_Rocky'. The 'Singular Label' is 'Rocky Job Template' and the 'Plural Label' is also 'Rocky Job Template'. The 'Versioning' section is expanded, showing 'Versionable' as an unchecked checkbox, 'Discipline' as 'Automatic', and 'Revisions' as a dropdown menu.

2. Use section 8.2. *Create a Job Template Item Type* of the **Configuration Guide** to add **Properties** entries that define which parameters will be used by the job template. The entries we are going to add are shown in the tables below:

Name	Label	Data Type	Data Source [...]	Default Value	Pattern
application	Application	Item	ans_Application		
name	Name	String			
rsm_queue	RSM Queue	Item	ans_JobServerQueue		
task	Task	Item	Ans_SimulationTask		
cores	Number of Cores	Integer		1	
gpus	Number of GPUs	Integer		0	
rocky_archive	Rocky Archive	Item	Ans_Data		.*_archive\$

use_gpu	Use GPUs	Boolean		0	
---------	----------	---------	--	---	--

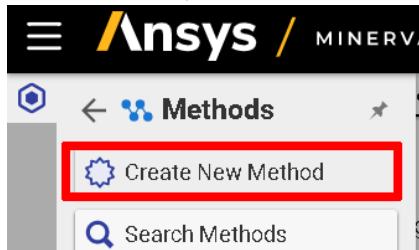
Create Server Methods

In this subsection, the **Methods** that are called during a **.rocky_archive** job submission will be created. Further information on the subject is presented in section 8.4. *Create a Server Method*. For **Rocky**, we already have the necessary methods provided in the **minerva_integration_files** folder you previously extracted:

1. ans_Job_SubmitRocky
2. ans_RockyJob_OnFormLoad
3. ans_RockyJob_OnSubmit

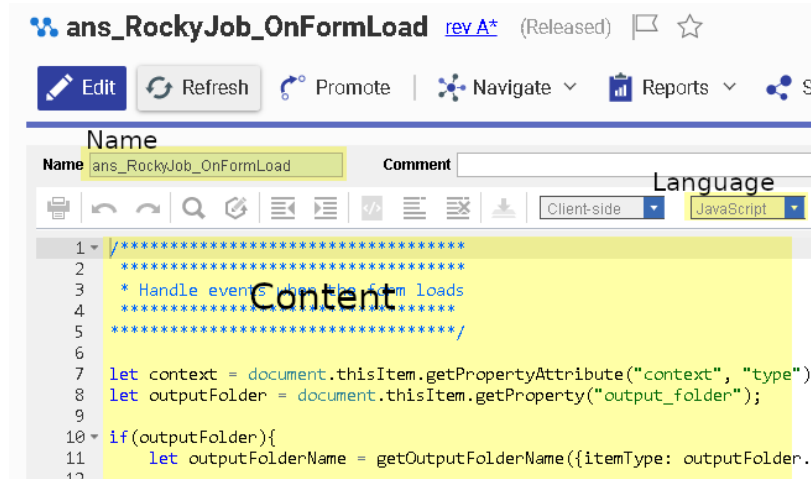
For each of these files, a new method inside **Ansys Minerva** will be created:

1. Open the **Navigation Panel**, then navigate through **Administration > Methods**, and then click **Create New Method**:



2. For each method, copy the **content** from the provided file, and then set the **Name** and the **Language** as shown below:

Name	Language
ans_Job_SubmitRocky	C Sharp
ans_RockyJob_OnFormLoad	Javascript
ans_RockyJob_OnSubmit	Javascript

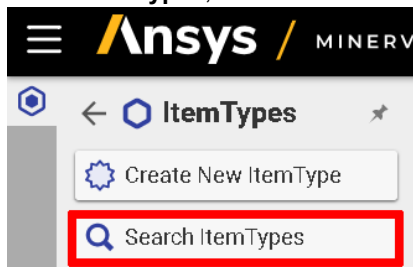


Define a Job Template Form / User Interface

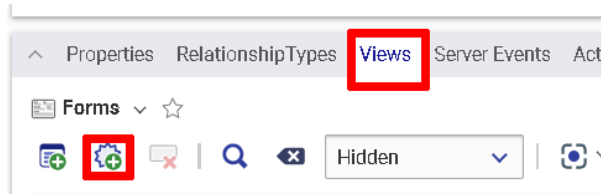
Create a new form

In this subsection, the **User Interface** of Rocky's job template is going to be defined.

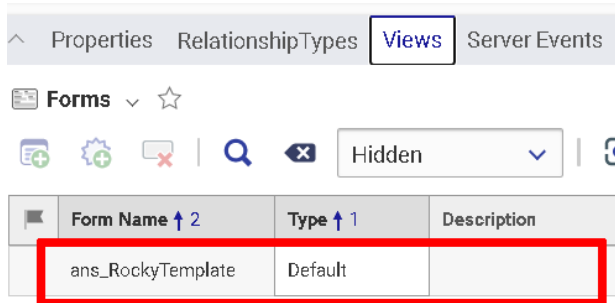
1. Open the **Navigation Panel**, navigate through **Administration** > **ItemTypes**, and then click **Search ItemTypes**:



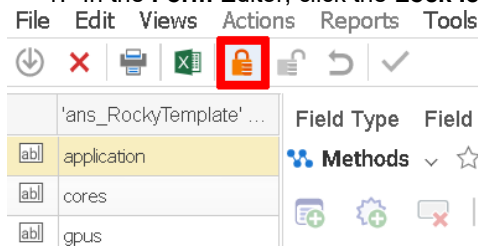
2. Search for the **Job Template** you previously created in subsection Job Template, and then open it.
3. Select the **Views** sub-tab and then create a new **Form**:



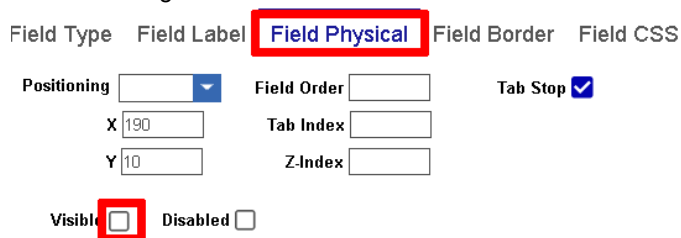
4. Name the form **ans_RockyTemplate** and then save it.
5. Double-click the newly created **Form** to edit it:



1. In the **Form Editor**, click the **Lock Icon**:



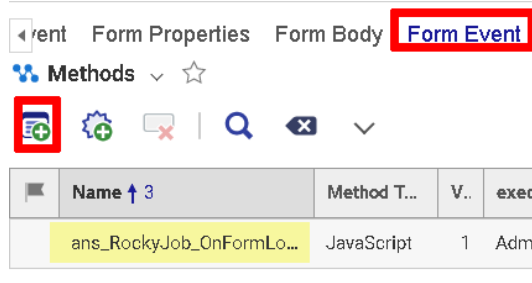
7. Hide the default fields by clicking the **Field Physical** tab and then clearing the **Visible** checkbox for each field:



Add "OnFormLoad" method

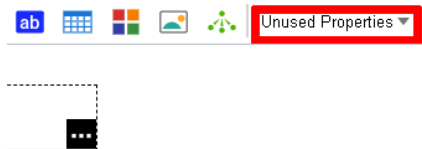
The next step is to add the method that will be launched when the **Form** is loaded.

1. Select the **Form Event** tab, click **Add Methods**, and then search for the **ans_RockyJob_OnFormLoad** method (previously created in subsection Methods).
The method will appear in the methods list as shown:



It is necessary to create a field that loads **.rocky_archive** projects for job submission.

2. To add the **rocky_archive** property to the form, from the **Unused Properties** drop-down list, select **rocky_archive**:



Define file properties control via HTML

The default control for file properties (like the control **.rocky_archive** field that is being created) does not allow the use of the standard file/folder view to browse for files. Instead of the default control, the usage of a HTML code is recommended.

1. Use section 8.3.2. *File Selection Control* of the **Configuration Guide** to set the file properties control properly.
2. After adding this HTML field, replace the occurrences of fileCase (as suggested in the **Configuration Guide**) by rockyArchive:

HTML Code

```
<div id="rockArchive" style="width:300px"></div>

<script>

require({ paths: { 'commons': './scripts/Ansys/commons' } },

['commons'],

function(SPDM) {

    SPDM.renderFileSelect({}, 'rockArchive');

});

</script>
```

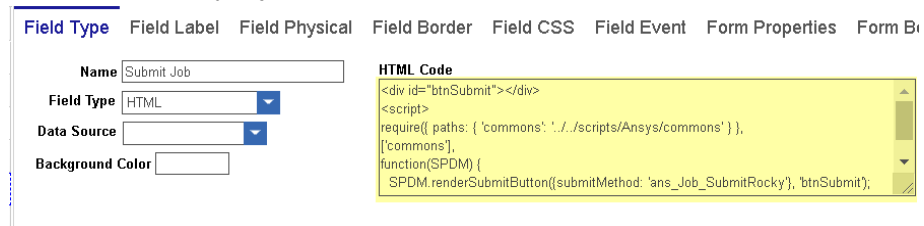
Create an HTML Submit button

The subsection 8.3.3. *Submit Button Control* provides the HTML code for a **Submit** button. In this guide, we are providing a HTML code based on that one, but modified to work with the methods created in subsection Methods.

1. Create a new HTML control by clicking **New HTML**:



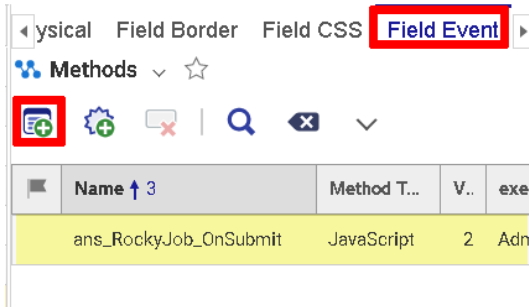
1. Open the **submit_button.html** provided in the **minerva_integration_files** you previously extracted, copy its content to the clipboard, and then paste it onto the **HTML Code** of the control you just created:



Configure Field Event for Submit button

The next step is to add the **ans_RockyJob_OnSubmit** method as a **Field Event** for the **Submit Button**.

1. From the **Field Event** tab, click **Add Methods** and search for **ans_RockyJob_OnSubmit**:



2. From the **Unused Properties**, add controls for **Cores**, **use_GPU** and **GPUs** (if you set it properly on the template, these controls will not need further configurations). The final template should be similar to what is presented below:

Rocky Archive

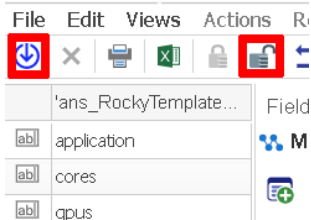
Cores

Use GPUs ☐ Number Of GPUS

Submit

Save form

1. After the template is configured, unlock and save the template by clicking the **Unlock Item** and the **Save Item** buttons:



Application Definition

It is necessary to create an **Application** definition the same way it was done in Local Launcher, but now it will be a **batch** application using the custom template created in the previous section.

1. Create a new **Application**.
2. For this new application, configure the fields as shown below:

Field	Value
Name	Rocky Batch
Executable Path	\${Job.PythonExecutable}
Type	Batch
Job Template	ans_JobTemplate_Rocky
Icon	Click Select an image... and then upload %ROCKY_DIR% / .. / all / rocky_logo.png
Arguments	job-wrapper.py "\${Job.File.0}" "batch" "\${Job.Cores}" "\${Job.UseGpu}" "\${Job.Gpus}"

Application

Name

Rocky Local

Executable Path

%AWP_ROOT211%\commonfiles\CPython\3_7\win...

Arguments

local-wrapper.py "\${Job.File.0}"

Type

Local

Job Template

ans_JobTemplate_L...

Icon

Select an image...

Solver Tag

Application Inputs

Export File Definition

Application Outputs

Import File Definition

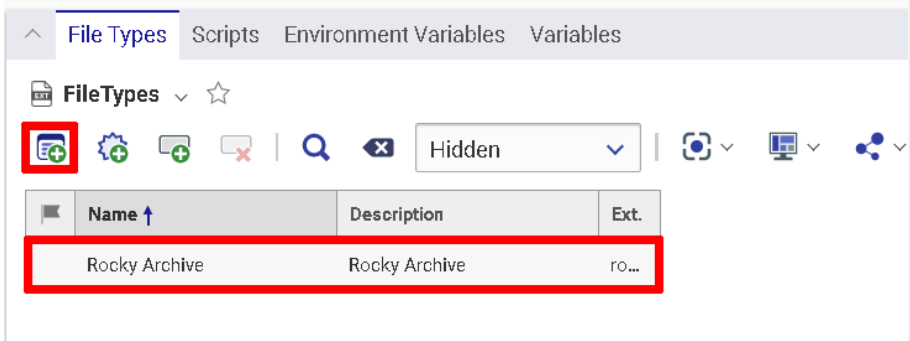
Export File Permissions

Read/Wri

It is also required that you add the **File Type** and a **Script** to this application.

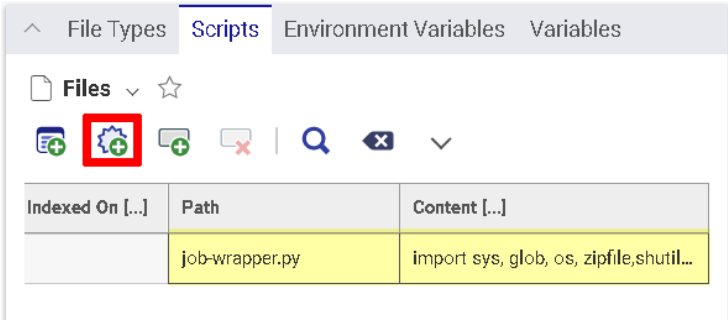
3. From the **FileTypes** sub-tab, add a new entry and then select the **Rocky Archive** file type you set in the sub-section Ansys Minerva Configuration

for Metadata Extraction:



4. Open the **job-wrapper.py** file provided inside the **minerva_integration_files** you previously extracted, and then copy its content to the clipboard.
5. From the **Scripts** sub-tab, add a new entry, and then set the **Path** and the **Content [...]** fields as shown below:

Field	Value
Path	job-wrapper.py
Content [...]	Paste the clipboard content from <i>job-wrapper.py</i> here



Text Dialog
X

```

#Make the zip file with the results
os.remove(archive_name)
MakeZipFileFromFiles(archive_name,'rocky_project')

#delete temp files so when minerva brings download the files
#back from the cluster we don't get a lot of auxiliary files
shutil.rmtree('rocky_project')
os.remove(sys.argv[0])

```

Save
Reset
Close

Run a simulation

Now, let's run the simulation:

- From the **Data** screen, select the **example.rocky_archive** you edited in previous sections, and then click **Run > Rocky Batch**.

Refresh
Copy URL
Download
Open
Run

	Name	Rev	Branch
☒	example.rocky_archive	001	admin/000001

- From the job submitting form, click **Submit**:

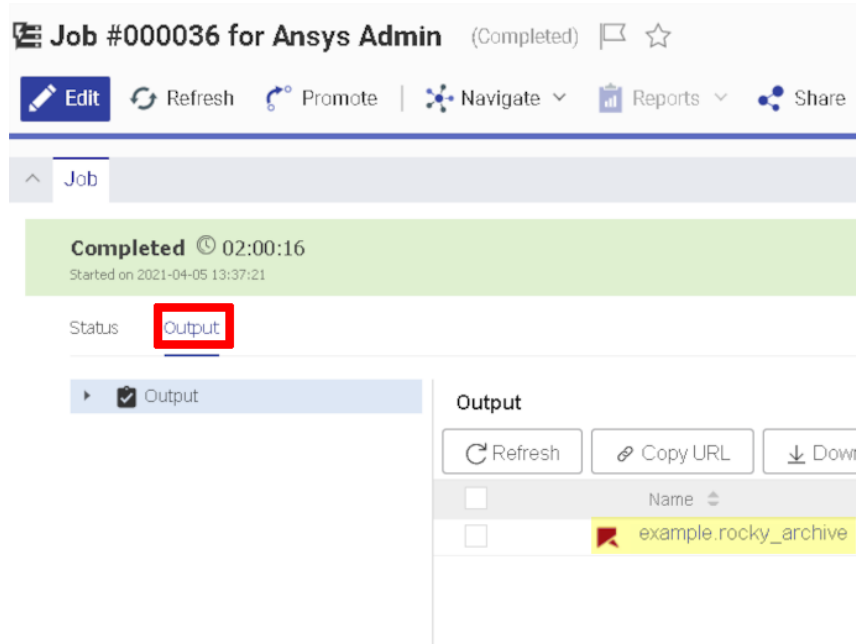
Run Batch (Rocky)
X

Rocky Archive
Cores
1
Use GPUs
Number Of GPUS
0
Submit

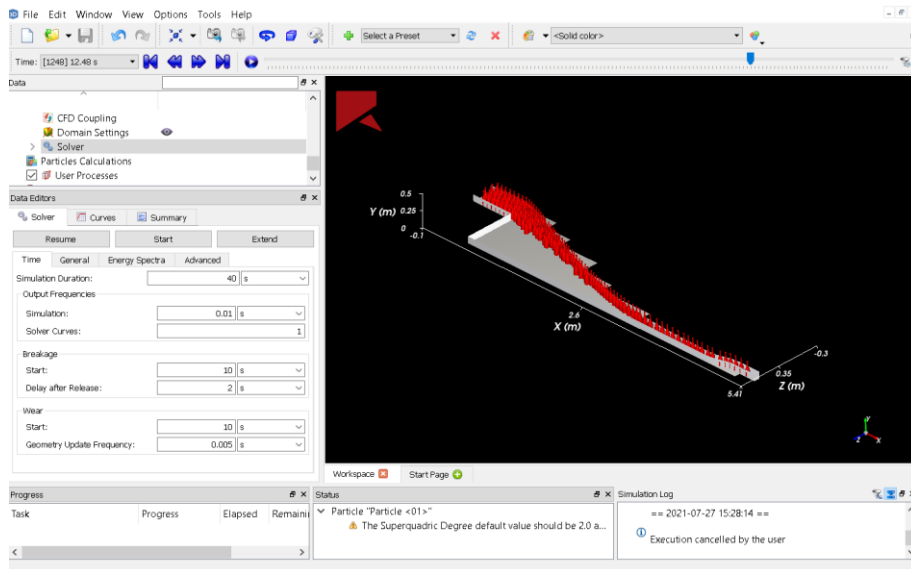
After the job is submitted, a new screen appears showing the simulation progress:

The screenshot displays the Ansys Admin interface for a specific job. At the top, the header reads "Job #000102 for Ansys Admin" with a "(Saved)" status and icons for a flag and a star. Below the header is a toolbar with buttons for "Edit", "Refresh", "Promote", "Navigate" (with a dropdown arrow), and "Reports" (with a dropdown arrow). A tab labeled "Job" is selected. Below the tab, a light blue banner shows "Saved" with a clock icon, a timer at "00:00:27", and the text "Started on 2021-04-13 15:17:40". Underneath, there are two sub-tabs: "Status" (selected) and "Output". The "Status" sub-tab contains a vertical list of five items, each with a clock icon: "Downloading Files", "Waiting for Resources", "Running Job", "Uploading Files", and "Organizing Files".

When simulation processing is done, it is possible to check the output files in the **Output** sub-tab:



3. Open the output file to check its results in Rocky:



/Installation: Ansys optiSLang

Improved integration between **Ansys optiSLang 2025 R2** and **Ansys Rocky** can boost your iterative design processes. By combining your Rocky cases with the robust design optimization (RDO) methods of optiSLang, processes requiring many repeat runs—such as those for material calibration—get faster and more efficient. And when combined with the powerful parametric modeling capabilities of Ansys Workbench, you can take these efficiencies even farther.

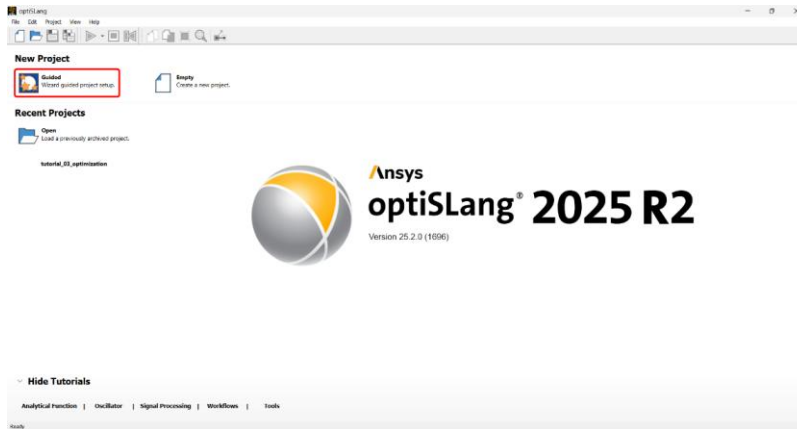
Using the optiSLang plug-in, the input parameter selection and variation for your Rocky projects are automatic based upon sensitivity analyses and meta-modeling techniques. These automatic methods reduce the number of iterative cases that need to be run and increase the confidence with which you can adjust the interaction parameters for your calibration projects.

Integration with Ansys Rocky 2025 R2

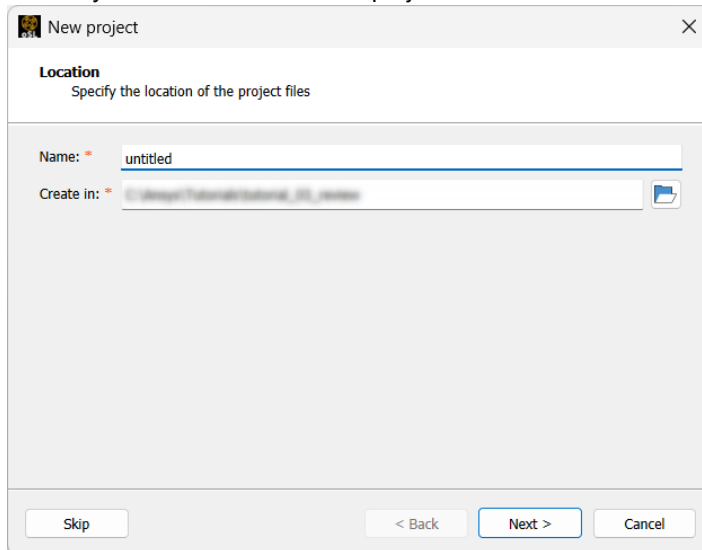
Important: Before you begin, ensure that the Rocky project you want to integrate with optiSLang has already been processed in Rocky and has simulation results. And if you only have one license of Rocky, ensure that your Rocky program is closed as optiSLang will need to open it as part of the integration process.

It is possible to integrate Ansys optiSLang 2025 R2 with Ansys Rocky 2025 R2, as show below:

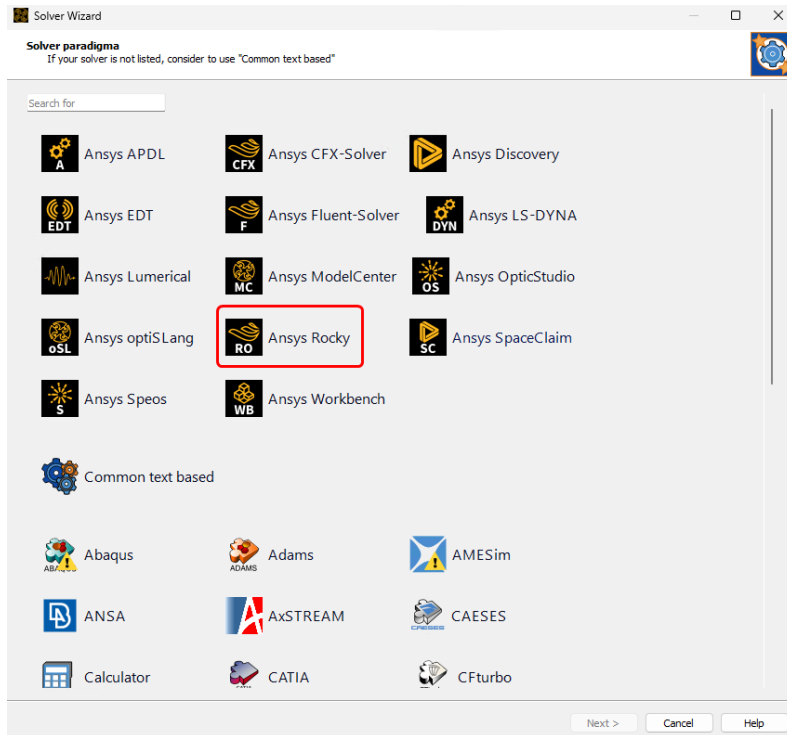
1. Open the **Ansys optiSLang 2025 R2** program.
2. From the main **optiSLang** screen, under **New Project**, click **Guided** (as shown) to open the Solver Wizard.



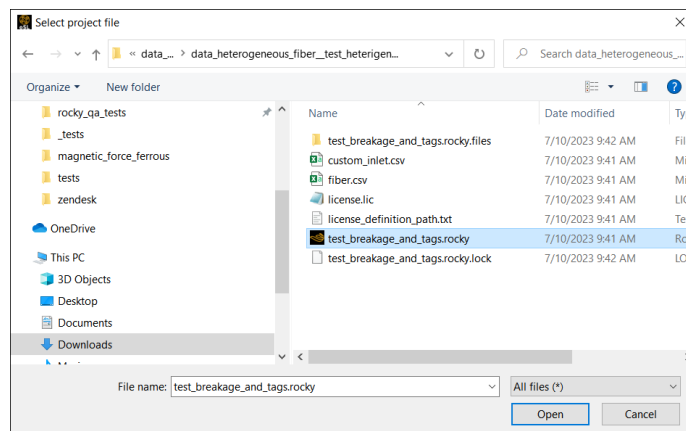
3. A **New Project** window opens. Name your project and select the directory where you wish to save the project. Click **Next** and then **Finish**.



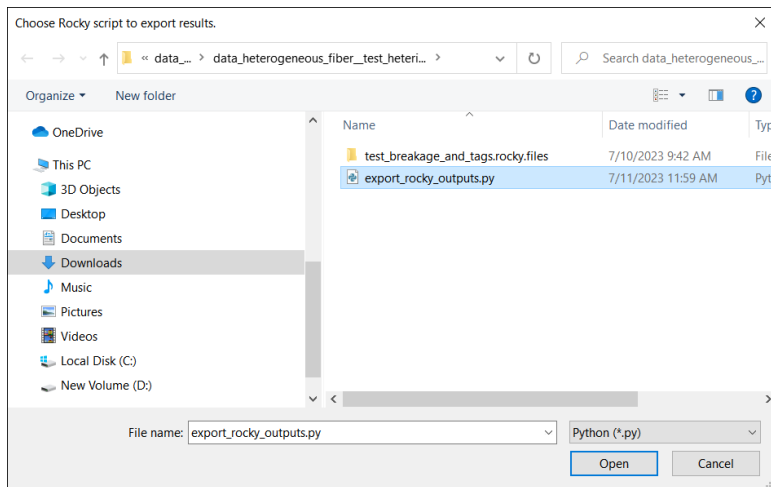
4. From the **Solver Wizard**, search for **Rocky**, and then under **Interfaces**, select the **Ansys Rocky** option (as shown).



- From the **Select project file** dialog, navigate to and then select the already-processed Rocky project file you want to use, and then click **Open** (as shown).



- From the **Choose Rocky script to export results** dialog, select the script (.py) file you want optiSLang to use for the exported output variables, and then click **Open** (as shown).



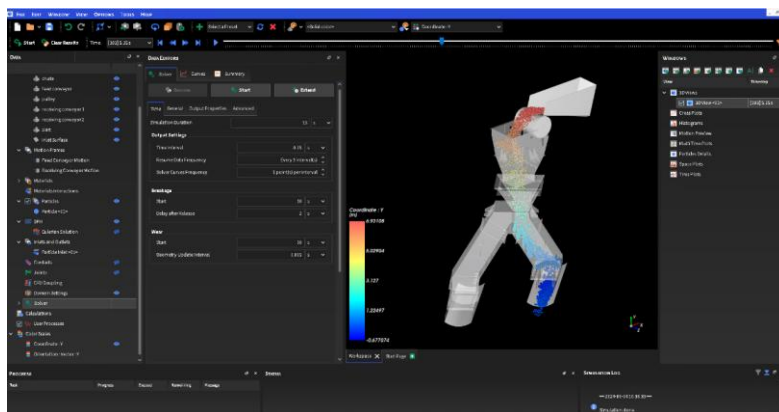
Note: By default, optiSLang creates .py files for this purpose in the same folder as the Rocky project you selected earlier. After optiSLang loads the input/output variables, Rocky will launch and you will receive a confirmation message.

Tip: For more information about using optiSLang, refer to the **Ansys optiSLang User Documentation**.

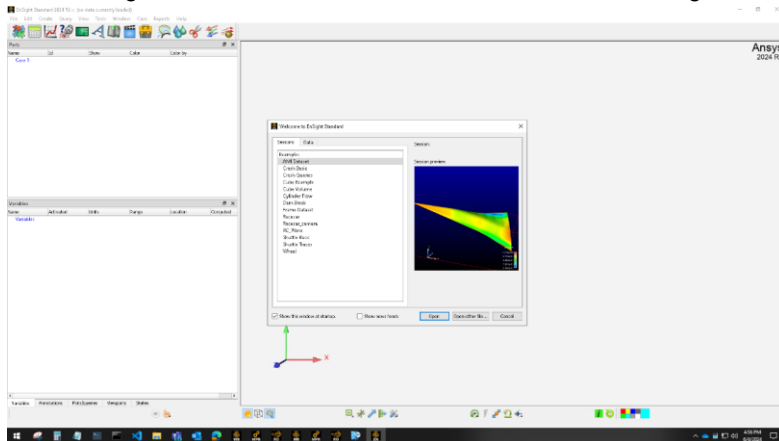
/Installation: Ansys EnSight

Rocky has its post-processing capabilities enhanced by coupling with EnSight. After running any case using the Rocky, with both DEM-SPH, you can open the .rocky file through the EnSight, and post-process using it all the environment, lights, and materials processing capabilities.

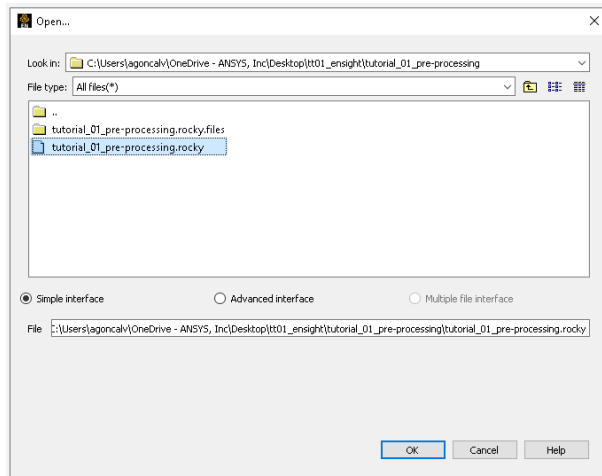
In the example below, the **Transfer Chute** case of our Rocky Tutorial 01 was normally solved in the Rocky.



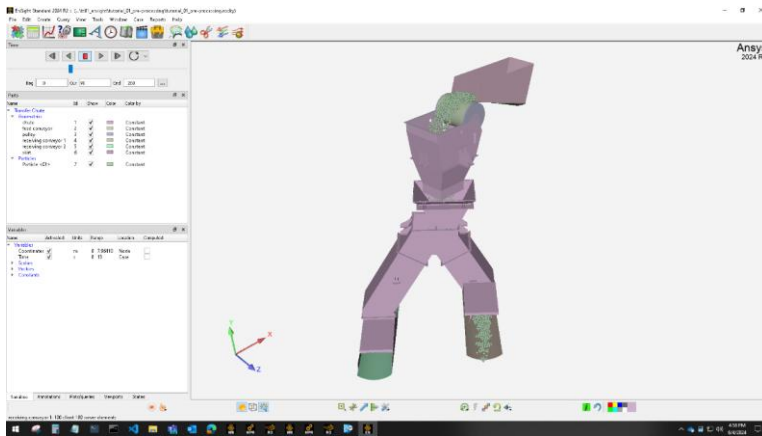
After finishing the simulation, the case was closed, and the EnSight was opened.



The file that must be selected through the EnSight it is the .rocky file, as shown below:



After that, the Transfer Chute case is normally opened by the EnSight, with all the results solved by the Rocky:



The Ansys Rocky and EnSight bring some interesting benefits, as the EnSight rendering capabilities, including lighting, environment, and material presets, as well as DEM-SPH representation enhancement.

For this Tutorial 01, the results using the EnSight capabilities are shown below:



/ Installation: Ansys System Coupling

Beyond the **1-Way Thermal** and **Structural** coupling with Mechanical, through Workbench, Rocky has the capability of **2-Way Coupling** using the **Ansys System Coupling**. For the 2-Way coupling cases, Rocky can send the Temperature results, from both DEM and/or SPH, and receive the Heat Flow from a geometry, due to its heat transfer solved by Mechanical, for example. The Temperature and Heat Rate data transfer process is through the System Coupling, which handles the send-back of the data for each timestep, configuring the 2-Way Thermal capability.

Also, Rocky can work with a 2-Way Structural case when the Forces that are generated by the DEM and/or the SPH collisions are sent, and the Incremental Displacement of the boundary, due to the particle collision Forces are calculated by Mechanical, for example, are sent back to Rocky, with recalculates the particle trajectory due to this new geometry configuration. The 2-Way Structural data transfer process is also handled by System Coupling.

Through the System Coupling, Rocky can couple with both Mechanical, Fluent, and any other Ansys software that returns **Incremental Displacement** and **Heat Rate** as outputs and receives **Force** and **Temperature** as inputs, which are exported by Rocky. A more detailed explanation of both 2-Way thermal and Structural coupling using Rocky and System coupling is available in the Rocky User Manual in the 2-Way Coupling with Mechanical section.

/ PrePost Scripts Repository

Rocky Scripts are a powerful tool in Ansys Rocky software, allowing users to automate and customize their simulation processes such as model configuration, data post-processing, and results analysis. Documentation containing technical and usability details for scripts created by Rocky Team Experts is in the **Rocky Scripts Manual** in [Ansys Help](#).

The scripts are created using the PrePost Scripting tool, for more information on how to create your own scripts please refer to the **PrePost Scripting Manual** on [Ansys Developer Portal](#).

/ Material Wizard

The Material Wizard is a complete application that runs inside Rocky and enables the user to set the material properties faster and easier.

It does so by taking what you already know about the bulk density, angle of repose, and cohesion of your real-world material, and translating that information into a set of ready-to-use Rocky parameters.

These initial parameters are a good starting place for your particle calibration work and can help you jump-start your DEM simulation project.

/ Material Storage

The Material Storage is a complete application that runs within Rocky and enables you to access and interact with a material repository.

The material repository can be the Ansys Granta or a local repository on your machine. The local repository is a database that the Material Storage can create and access.

The Material Storage was built to facilitate three common situations related to the calibration procedure:

- Get previously calibrated material and use it for a new setup
- Save a new calibrated material into the database for later use
- Remove specific information from the database that was saved incorrectly or is outdated

/ Calibration Suite

Rocky DEM's **Calibration Suite** makes it easy to determine real-life input parameters. This collection of simulation projects incorporates an automatic post-processing script that uses advanced techniques to perform a high-level analysis of data.

The **Calibration Suite** contains the following ready-made simulation projects:

- SAOR: Static Angle of Repose
- PAOR: Pour Angle of Repose
- DyAOR: Dynamic Angle of Repose
- DAOR: Drawdown Angle of Repose
- LBD: Loose Bulk Density
- IPT: Inclined Plane Test
- FT4: FT4 Rheometer
- RSC: Ring Shear Cell

/ Coating Visibility Wizard

The **Coating Visibility Wizard** is a complete application that runs inside Rocky and **enables you to quickly and easily define the nozzle parameters for your coating applications.**

The coating variability analysis is made through particle visibility, which is the percentage that a particle (or particle region) occupies within the camera view (representing the nozzle) in each time and is accounted by a pixel-by-pixel verification.

Wizard prerequisite

- This wizard must run only on **2025 R2** or later releases.

/ Job Schedulers Scripts

- Job schedulers provide additional benefits, including workload management, load balancing, and resource monitoring.
- The tests performed had a machine with several GPUs and a machine with N CPUs, being possible to use the systems of queues below:

Job Schedulers & Queuing Systems Tested*	Scripts
Altair PBS Pro (Linux)	• rocky.pbs
Slurm (Linux)	• rocky.slurm
Torque with Moab (Linux)	• rocky.moab
Microsoft 2016 HPC Server	• rocky_hpc.bat

***Note 1:** To run **Rocky on GPU** and **Fluent on CPU**, it is necessary to use the argument "--script [load_hosts.py](#)" in Rocky's command line, save the list of hosts in the project folder with the file name "hosts" and leave the [load_hosts.py](#) script in the same folder.

Commented [AG6]: Link to ESSS portal

- The script will read the hosts file, change the parameters within Rocky so that Fluent runs with the list of hosts created.

***Note 2:** Job schedulers do not support all versions of Linux and Windows. Confirm with the vendor that the job scheduler is compatible with your chosen operating system.

****Currently the Rocky DEM does not support parallel distribution on more than one machine, we are working to be supported in future versions.**

/ System Requirements

Please see below for minimum system requirements and optional recommendations in the System Requirement Page on Ansys ([Platform Support and Recommendations | Ansys](#))

Note: To use one or more graphics cards for processing your simulations (also known as "GPU or Multi-GPU processing"), your graphics card must meet the latest requirements listed in the previous link so that it is able to perform computational tasks.

/ Batch Mode Execution

If you need to perform Rocky tasks without using the Rocky UI, also known as running in batch mode (sometimes known as hidden mode or silent mode), it is possible to do so from the command line on both Windows and Linux-based machines. For more information, see the Rocky User manual (7.4. Appendix G: Perform Tasks from the Command Line or Journal Appendix).

The Ansys logo is positioned in the top-left corner of the image. It features a stylized yellow 'A' icon followed by the word 'Ansys' in a white, sans-serif font. The background of the entire image is composed of three main sections: a black area on the left, a large yellow diagonal band in the center, and a light gray area on the right.

Ansys