APRIL 2024

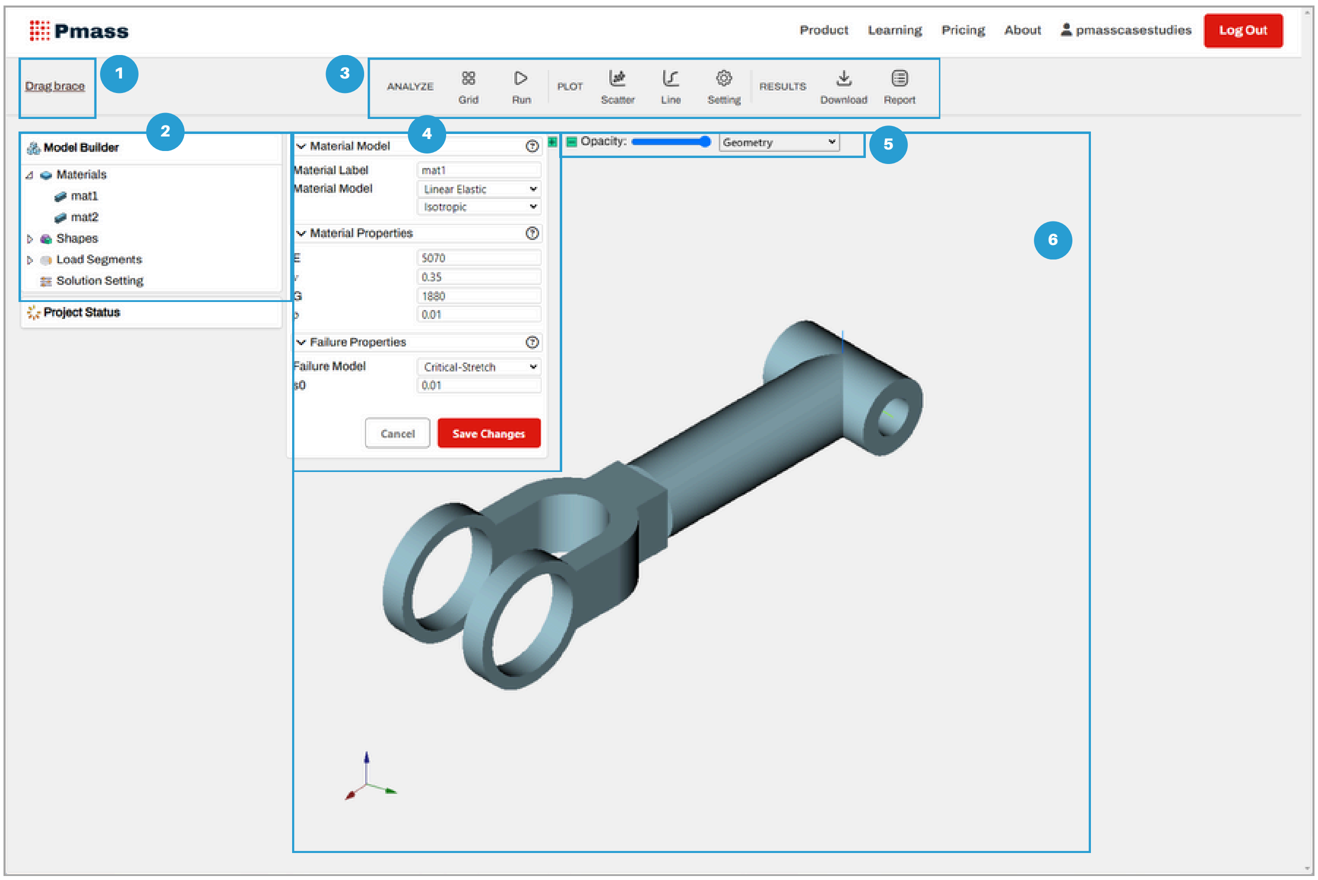
Pmass Manual

Workbench

www.pmass.io

Workbench

The **Workbench** is the primary component where you'll conduct your simulations. It includes the **Model Builder**, which is essential for creating geometries, defining materials, and setting up loads and simulations. Additionally, the Workbench encompasses both grid and simulation configuration tools, as well as an integrated online post-processor for analyzing results. This setup provides a comprehensive environment for managing and executing simulation projects effectively.



Here is a brief overview of the main interface components of the Workbench. The terms defined in this section will be consistently used throughout the documentation. This nomenclature is essential for understanding and navigating the Workbench effectively.

1. Project Info: Clicking on the project info link (specified by the project name) opens a pop-up window that displays essential details about the current project. This includes the project name, description, owner, type of analysis, and the solver used. This feature provides a quick and easy reference to key project parameters.

2. Model Builder: The Model Builder Tree is a critical component where you manage the inputs for your project. Here, you can add, edit, or remove materials, geometries (shapes), and load segments. Additionally, you can configure solution parameters directly from the Model Builder Tree. A detailed description of the Model Builder is provided on Page XX of this document. This tool is designed to streamline project setup and modifications efficiently.

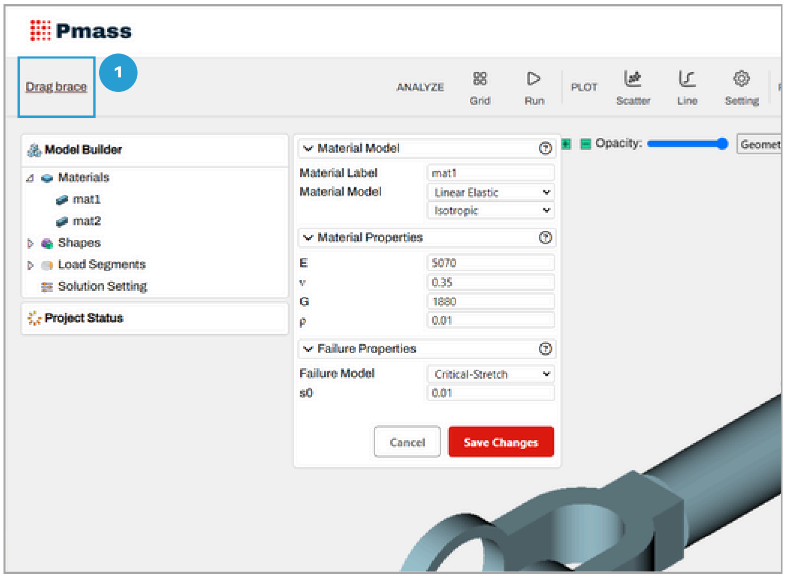
3. Controlbar: This component offers tools to perform analyses on the cloud, post-process results online, or download the results in text format. A detailed description of the control bar can be found on Page XX of this document. These features enhance flexibility and accessibility, allowing for efficient management and analysis of data.

4. Setting Panel: The Settings Panel is where you can modify the configuration of your model, including materials, shapes, load segments, and solution settings. Additionally, this panel allows you to set up the simulation or post-processing using specific settings. This centralized control hub streamlines adjustments and setups, enhancing the efficiency of model management and execution.

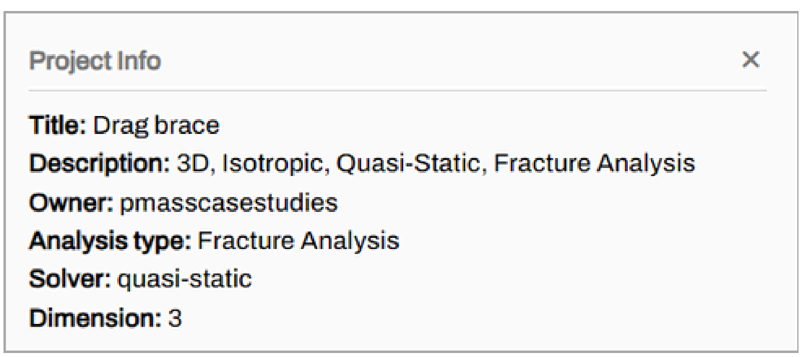
5. Viewbar Toolbar: The Viewbar toolbar features the primary controls for interacting with the viewer. This toolbar facilitates the manipulation of the visual display, allowing for adjustments to how models and results are viewed. A detailed description of the Viewbar is provided on page XX of this document, offering comprehensive guidance on its functionalities.

6. Viewer: The viewer is the central component of the interface, showcasing the entire scene of the simulation. It displays the original geometries, and other geometry primitives such as boundary conditions, no-failure zones, and output areas. Utilize the viewer to inspect your setup, scatter the undeformed grid, and post-process the simulation results. This tool provides a comprehensive visual overview, enabling detailed examination and analysis of various simulation stages.

Project Info

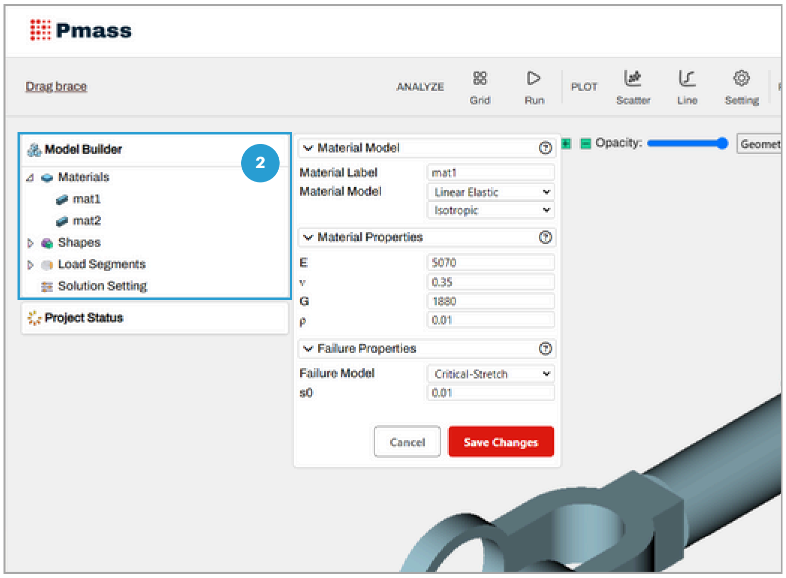


The **Project Info** is a readily accessible component that allows you to review the basic information set up during the creation of the project. Located in the top-left corner of the Workbench page, clicking on the project's name brings up a pop-up window. This window displays essential details about the current project, including the project name, description, owner, analysis type, and solver. This feature ensures that key project information is easily accessible at any stage of your work.



To close the Project Info and return to the dashboard, simply click on the close icon.

Model Builder



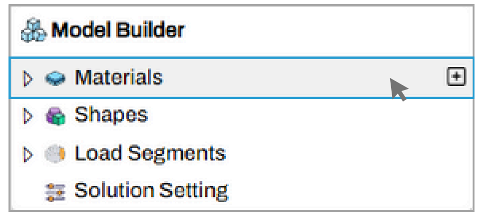
The **Model Builder** tree is a key component on the Workbench page, displaying a list of the project's materials, shapes, load segments, and solution parameters. Positioned on the left side of the Workbench page, the Model Builder allows for efficient organization and access to essential elements of your project. To hide or show the components within the Model Builder, you can click on the header box in the Model Builder component.

If the project is created by uploading a JSON file or by cloning an existing project, the project's inputs are automatically populated in the Model Builder tree. Conversely, if the project is created using the "Use wizard" option, the Model Builder tree will initially be empty, requiring you to manually add components to construct the model.

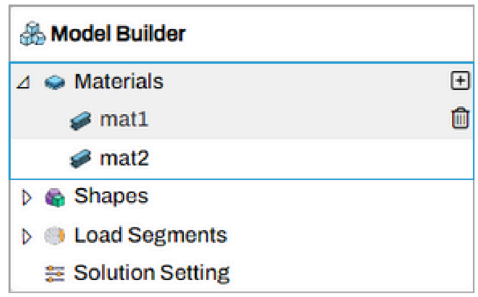
In the following pages, we will thoroughly explore the components of the Model Builder. This detailed examination will provide a deeper understanding of each element and how they can be effectively utilized in your projects.

Materials

Materials are a key component within the Model Builder where you can manage the materials for your project. To access and expand the material tree, simply click on the "**Materials**" box. This action will reveal the list of materials available and allow for further modifications and management of material properties essential for your project.



The Materials tree displays a list of all the materials used in the current project, each identified by a unique label. For instance, in this case, the project includes two materials, labeled as 'mat1' and 'mat2'.



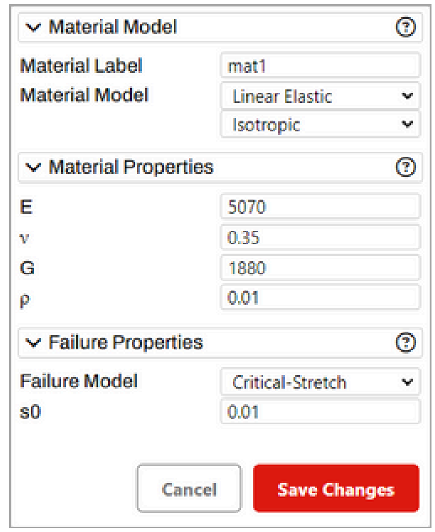
Add a new material

To add a new material to your project, click on the "**add**" icon located on the Material box. Doing so will introduce a default material to the Materials tree, which you can then customize according to the specific requirements of your project.

Delete a material

To delete a material, click on the delete icon located next to the selected material. You will be prompted to confirm the deletion to ensure that you want to remove the material. This verification step helps prevent accidental deletions and safeguards your project's data integrity.

Modify a material



To modify a material, simply click on the specific material box. This action will open a settings panel dedicated to that material. The material settings panel is divided into three main sections: Material Model, Material Properties, and Failure Properties. Each of these sections and their respective fields will be explained in detail in the following pages.

After making the desired adjustments to the material, click on the "**Save Changes**" button to save these modifications to the project's database. If you decide not to proceed with the changes, click on the "**Cancel**" button to discard any modifications.

Material Model

- **Material Label**: Assign a meaningful label to the material to ensure it is easily recognizable to you and your teammates. If you need to add multiple materials to the project, ensure each material has a unique name to avoid confusion.
- **Material Model**: This section requires two inputs from the user. The first field specifies the material's nonlinearity. The second field pertains to the material type. Currently, the system supports only Linear Elastic materials within the Pmass framework. For this type, you have two options to choose from: Isotropic and Orthotropic. Choosing a specific material model will update the required fields in subsequent sections to match the properties relevant to the selected material type.

Material Properties

As mentioned, the current version of Pmass supports only Linear Elastic materials, with options for either Isotropic or Orthotropic material models. Depending on the project dimension and the selected material model, the fields in the Material Properties section will adjust accordingly. Below is a table that outlines the variations in required fields based on the project dimension and material model:

Material Model		Project Dimension	Parameters
Linear Elastic	Isotropic	2D	Young's Modulus (E) Poisson's Ratio (v) Shear Modulus (G) Density (ρ)
Linear Elastic	Isotropic	2D	Young's Modulus (E) Poisson's Ratio (v) Shear Modulus (G) Density (ρ)
Linear Elastic	Orthotropic	3D	Young's Moduli (E1, E2) Poisson's Ratio (v12) Shear Modulus (G12) Density (ρ)
Linear Elastic	Orthotropic	3D	Young's Moduli (E1, E2, E3) Poisson's Ratios (v12, v23, v31) Shear Moduli (G12, G23, G31) Density (ρ)

This table ensures you input the correct properties for your material based on the structural specifications of your project.

Failure Properties

Pmass provides a comprehensive range of failure models that are extensively utilized in the failure analysis of engineering structures, whether in peridynamic studies or based on classical mechanics. Below is a table that lists the available failure models along with the required parameters for each. For a detailed description of how each failure model operates within Pmass, please refer to the "Failure Modeling in Pmass" document.

Failure Model	Parameters
Principle-Stress	Critical Tensile Stress (Stcr) Critical Compressive Stress (Sccr)
Principle-Strain	Critical Tensile Strain (Etcr) Critical Compressive Strain (Eccr)
vonMises-Stress	Critical Stress (Scr)
Volumetric-Strain	Critical Strain (Ecr)
vonMises-Strain	Critical Strain (Ecr)
Modified-vonMises-Stress	Tensile Strength (ft) Compressive Strength (fc) Softening Parameter (ef)
Johnson-Cook	D1, D2, D3, D4, D5
Mohr-Coulomb	C, phi
Hashin	Longitudinal Tensile Strength (Xt) Longitudinal Compressive Strength (Xc) Transverse Tensile Strength (Yt) Transverse Compressive Strength (Yc) Longitudinal Shear Strength (S12) Transverse Shear Strength (S23)
Hashin-Strain Energy	Longitudinal Tensile Strength (Xt) Longitudinal Compressive Strength (Xc) Transverse Tensile Strength (Yt) Transverse Compressive Strength (Yc) Longitudinal Shear Strength (S12) Transverse Shear Strength (S23) Critical Strain Energy (w0)
Strain Energy	Critical Strain Energy (w0)
Critical Stretch	Critical Stretch (s0)

This table assists in selecting the appropriate failure model and understanding the parameters needed to effectively simulate and analyze potential failures within your projects using Pmass.

Shapes

In Pmass, the component for managing shapes allows you to handle various geometries that define different aspects of your model, such as boundary conditions, failure/postprocessing setups, and the structure of the model itself. The term "shape" in Pmass is broadly used to describe any geometrical configuration that contributes to the simulation's setup.

Pmass categorizes shapes into six distinct types, each serving a specific function within the simulation framework:

1. Part: Parts collectively define the geometry of the model used in peridynamics simulations. Pmass generates the peridynamic grid by positioning peridynamic points inside these parts. For creating models for peridynamic analysis, Pmass supports two approaches: using the built-in model builder to create two- and three-dimensional geometries or importing a finite element (FE) model. This functionality allows for the modeling of complex engineering components, with specific materials and discretization schemes assignable to each part.

2. Void: This category is utilized to represent voids within the model. After the peridynamic grid is established, points that fall inside a void are removed, effectively creating empty spaces within the grid structure.

3. Inclusion: Inclusions are used to introduce materials with differing properties within the parts. Similar to voids, after the grid is generated, peridynamic points within an inclusion have their material properties replaced with those of the inclusion, thus integrating different material characteristics within the same part.

4. Crack: Predefined cracks are specified using this shape category. If a peridynamic bond (the interaction between two peridynamic points) intersects with a predefined crack, that bond is removed from the system, rendering it incapable of carrying loads.

5. Boundary: This shape category is essential for applying boundary conditions to the model. The kinetics of any peridynamic point that intersects with a boundary condition shape are dictated by that specific boundary condition, ensuring appropriate constraints and forces are applied to the model as required.

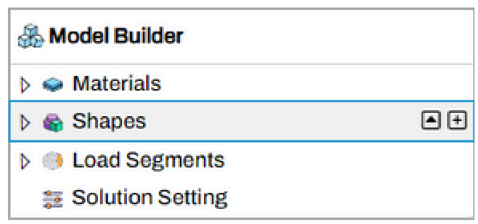
6. No-Failure Zone: To prevent damage in certain areas of the model, a no-failure zone can be designated. The failure analysis algorithm within Pmass will then skip points located within these zones, focusing computational efforts and damage simulations on more critical areas.

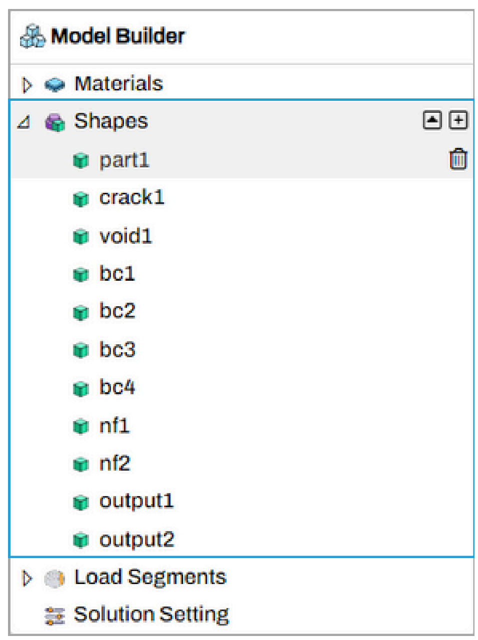
7. Output-Zone: Used primarily for post-processing, this category allows Pmass to calculate average values of displacement, force, strain, and stress among all points within an output zone during each loading increment. This data is crucial for post-processing the simulation results through graphical representations like displacement-vs-force or strain-vs-stress curves.

Each shape category in Pmass is designed to offer flexibility and precision in modeling, ensuring that both the geometrical configuration and simulation conditions are meticulously managed for optimal analysis and results.

The Shapes tree is the second component of the Model Builder box in Pmass. Similar to the Materials tree, it allows you to add new shapes, modify existing shapes, or delete shapes from the tree. To access and view the list of shapes, simply click on the "Shapes" box to expand the Shapes tree. This functionality enables efficient management of all geometric and boundary elements within your project, allowing for straightforward customization and adjustment of the simulation's setup.

The Shapes tree in the Model Builder holds a list of all the shapes included in the current project, each identified by a unique label.





Add a new shape (Built-in)

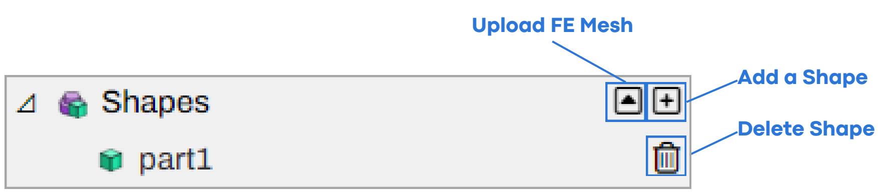
To add a new shape to your project using the built-in model builder, simply click on the "add" icon located on the Shapes box. This action will insert a default shape into the Shapes tree. From there, you can customize and define the shape's properties to fit the specific requirements of your simulation.

Add a new shape (Mesh)

To add a new shape as a Finite Element (FE) mesh, click on the "upload FE Mesh" icon in the Shapes box. This will insert a default FE-shaped mesh into the Shapes tree. You can then proceed to upload and define the FE mesh details, tailoring it to suit the specific needs and parameters of your project.

Delete a shape

To delete a shape, click on the delete icon located next to the selected shape. You will be prompted to confirm this action to ensure you are certain about removing the shape. This verification step helps prevent accidental deletions and maintains the integrity of your project setup.

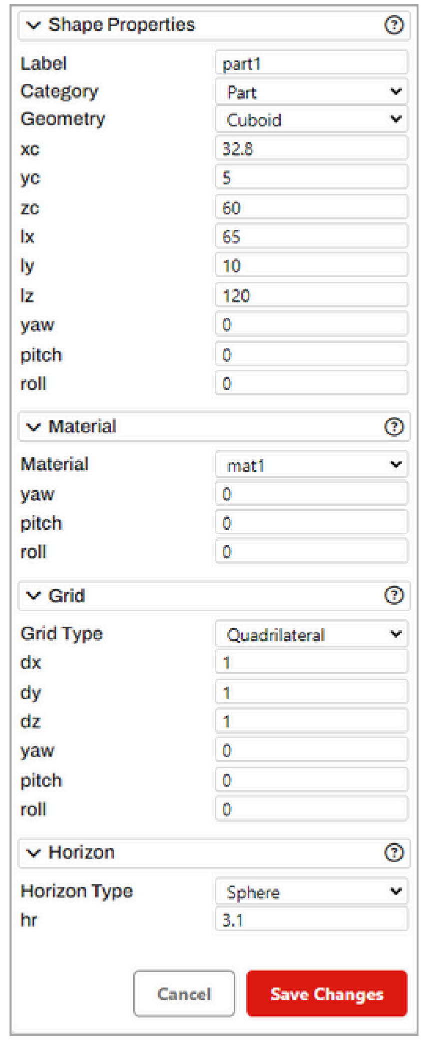


Modify a Built-in Shape

To modify a built-in shape, click on the specific shape box you wish to adjust. This action will bring up a settings panel for the selected shape. The content of this panel can vary depending on the shape's category, potentially including sections like Shape Properties, Material, Grid, Horizon, and Boundary Values.

Within this panel, you can make the desired adjustments to the fields. After making your modifications, click on the "**Save Changes**" button to save these changes to the project's database. If you need to revert any changes, simply click on the "**Cancel**" button to discard any adjustments made. This process ensures that you can tailor each shape precisely to the needs of your simulation while maintaining the flexibility to revise settings as needed.

Here's a detailed explanation of each section within the shape settings panel, allowing you to fully understand and utilize the customization options for each shape in your project:



Shape Properties

- **Shape Label**: Assign a unique and meaningful label to each shape to ensure it can be easily recognized by you and your teammates. It is advisable to use a label that reflects the category of the shape, making it easier to identify its role within the model at a glance.
- **Category**: Choose the category of the shape from the six available categories previously described. This classification helps in defining the specific function and interaction of the shape within the simulation.
- **Geometry**: Select the geometry of the shape from the available options. The options for geometry will vary based on the project dimension (2D or 3D) and the selected shape category. This selection is crucial as it defines the foundational structure of the shape.
- **Geometry Parameters**: These parameters determine the size and location of the geometry within the model. The list of parameters available will change depending on your geometry selection, allowing you to specify dimensions and positioning with precision.

Below is a table that outlines typical geometries along with their corresponding parameters to guide you in setting up your shapes:

Geometry	Project Dimension	Shape Category	Parameters
Rectangle	2D	Excluding Crack	Position (xc, yc) Length (lx), Width(ly) Rotation Angle (yaw)
Circle	2D	Excluding Crack	Position (xc, yc) Radius (r)
Sector	2D	Excluding Crack	Position (xc, yc) Inner Radius (r1), Outer Radius (r2) Polar Angles (θ1, θ2)
Line	2D	Crack	Start Point (x1, y1) End Point (x2, y2)
Arc	2D	Crack	Position (xc, yc) Radius (r) Polar Angles (θ1, θ2)
Cuboid	3D	Excluding Crack	Position (xc, yc, zc) Length (lx), Width(ly), Height (lz) Rotation angles (yaw, pitch, roll)
Sphere	3D	Excluding Crack	Position (xc, yc, zc) Radius (r)
Cylinder	3D	Excluding Crack	Position (xc, yc, zc) Height (lz) Radius (r) Rotation Angles (yaw, pitch, roll)
Sphere-Sector	3D	Excluding Crack	Position (xc, yc, zc) Inner Radius (r1), Outer Radius (r2) Polar Angles (θ1, θ2) Azimuthal Angles (ϕ1, ϕ2)
Cylinder-Sector	3D	Excluding Crack	Position (xc, yc, zc) Height (lz) Inner Radius (r1), Outer Radius (r2) Polar Angles (θ1, θ2) Rotation Angles (yaw, pitch, roll)
Rectangle	3D	Crack	Position (xc, yc, zc) Length (lx), Width(ly) Rotation Angles (yaw, pitch, roll)
Circle	3D	Crack	Position (xc, yc, zc) Radius (r) Rotation Angles (yaw, pitch, roll)
Sector	3D	Crack	Position (xc, yc, zc) Inner Radius (r1), Outer Radius (r2) Polar Angles (θ1, θ2) Rotation Angles (yaw, pitch, roll)

Shape Material

This section is specifically designed for the "Part" and "Inclusion" shape categories. Here, you can select a material from the list that has been added to this project. If the material is orthotropic, you will need to specify the material direction in this section. For isotropic materials, this step can be skipped as material direction is not relevant.

Grid

This section is exclusively for the "Part" shape category. Here, you are required to select the grid type and enter the related parameters. Below is a table that outlines the available grid schemes in Pmass along with their corresponding parameters, helping you choose the most suitable option for your simulation:

Grid Type	Project Dimension	Parameters
Quadrilateral	2D	Spacing (dx, dy) Rotation Angle (yaw)
Circular	2D	Spacing (dr) Center Position (xc, yc)
Quadrilateral	3D	Spacing (dx, dy, dz) Rotation Angles (yaw, pitch, roll)
Spherical	2D	Spacing (dr) Center Position (xc, yc, zc)
Cylindrical	2D	Spacing (dr, dz) Center Position (xc, yc) Rotation angles (yaw, pitch, roll)

Horizon

In the non-local theory of peridynamics, each point interacts with surrounding points within a finite distance known as the "horizon." The horizon determines the area within which material points affect each other, which is crucial for modeling the mechanical behavior of materials under various loads and conditions.
In 2D and 3D models within Pmass, the horizon typically assumes shapes that are geometrically suited to the dimensions of the model:

- In 2D models, the horizon can be a circle or rectangle, reflecting the planar nature of the simulation.
- In 3D models, the horizon can take the form of a sphere, cylinder, or cuboid, accommodating the additional spatial dimension.

These options allow for flexibility in defining how interactions within the material are modeled, enhancing the realism and precision of the simulation outcomes. Here is how you can set the horizon in Pmass based on the shape and dimensionality of your model:

Horizon Type	Project Dimension	Parameters
Quadrilateral	2D	Spacing (dx, dy) Rotation Angle (yaw)
Circular	2D	Spacing (dr) Center Position (xc, yc)
Cuboid	3D	Spacing (dx, dy, dz) Rotation Angles (yaw, pitch, roll)
Sphere	2D	Spacing (dr) Center Position (xc, yc, zc)
Cylinder	2D	Spacing (dr, dz) Center Position (xc, yc) Rotation angles (yaw, pitch, roll)

Choosing the correct horizon shape is essential for accurately capturing the behavior of materials and structures in peridynamic simulations.

Boundary

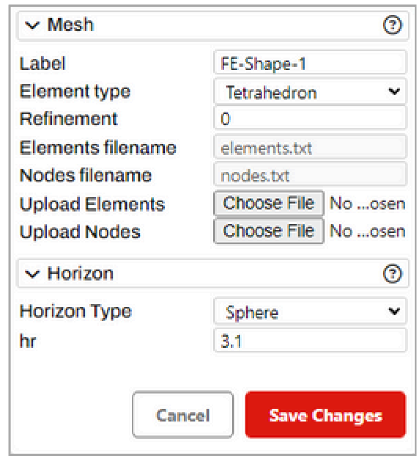
This section is exclusively for managing boundary conditions within the "Boundary" shape category in Pmass. Here's how you can set up boundary conditions for your simulation:

1. **Type**: In Pmass, boundary conditions can be implemented through either displacement-control or force-control methods:
   - **Displacement**-Control: If selected, the boundary value will be gradually applied to the boundary points as a form of displacement.
   - **Force**-Control: If selected, the boundary value will be gradually applied as a body force.
   - **Initial-Displacement**: If selected, the total displacement load is applied to the specified boundary and remains constant throughout the simulation.
   - **Initial-Force**: If selected, the total force load is applied to the specified boundary and remains constant throughout the simulation.
2. **Value**: Enter the final value of the boundary loading. This could be the magnitude of displacement or force, depending on the type of boundary condition selected.
3. **Direction**: Select the direction in which you want to apply the boundary condition. This specifies how the load or displacement will be oriented relative to the model.

Note: If you need to assign boundary conditions in two or more directions for a given area or volume, you should add two different boundary shapes and specify the desired directions for each. This allows for more complex and realistic simulation setups where multiple forces or displacements act simultaneously or sequentially on different parts of the model.
This structured approach ensures precise control over how external constraints and loads are applied to your simulation, crucial for accurate modeling of physical behaviors and interactions.

Upload FE Mesh

When creating models in Pmass, importing a Finite Element (FE) mesh is a powerful method that allows you to add detailed and customized shapes to your project. To add a FE-shape, click on the "**upload FE Mesh**" icon within the Shapes box on the Model Builder interface. This action will automatically insert a default FE-shape into the Shapes tree. You can proceed to upload and configure the FE mesh details, tailoring the mesh to meet the specific requirements of your project.



To access Shape Settings, click on the specific FE-shape box you wish to adjust. This opens the settings panel for that shape, which includes the sections "Mesh" and "Horizon." Within this panel, you can modify the settings and upload necessary files. After making your modifications, click "Save Changes" to update the project's database with your new configurations. If you decide against the changes, click "Cancel" to revert any adjustments.

Here's a detailed explanation of each section within the FE-shape settings panel, allowing you to fully understand and utilize the customization options for each shape in your project:

Mesh

- **Label**: Assign a unique and meaningful label to each shape to ensure it can be easily recognized by you and your teammates.
- **Element type**: Choose the appropriate element type based on your project's dimensionality:
  - **2D Options**: Triangle (3-Node), Quadrilateral (4-Node)
  - **3D Options**: Tetrahedron (4-Node), Hexahedron (8-Node)
- **Refinement**: Specify an integer for mesh refinement. This parameter controls how the FE mesh is subdivided to generate the peridynamic grid. Higher refinement levels increase the number of peridynamic points, enhancing model detail and accuracy. Refinement Level 0 (Default) means Pmass uses the original uploaded elements to generate the peridynamic grid. Refinement Level 1 and higher means each element is divided into sub-elements within each refinement step, with the edges of these sub-elements being half the length of the original element's edges.

Number of Sub-elements Based on Element Type:
- **Triangle**: Splits into 4 sub-elements per step of refinement.
- **Quadrilateral**: Also splits into 4 sub-elements per step of refinement.
- **Tetrahedron**: Splits into 6 sub-elements per step of refinement.
- **Hexahedron**: Splits into 8 sub-elements per step of refinement.

- **Elements filename and Nodes filename:** These fields are automatically filled based on the uploaded files and are not editable.
- **Upload Elements:** Upload a text file that lists elements, where each line includes the material ID followed by the element's node IDs.
- **Upload Nodes:** Upload a text file that lists nodes, where each line starts with the node ID followed by its position (X, Y for 2D; X, Y, Z for 3D).

Horizon

- **Horizon Type:** Select the shape of the horizon suitable for your model's dimension:
  - **2D Models:** Options include circle or rectangle.
  - **3D Models:** Options include sphere, cylinder, or cuboid.
- **Horizon Parameters:** Define the size of the horizon, expressed as a ratio of the element size. This means the actual horizon size is calculated by multiplying these parameters by the nominal size of the element, derived from the square root of the element's area in 2D or the cube root of the element's volume in 3D.

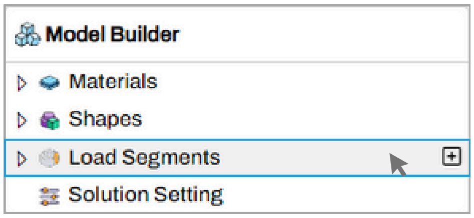
Load Segments

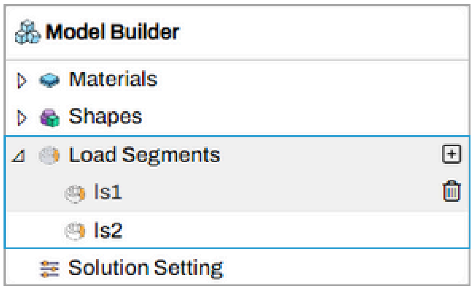
In Pmass, to enhance computational efficiency and manage complex loading scenarios effectively, you can add multiple loading segments to your simulation. Each segment can have different load ratios and number of increments. This approach allows for more detailed control over how loads are applied throughout the simulation, accommodating varying conditions and stages of analysis. Here's how this functionality can be utilized:

1. Load Ratio: This specifies the proportion of the total load applied during the specific segment. Adjusting the load ratio allows for gradual application of force or displacement, simulating real-world conditions where loads may not be applied all at once.
2. Number of Increments: This defines how many steps the particular load segment will be divided into. More increments mean a smoother application of the load, which can be critical for capturing the material response accurately, especially under complex load conditions.

By configuring multiple loading segments with tailored load ratios and increments, you can simulate a wide range of loading scenarios more realistically and efficiently, helping to reduce computational time while improving the quality of the simulation outcomes.

The Load Segments tree in Pmass organizes and displays all the load segments of your current project, with each segment identified by a unique label. This feature allows for easy management and reference of different loading scenarios within the simulation. Each label helps you quickly identify specific segments, facilitating adjustments and analysis as needed, ensuring that the project's setup remains clear and navigable.





Add a new load segment

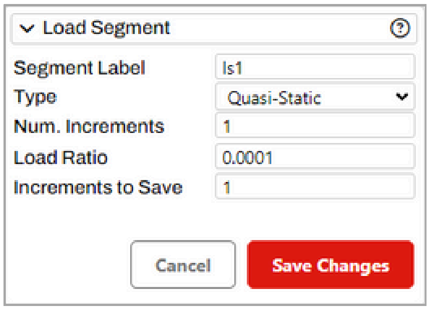
To add a new load segment to your project in Pmass, simply click on the "add" icon located in the Load Segment box. This action will insert a default load segment into the Load Segments tree. From there, you can customize this segment according to the specific loading conditions and parameters you want to simulate, such as adjusting the load ratio and setting the number of increments to suit your project's requirements.

Delete a load segment

To delete a load segment in Pmass, click on the delete icon next to the load segment you wish to remove. You will be prompted to confirm this action to ensure that you want to permanently delete the load segment. This verification helps prevent accidental deletions and ensures that you are certain about removing specific simulation data from your project.

Modify a load segment

To modify a load segment, click on the box corresponding to the specific load segment you want to adjust. This action will open a settings panel for that load segment. Here, you can make the necessary changes to various fields within the panel.



After making the desired adjustments, click on the "**Save Changes**" button to save these changes to the project's database. If you decide against the changes, click on the "Cancel" button to discard any modifications made. This process ensures that your simulation parameters reflect the precise conditions needed for your analysis.

1. **Label**: Provide a unique and meaningful label for the load segment to ensure it can be easily recognized by you or your teammates. This label should ideally convey the nature or purpose of the load segment to facilitate quick identification within the project.
2. **Type**: Select the type of the load segment based on the solver used in the project:
    - If a **Quasi-Static** solver was selected during project creation, the only option available for the load segment type will be Quasi-Static.
    - If the solver is **Transient**, the load segment type available will be Transient.
    - If the solver is **Hybrid**, you will have two options: Quasi-Static and Transient. Choose based on the specific dynamics and time-dependence of the load being simulated.
3. **Num. Increments:** Specify the total number of increments for this load segment. This determines how the total load is divided over time, allowing for a detailed and gradual application of load which can be crucial for understanding material response.
4. **Load Ratio:** Define the ratio of the total load that you want to apply to the model during this load segment. This ratio will be applied evenly across the specified number of increments, ensuring controlled and precise loading conditions.
5. **Increment to save:** Determine the number of increments for which you want to save the results. If set to one, Pmass will save the results after every increment. Choosing a higher number will save results less frequently, which can be useful for reducing data storage requirements and focusing on specific points in the simulation for detailed analysis.

When adding load segments to your project in Pmass, it's essential to tailor the setup based on the type of analysis and the specific conditions you're investigating. Here are some tailored tips for setting up load segments effectively for different scenarios:

1. Stress Analysis under Quasi-Static Loading

- **Increments**: Use 1 increment to apply the total load at once, which is typical for simple stress analysis where dynamic effects are negligible.
- **Increment to Save**: Set this to 1 to ensure that results are saved immediately after the load is applied, capturing the peak stress state.

2. Quasi-Static Fracture Analysis

- For the initial stages of fracture analysis where no damage is expected yet, use the following settings:
    - **Load Ratio**: Choose a load ratio that brings the model up to but not exceeding the stress state before damage begins.
    - **Increments**: Use 1 or 2 increments to quickly reach this pre-damage load state.
- For subsequent stages during crack propagation:
    - **Increments**: Use a larger number of increments to apply the load more gradually. This helps simulate the crack propagation process more realistically by allowing the model to adjust and redistribute stresses between increments.
    - **Load Ratio**: Gradually increase the load to simulate the stress increase that would cause further cracking.

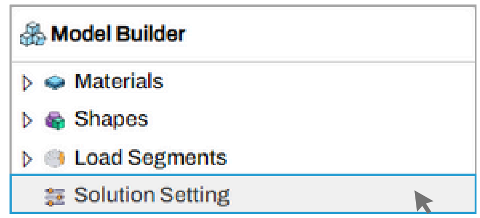
3. Hybrid Solution for Dynamic Loading Conditions

- **Initial Quasi-Static Segment**: Use a quasi-static load segment first to apply a load until the first signs of damage or critical stress points are observed.
    - **Type**: Quasi-Static
    - **Increments**: Typically, this would be a smaller number, enough to reach the critical stress point but not cause significant damage.
- **Subsequent Transient Segment**: After the initial damage points are identified, switch to a transient load segment to simulate the dynamic response under continued loading.
    - **Type**: Transient
    - **Increments**: Use more increments to closely simulate the dynamic effects and damage progression post-initial damage.

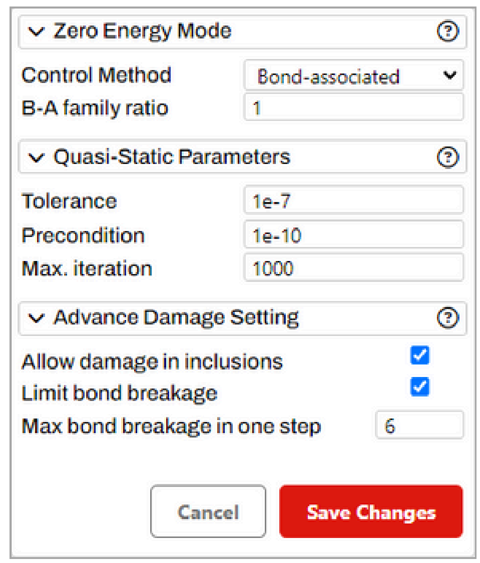
By carefully considering these tips and adjusting the load segments accordingly, you can significantly enhance the accuracy and relevance of your simulations, particularly when dealing with complex loading scenarios and fracture mechanics.

Solution Setting

The final component of the Model Builder in Pmass is where you set up parameters specific to the non-ordinary state based peridynamic solution. This setup is crucial for tailoring the simulation to accurately reflect the unique behavior of materials and structures under various conditions.



1. Accessing the Settings: Click on the "Solution Setting" box within the Model Builder. This action opens the settings panel where you can configure the necessary parameters.
2. Adjusting the Settings: The available sections and options in the setting panel may vary depending on the project's dimension (2D or 3D) and the type of solver you selected at the project's creation. This customization ensures that you have the appropriate tools and options to accurately model the peridynamic behavior of the system under study.



Zero Energy Model

In the "**Control Method**" section of Pmass, there are specific settings designed to suppress the zero-energy mode often observed in non-ordinary state-based peridynamic models (NOSB-PD). Here's a breakdown of the methods available and how to configure them:

Control Methods Available:

1. **Bond-associated** (B-A): This method involves associating additional constraints or properties with the peridynamic bonds to mitigate zero-energy deformations that do not contribute to the overall stress state of the material.
2. **Supplemental-force** (S-F): Involves applying additional forces calculated to counteract the zero-energy modes, effectively stabilizing the simulation.
3. **Averaging**: This method uses averaging techniques over a set of bonds or within a neighborhood to smooth out fluctuations that may lead to zero-energy modes.

Setting the Parameters:

- **B-A Family Ratio:**
    - If the Bond-associated method is selected, choose a ratio between 1 and 1.5. This ratio determines the extent of the bond-associated stabilization effect, with higher values increasing the rigidity of the model against zero-energy modes.
- **S-F Spring Constant:**
    - If the Supplemental-force method is selected, you need to provide a value for the S-F spring constant. This constant defines the stiffness of the supplemental springs used to stabilize the model. The appropriate value can vary based on the material properties and the specific dynamics of the simulation.

Further Details:

For more comprehensive information on each method, please refer to the "NOSB-PD" section in the Pmass documents. These documents will provide detailed explanations of how each method works, the theoretical basis for their effectiveness, and guidelines on choosing the most suitable method and settings for your specific simulation needs.

Thickness

In Pmass, when setting up a 2D project, it's important to specify the thickness of the whole model. This parameter is crucial because, even though the simulation is conducted in two dimensions, the third dimension (thickness) affects the overall mass, stiffness, and response of the model under load. Here’s how to set this up:

Transient Parameter

When using a Transient or Hybrid solver in Pmass, it's crucial to specify the time increment, which dictates the simulation's time resolution. Here's how you can effectively set this up:

Choosing the Right Increment:

- Transient Solver: For dynamic simulations where the solver captures the time-dependent response of materials and structures to loads, choosing a smaller time increment can improve accuracy but increase computational time.
- Hybrid Solver: In scenarios involving both static and dynamic analyses, the time increment should be chosen based on the dynamic phase of the analysis. Ensure that it's sufficiently small to accurately capture the quick changes in conditions typical of dynamic responses.

Quasi-Static Parameter

For simulations using a Quasi-static or Hybrid solver in Pmass, it's important to configure the quasi-static parameters effectively to ensure accurate and efficient results. These settings influence how the Biconjugate Gradient Stabilized (BiCGSTAB) method solves the system of equilibrium equations incrementally.

Setting Quasi-Static Parameters:

1. **Tolerance**: This is the threshold for exiting the BiCGSTAB solution process. If the residual error of the solution falls below this value, the current solution is deemed acceptable, and the iterative process is terminated. Set the tolerance based on the desired accuracy of your results. Lower values lead to higher accuracy but may increase computational time.
2. **Precondition**: A preconditioner improves the convergence rate of the BiCGSTAB method. It modifies the system of equations to be more amenable to iterative solving methods.
3. **Max-Iteration**: This parameter sets a cap on the number of iterations the BiCGSTAB method will execute. It acts as a safeguard against excessively long computation times in cases where the desired tolerance is not quickly met. If the error decreases very slowly and the set number of iterations is reached without satisfying the error threshold, the solution process will stop. This prevents wasteful computation and ensures efficiency. The value for Max-Iteration should balance between giving enough iterations to achieve convergence and not allowing the solver to run indefinitely. Consider the complexity of your model and the expected difficulty in reaching convergence when setting this parameter.

Implementing the Settings:

- When entering these parameters in Pmass, ensure they are configured within the solver settings section of the software interface. Accurate setting of these parameters is crucial for the stability and efficiency of the quasi-static analysis.

By carefully adjusting the tolerance, selecting an appropriate preconditioner, and setting a practical limit for maximum iterations, you optimize the performance of the BiCGSTAB solver in Pmass, leading to reliable and computationally efficient simulations.

Advanced Damage Setting

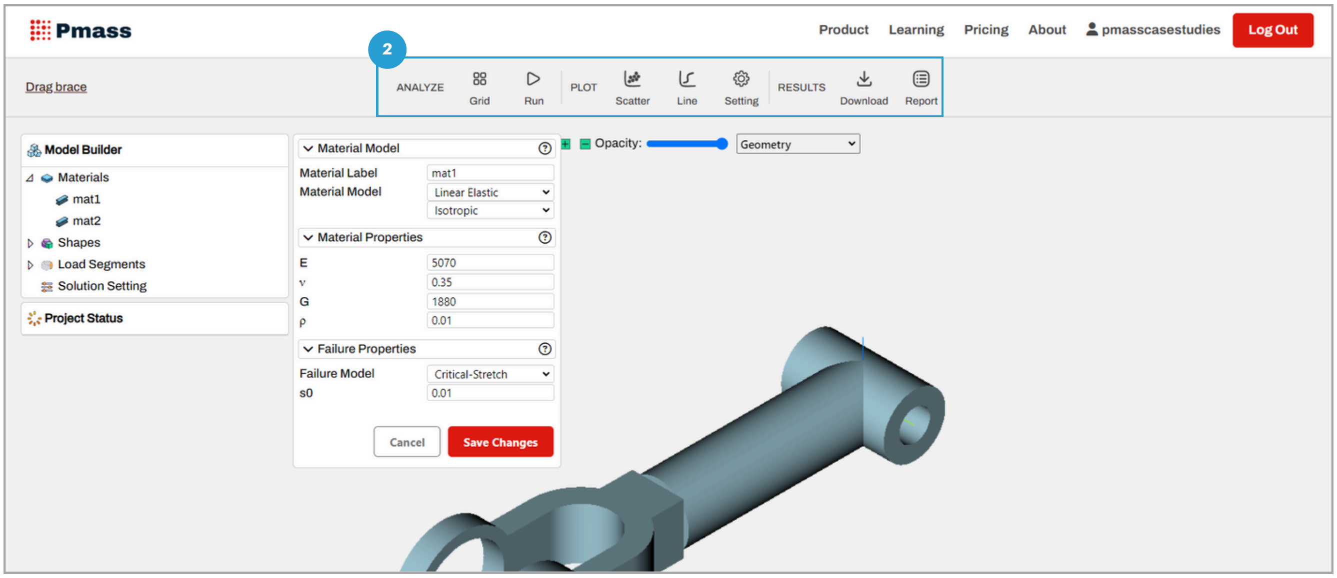
When setting up simulations in Pmass, particularly for materials such as fiber reinforced composites or particle reinforced cementitious composites, you have the option to configure how damage propagates within the material. This setting is crucial for achieving realistic and computationally efficient results.

Configuring Damage Propagation in Inclusions:

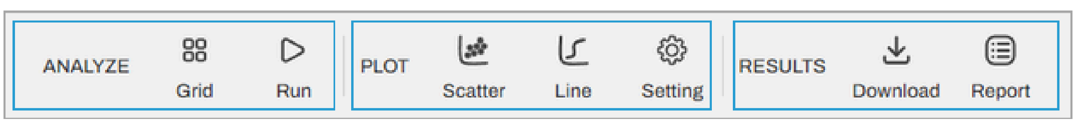
1. **Allowing Damage in Inclusions**: Choose whether to allow damage propagation within inclusions. For materials that are designed to be reinforced, like composites, it might be computationally efficient or realistic to prevent damage within the inclusions.
2. **Controlling Damage Propagation**: Normally, Pmass checks for damage at the end of each loading increment and breaks all bonds that meet the failure criteria. However, using large load increments can result in breaking too many bonds simultaneously, leading to an unrealistic damage path. To mitigate this, Pmass allows you to limit the number of bond breakages in each step.
    - **Max Number of Bonds**: Specify the maximum number of bonds allowed to break in each step. This ensures that only the most critical bonds are broken first.
    - **Process**: At the end of the increment, if the maximum bond breakage limit is reached but more bonds still meet the failure criteria, the same load increment is repeated. This process continues until no additional bonds meet the failure criteria.
    - **Next Steps**: Once no more bonds satisfy the failure criteria under the current load, the next increment is applied, allowing for a more gradual and controlled damage progression.

By carefully configuring whether to allow damage within inclusions and controlling how bonds break during the simulation, you can enhance both the realism and computational efficiency of your simulations in Pmass. This approach helps in better understanding the material behavior under various loading conditions and in predicting the structural failure more accurately.

Controlbar



The Controlbar, strategically positioned at the top-center of the workbench page in Pmass, serves as a central hub for managing the simulation process from start to finish. It's designed to facilitate preprocessing, running simulations, and post-processing the results efficiently.



Analyze

- **Grid:** The Grid button is crucial for preparing your model for simulation. It generates a peridynamic grid which is computed on the cloud using AWS Lambda Function, ensuring that your model's geometry is discretized into points necessary for the peridynamic analysis.
- **Run:** This button launches the simulation. By clicking Run, you initiate the setup and execution of the simulation on the cloud, leveraging cloud computing resources to handle complex computations efficiently.

Plot

- **Scatter:** Use this button to visualize the undeformed grid, helping you check the initial setup before simulation runs. After the simulation, it also assists in post-processing the results to understand the deformation or failure mechanisms visually.
- **Line:** This tool is for creating line plots, which are essential for analyzing relationships in the data, such as displacement versus force or strain versus stress curves. These plots help in understanding the material behavior under applied loads.
- **Setting:** The Plot Setting option allows for customization of the visualization parameters, such as color schemes, line thickness, and the perspective from which you view the plots. These settings help tailor the plot appearances to better suit presentation or detailed analysis needs.

Results

- **Download:** This function enables you to download simulation results in a text format. It's particularly useful if you prefer using a third-party tool for further post-processing, offering flexibility in how you analyze and interpret the simulation data.
- **Report:** Users can download a comprehensive analysis report in PDF format. This report includes all relevant details and findings from the simulation, formatted for easy sharing and review.