Ansys

2025 R1

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

® 2025 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Mechanical Beta Features

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R1
January 2025

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2025 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

BETA

Table of Contents

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BETA

List of Figures

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beta Features Disclaimer

This is beta documentation for one or more beta software features.

- Beta features are considered unreleased and have not been fully tested nor fully validated. The results are not guaranteed by Ansys, Inc. (Ansys) to be correct. You assume the risk of using beta features.
- At its discretion, Ansys may release, change, or withdraw beta features in future revisions.
- Beta features are not subject to the Ansys Class 3 error reporting system. Ansys makes no commitment to resolve defects reported against beta features; however, your feedback will help us improve the quality of the product.
- Ansys does not guarantee that database and/or input files used with beta features will run successfully from version to version of the software, nor with the final released version of the features. You may need to modify the database and/or input files before running them on other versions.
- Documentation for beta features is called beta documentation, and it may not be written to the same standard as documentation for released features. Beta documentation may not be complete at the time of product release. At its discretion, Ansys may add, change, or delete beta documentation at any time.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Activating Mechanical Beta Options

To make beta options available:

From Workbench

1. On the Workbench Project page, from the **Tools** menu, **Options** and then selection of the option **Appearance**.
2. Scroll down and activate the **Beta Options** check box.
3. Click **OK**.
4. Open or return to Mechanical.

From Mechanical

1. Open the Options dialog by selecting the **File** tab and then **Options**. You can also select the **Options** icon beside the **Quick Launch** field on the title bar.
2. Open the **Common Setting** option and select the **User Interface** category.
3. Set the **Show Beta Options** preference to **Yes**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BETA

Chapter 2: Two-dimensional (2D) Acoustics Analyses

Using Beta Options (p. 1), you can perform a two-dimensional (2D) acoustics analysis.

- 2D Acoustics is supported in Coupled Field Analyses. Supported in the following analyses:
  - Coupled Field Harmonic
  - Coupled Field Modal
  - Coupled Field Static
  - Coupled Field Transient
- Supported with other physics and coupling such as:
  - Structural Acoustics
  - Acoustics with Piezoelectric Coupling
  - Acoustics with Electrostatic force coupling
  - Applicable Boundary Conditions are supported
  - Applicable results are supported

Note:

Far Field results are not yet supported.

BETA

BETA

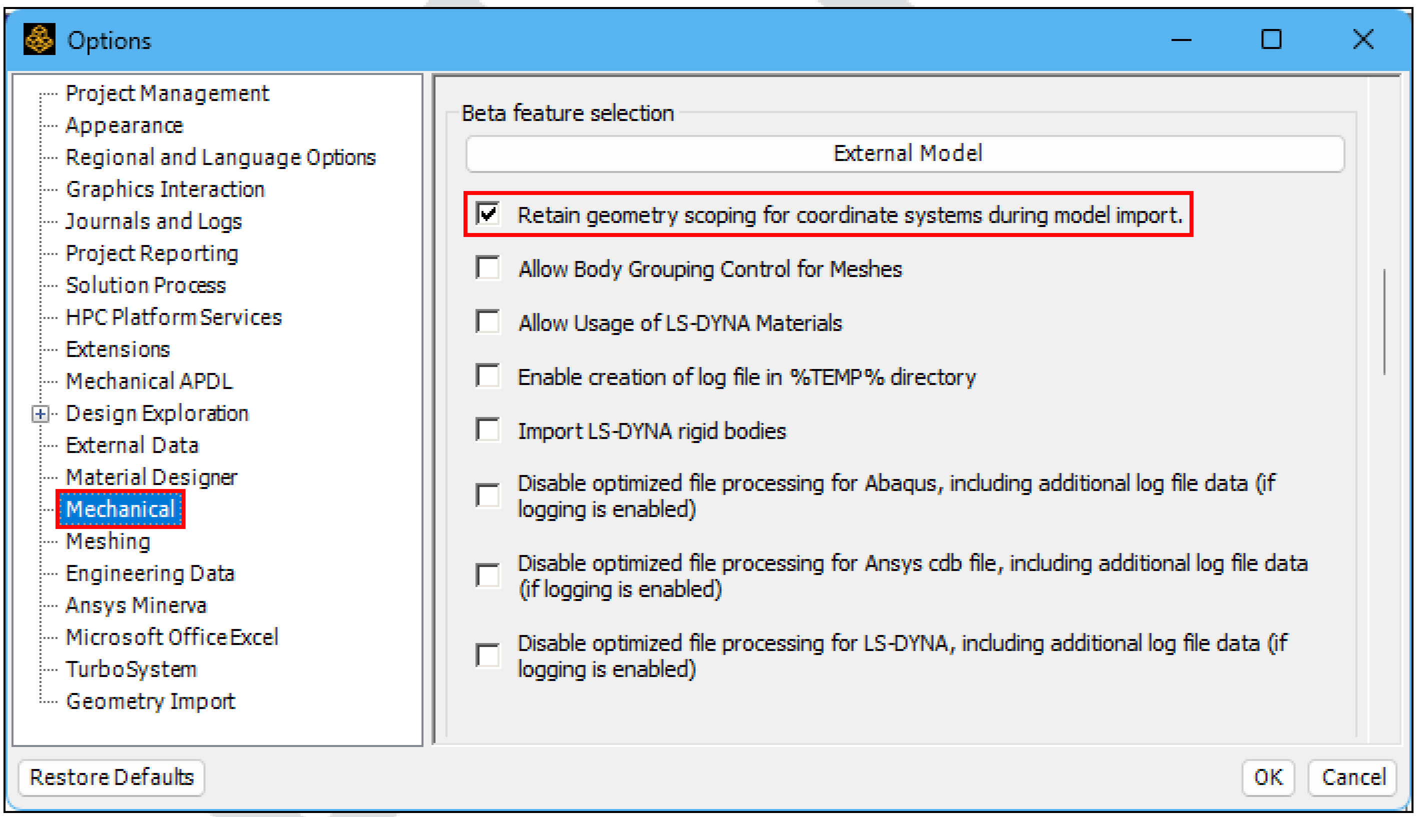
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Maintain Coordinate System Scoping During Model Assembly

The model assembly feature of Workbench and Mechanical enables you to import multiple meshed-based geometries into a single downstream Mechanical analysis system by linking the upstream **Model** cells to the **Model** cell of the downstream system.

Using a beta option, you can now maintain the geometry-based scoping of **Coordinate System** objects from upstream system(s). Currently, imported Coordinate Systems only include their location and directions and not the geometry selection specified in the upstream system.

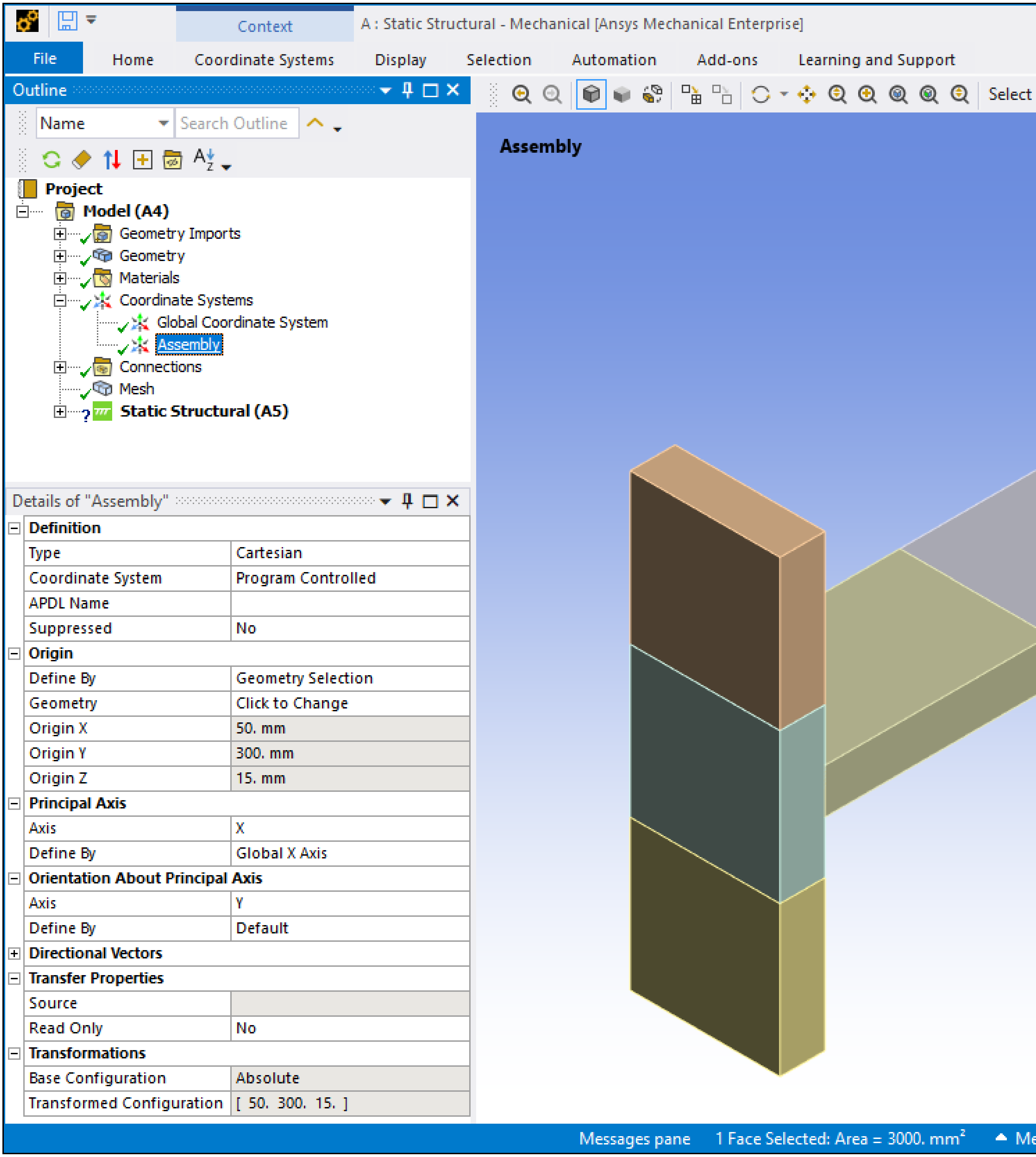
In Workbench, in addition to activating the Beta Options (p. 1), you must also activate the feature selection option (**Options** > **Tools**> Mechanical) highlighted below. You need to close and reopen the Workbench application before the capability becomes available.



The following examples show a **Coordinate System** created in an upstream system and imported into a downstream system.

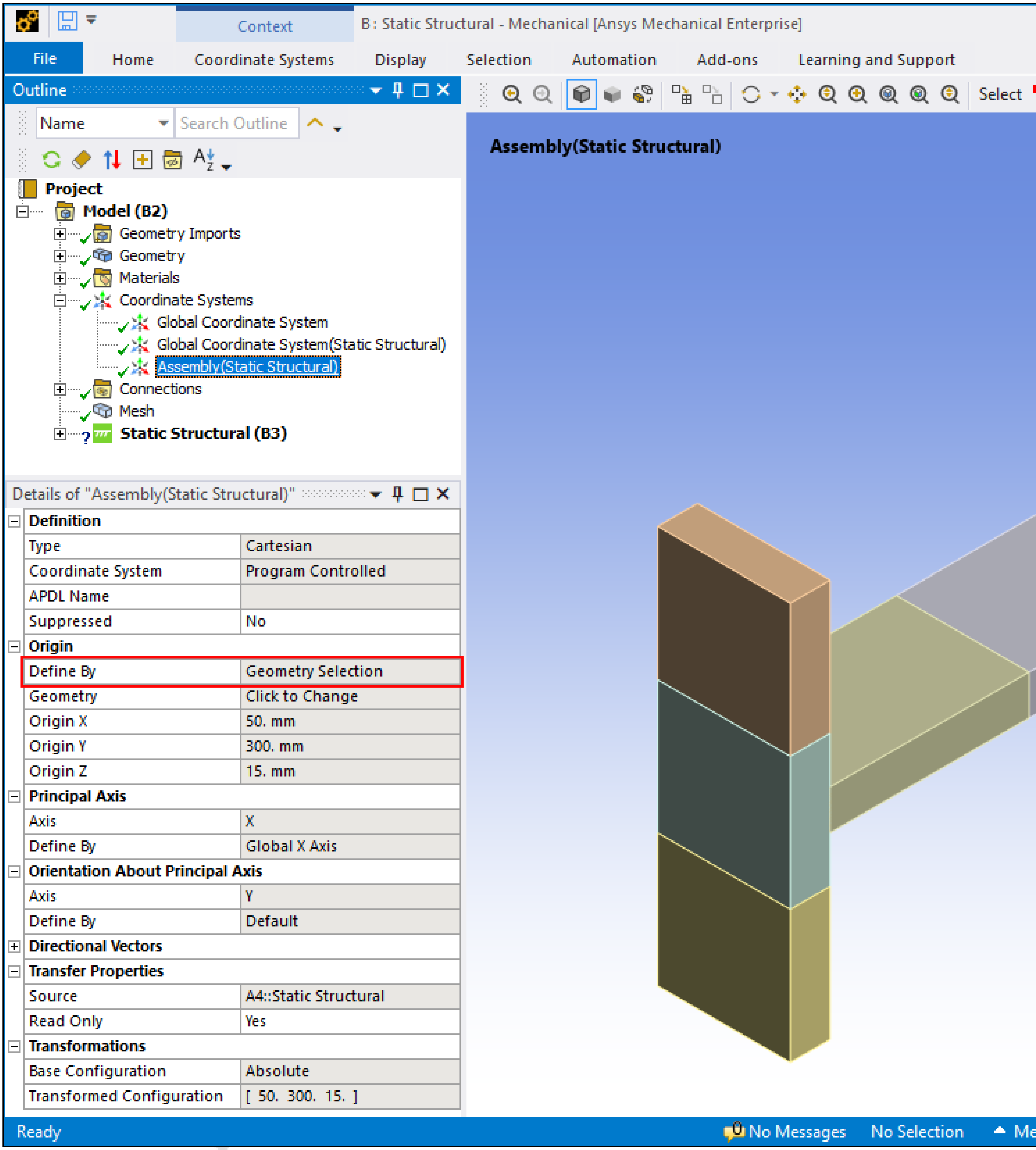
Upstream System

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Downstream System

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BETA

Chapter 4: Using Degenerated Tetrahedral Elements

For structural and thermal analyses, the **Degenerated Shape (Beta)** property can be specified to have the application use degenerated tetrahedral elements on geometry parts.

Application

1. Activate Beta Options (p. 1).
2. Select the **Geometry** object and set the **Element Control** property to **Manual**.
3. Select the desired body object and specify one of the following for the **Degenerated Shape (Beta)** property:
    - **Program Controlled** (default): The application selects the suitable element type.
    - **Yes**: The solver will use linear order degenerated tetrahedral elements. For example, in the case of a structural analysis the application would use SOLID185.
    - **No**: The solver will use linear order tetrahedral elements. For example, in the case of a structural analysis, the application would use SOLID285.

Note:

This property is only applicable for a solid geometry that:

- Is not generated from **Pull** objects.
- Is not a rigid body.
- Has a linear order tetrahedral mesh.

Supported Analysis Types

- Eigenvalue Buckling
- Harmonic Response
- Modal
- Static Structural
- Substructure Generation
- Steady-State Thermal

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Transient Structural
- Transient Thermal

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

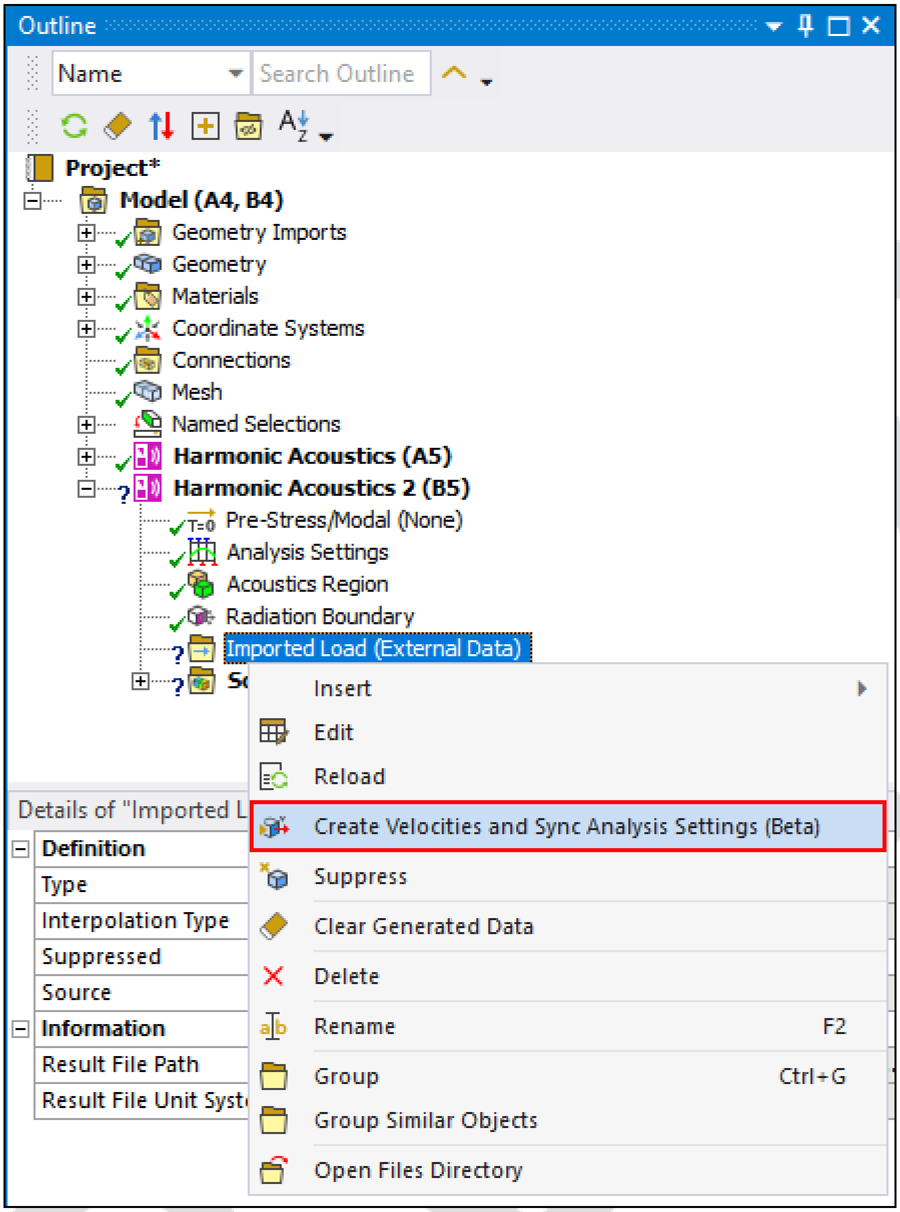
Chapter 5: Import Velocity and Synchronize Analysis Settings for H5DPF Files

When importing velocities from an upstream External Data system using an `*.h5` file, the imported data often contains numerous orders or frequencies, each with multiple RPM values. Use the context (right-click) menu beta option, **Create Velocities and Sync Analysis Settings (Beta)**, to automatically synchronize the **Analysis Settings** of the downstream system with the imported velocity data.

Procedure

1. Activate Beta Options (p. 1).
2. Once you import the file into the application, right-click the **Imported Load (External Data)** object and select **Create Velocities and Sync Analysis Settings (Beta)**.

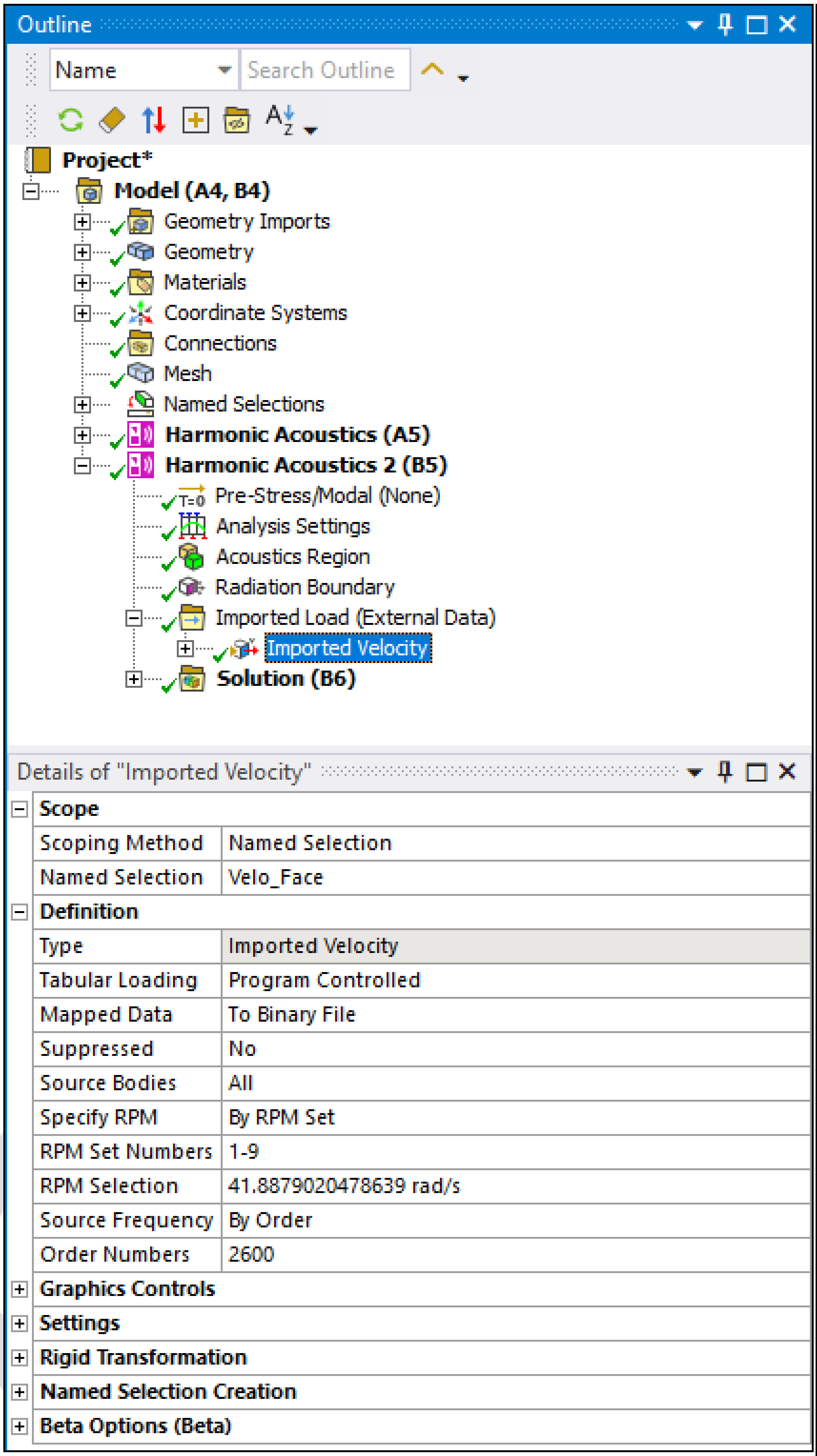
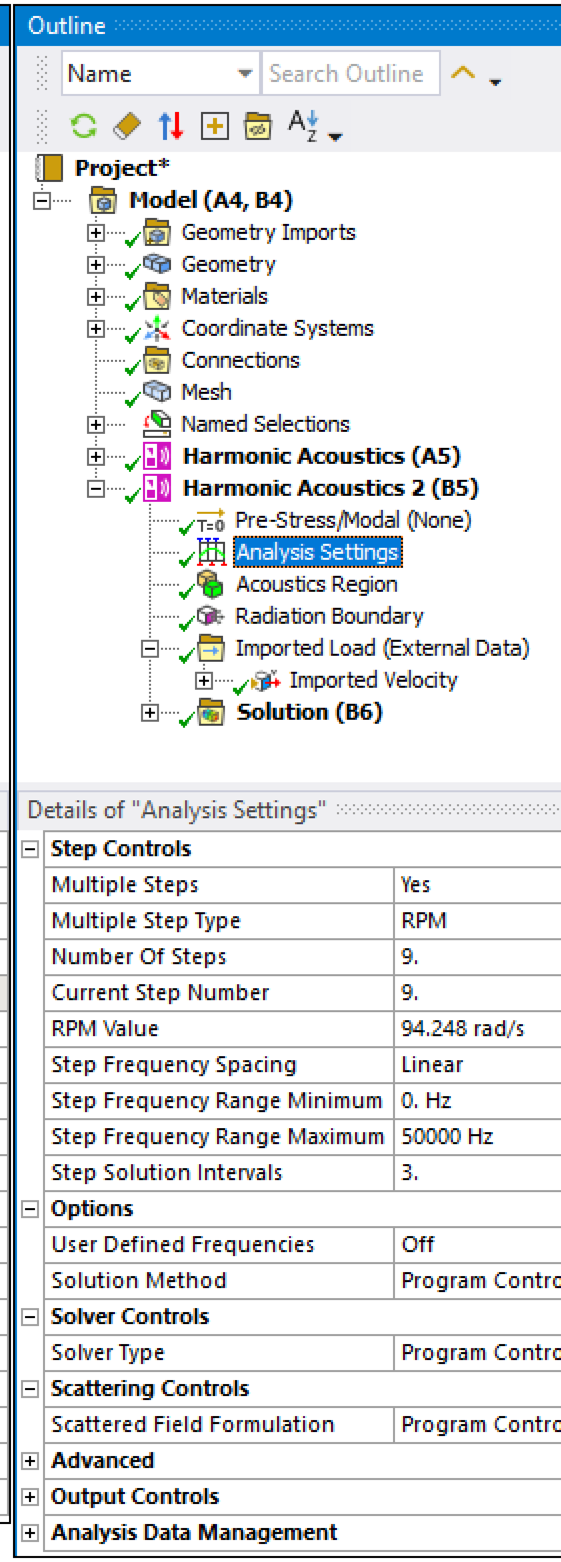
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once selected, the application automatically:

- Creates an Imported Velocity child object.
- Sets the **Mapped Data** property to **To Binary File** and the **Specify RPM** property to **All**.
- Synchronizes the **Analysis Settings** object with the settings of the upstream system.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.


Outline
Name
Search Outline
Project*
Model (A4, B4)
Geometry Imports
Geometry
Materials
Coordinate Systems
Connections
Mesh
Named Selections
Harmonic Acoustics (A5)
Harmonic Acoustics 2 (B5)
Pre-Stress/Modal (None)
Analysis Settings
Acoustics Region
Radiation Boundary
Imported Load (External Data)
Imported Velocity
Solution (B6)
Details of "Imported Velocity"
Scope
Scoping Method
Named Selection
Named Selection
Velo_Face
Definition
Type
Imported Velocity
Tabular Loading
Program Controlled
Mapped Data
To Binary File
Suppressed
No
Source Bodies
All
Specify RPM
By RPM Set
RPM Set Numbers
1-9
RPM Selection
41.8879020478639 rad/s
Source Frequency
By Order
Order Numbers
2600
Graphics Controls
Settings
Rigid Transformation
Named Selection Creation
Beta Options (Beta)

Outline
Name
Search Outline
Project*
Model (A4, B4)
Geometry Imports
Geometry
Materials
Coordinate Systems
Connections
Mesh
Named Selections
Harmonic Acoustics (A5)
Harmonic Acoustics 2 (B5)
Pre-Stress/Modal (None)
Analysis Settings
Acoustics Region
Radiation Boundary
Imported Load (External Data)
Imported Velocity
Solution (B6)
Details of "Analysis Settings"
Step Controls
Multiple Steps
Yes
Multiple Step Type
RPM
Number Of Steps
9.
Current Step Number
9.
RPM Value
94.248 rad/s
Step Frequency Spacing
Linear
Step Frequency Range Minimum
0. Hz
Step Frequency Range Maximum
50000 Hz
Step Solution Intervals
3.
Options
User Defined Frequencies
Off
Solution Method
Solver Controls
Solver Type
Scattering Controls
Scattered Field Formulation
Advanced
Output Controls
Analysis Data Management

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Scoping Substructure Generation Analyses

By default, the Substructure Generation analysis creates one superelement for all active bodies. Now, when the Beta Options (p. 1) feature is active, the Substructure Definition object provides scoping properties that enable you to select desired geometry including body selection or body-based Named Selections. See the example below.

And when you specify a **Total Deformation** result (the only supported result), the application automatically scopes the result to the specified bodies.

Important:

When using this beta scoping capability, note the following limitations:

- Your **Substructure Generation** analysis will fail if you have a linked upstream **Static Structural** analysis that includes either of the following loads scoped to bodies that <u>are not</u> included on the scoping of the downstream **Substructure Definition** object:

  - A **Pressure** load specified with the **Define By** property set to **Normal To** or **Vector**.

    Or...

  - A **Hydrostatic Pressure** load specified with the **Define By** property is set to **Vector**.

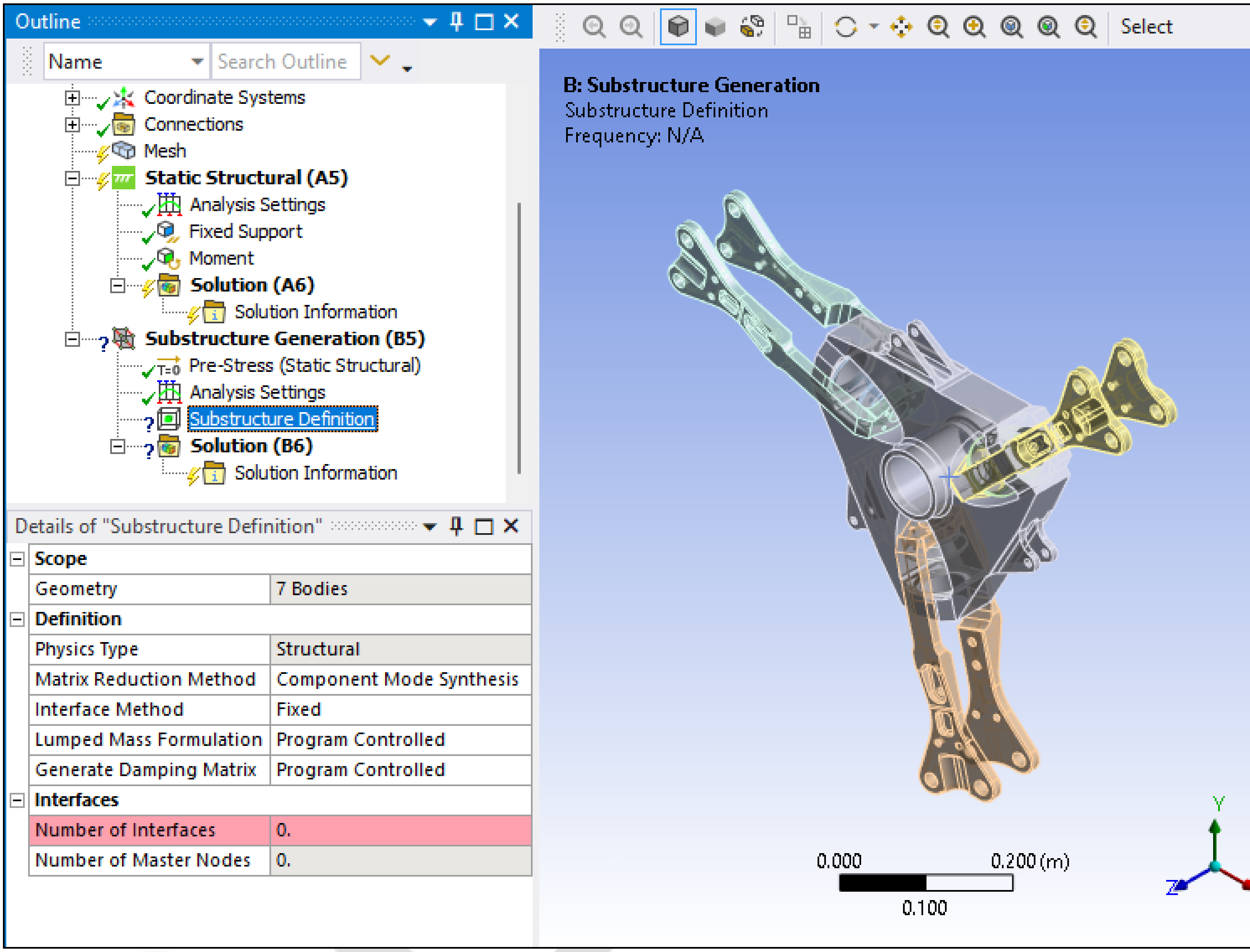
  You can avoid this issue by applying an equivalent load for either of the above loads using the **Components** option of the **Define By** property. Or you can use a **Nodal Pressure** load instead of **Pressure**/**Hydrostatic Pressure**.

- In addition, Ansys recommends that you only include **Contacts** and **Joints** in the model to avoid failures for substructure generation.

Example Scoping

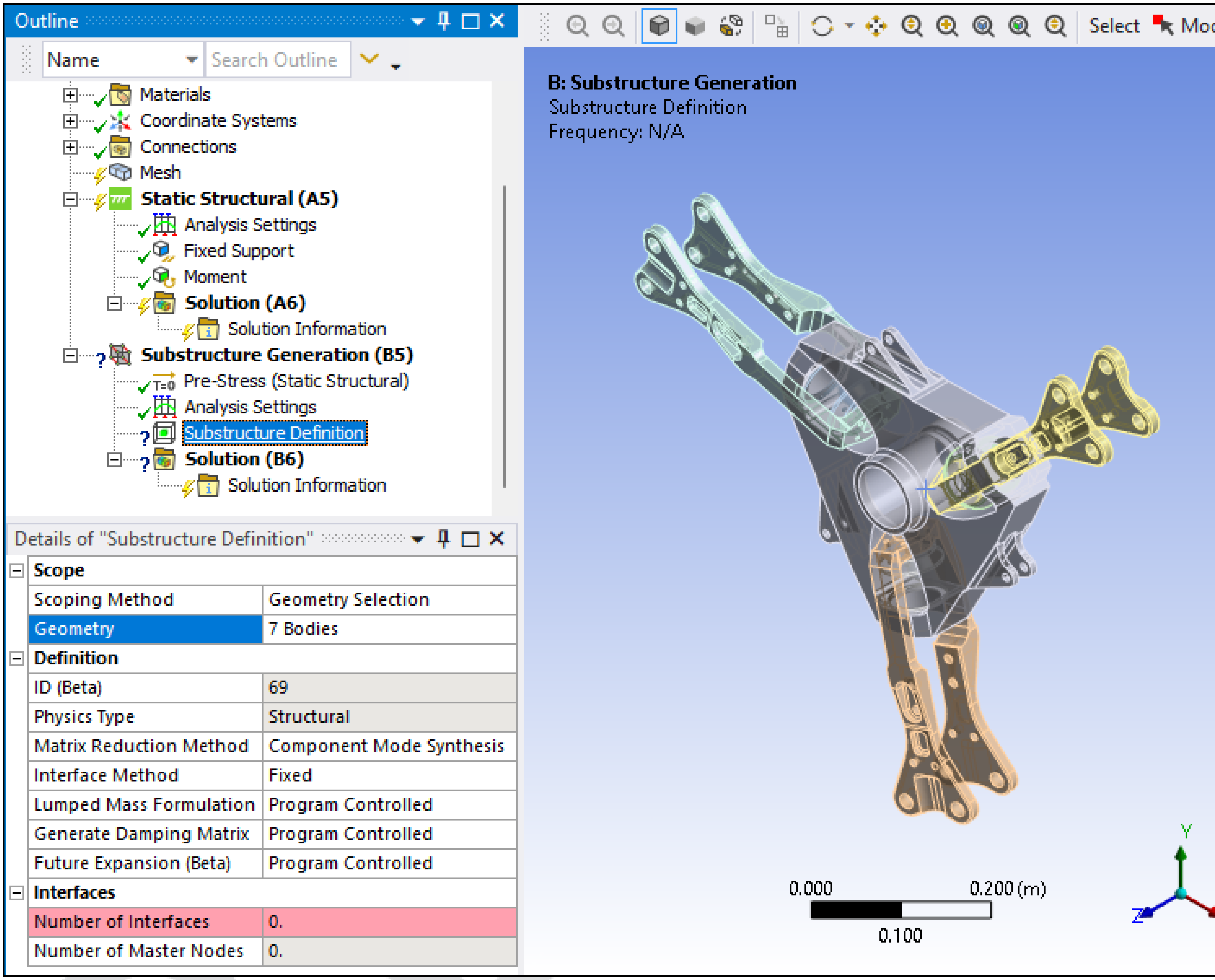
Here is an example of a pre-stressed **Substructure Generation** analysis without beta options active. All bodies are automatically scoped to the **Substructure Definitions** object.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

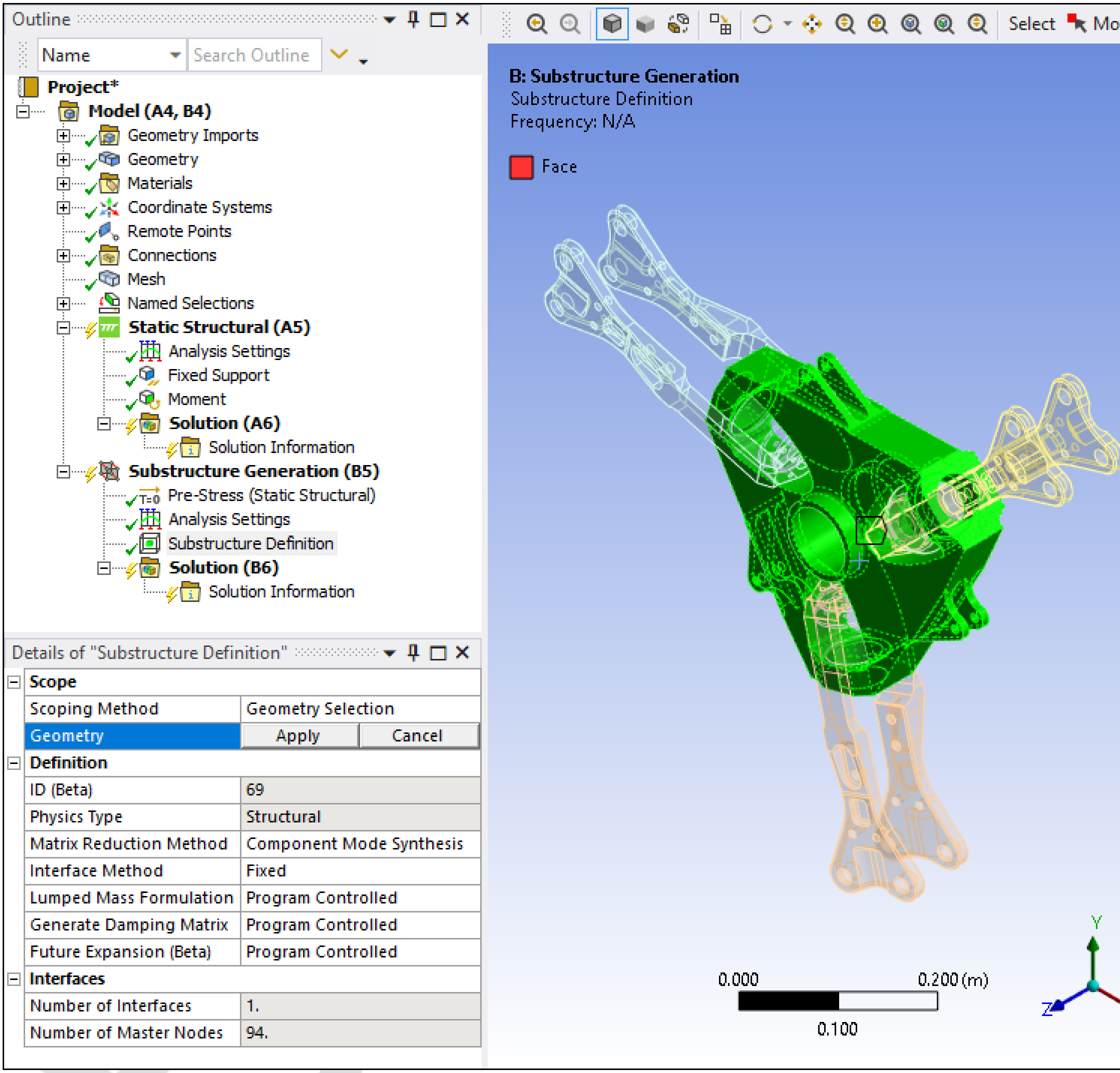


Once you activate the beta options, the ability to scope specific bodies becomes available.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

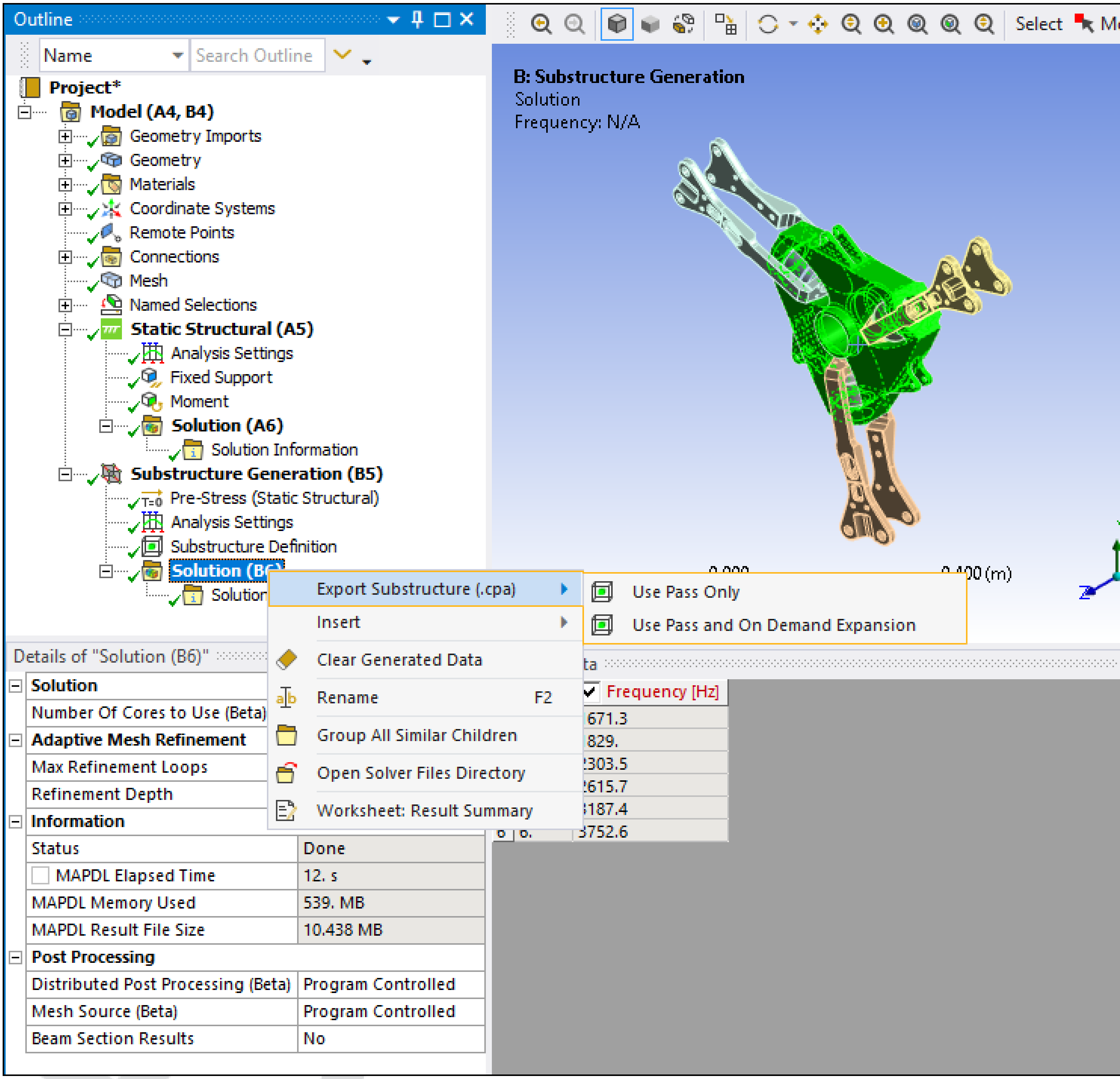


You can then specify a desired body.



And following the solution, you can export solved substructure for use in another analysis as an Imported Condensed Part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.


Outline
Name
Search Outline
Project*
Model (A4, B4)
Geometry Imports
Geometry
Materials
Coordinate Systems
Remote Points
Connections
Mesh
Named Selections
Static Structural (A5)
Analysis Settings
Fixed Support
Moment
Solution (A6)
Solution Information
Substructure Generation (B5)
Pre-Stress (Static Structural)
Analysis Settings
Substructure Definition
Solution (B6)
Select
B: Substructure Generation
Solution
Frequency: N/A
Export Substructure (.cpa)
Use Pass Only
Use Pass and On Demand Expansion
Insert
Clear Generated Data
Rename F2
Group All Similar Children
Open Solver Files Directory
Worksheet: Result Summary
Details of "Solution (B6)"
Solution
Number Of Cores to Use (Beta)
Adaptive Mesh Refinement
Max Refinement Loops
Refinement Depth
Information
Status Done
MAPDL Elapsed Time 12. s
MAPDL Memory Used 539. MB
MAPDL Result File Size 10.438 MB
Post Processing
Distributed Post Processing (Beta) Program Controlled
Mesh Source (Beta) Program Controlled
Beam Section Results No
Frequency [Hz]
671.3
829.
0.400 (m)

BETA

Chapter 7: Fracture

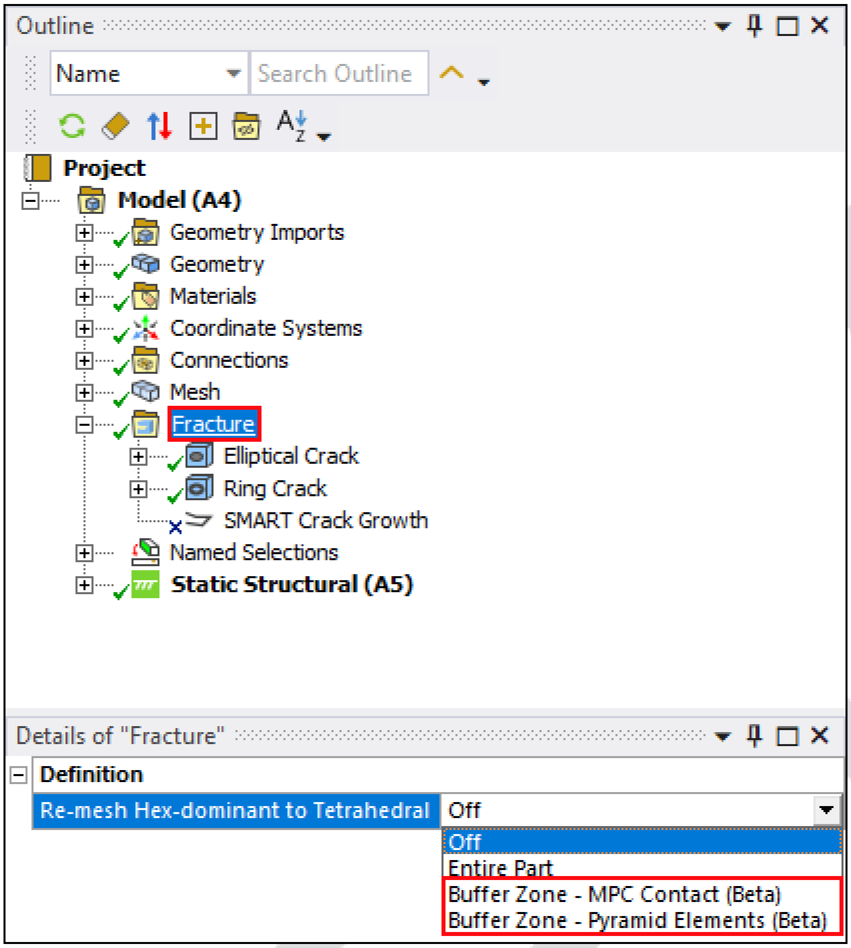
When Beta Options (p. 1) are active, the following features are available for Fracture analyses:

7.1. Crack Meshing - Re-mesh Hex-Dominant Mesh to Tetrahedrons in the Crack Buffer Zone

When Beta Options (p. 1) are active, the sole property of the Fracture object, **Re-Mesh Hex-dominant to Tetrahedral**, provides the following beta options:

- **Crack Buffer Zone - MPC Contact (Beta)**
- **Buffer Zone - Pyramid Elements (Beta)**

BETA



These options enable you to convert the hex-dominant base mesh to tetrahedrons within the crack buffer zone only, instead of converting the entire part associated with the crack's scoped bodies to tetrahedrons.

Important:

When you select either of these options, the size of the buffer zone should be large enough to accommodate the remeshing and crack propagation required for **SMART Crack Growth**. However, this size can vary. As needed, use the properties of the **Buffer Zone Scale Factor** category in the Details pane to enlarge the buffer zone. In the event your crack grows beyond the quadratic tetrahedral mesh buffer zone, the application will produce a re-meshing error during the solution.

Requirements/Limitations/Recommendations

Note the following for mesh conversion within the crack buffer zone:

- These settings apply only when the base mesh of the scoped body is hex-dominant.
- If you include **SMART Crack Growth**, and during the solution the associated re-meshing reaches the contact elements created at the crack buffer zone, the re-mesh process for the **SMART Crack Growth** could fail. Increasing the size of the buffer zone, through the **Buffer Zone Scale Factor** properties, should address the issue.

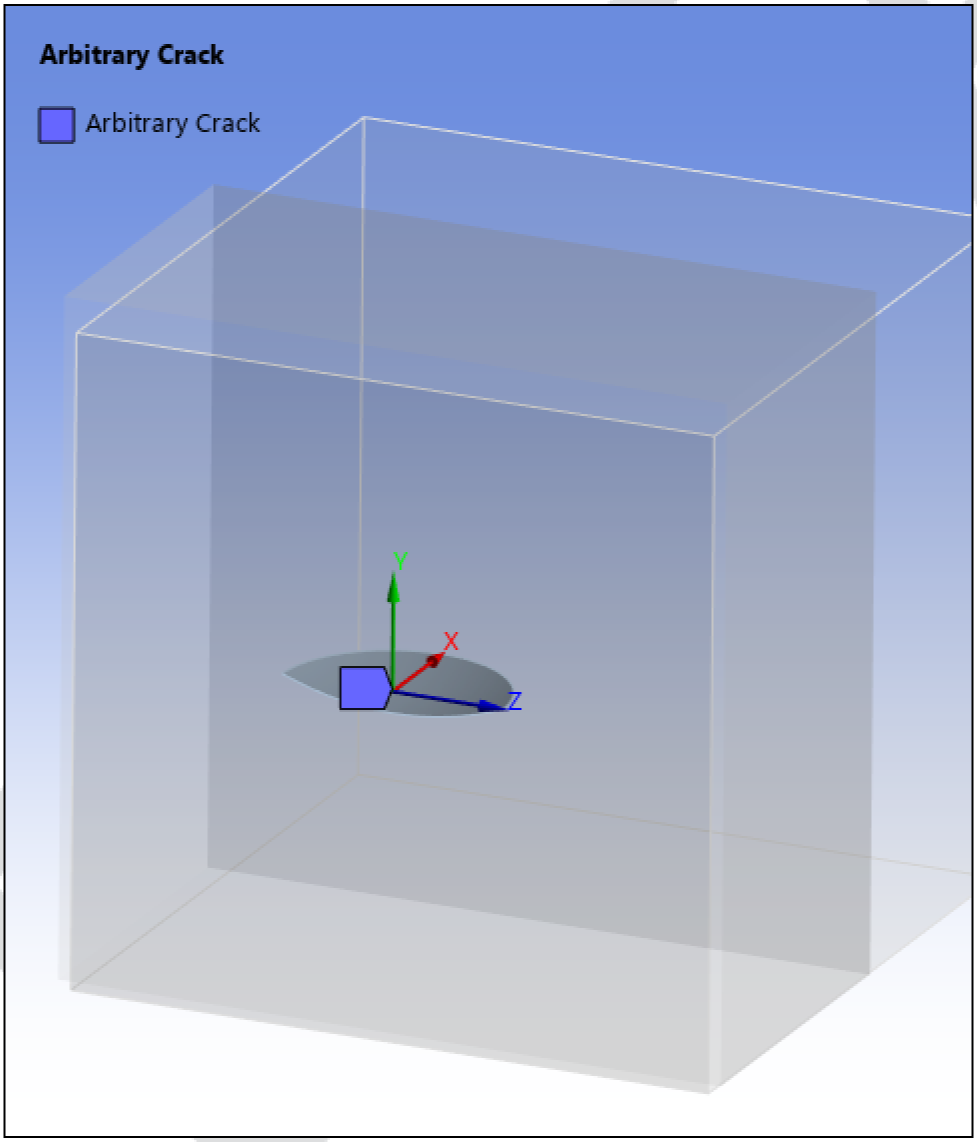
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

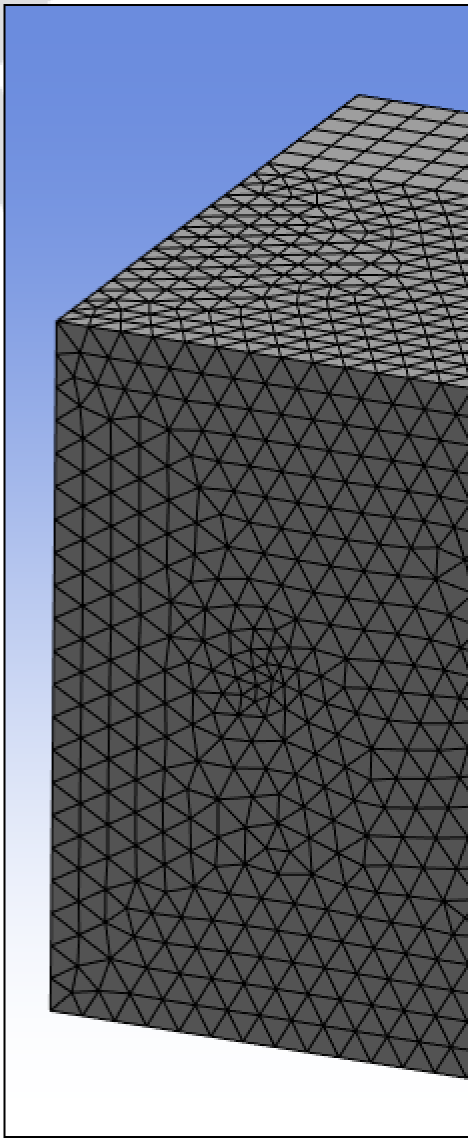
- You can use a **Body Sizing** mesh control specified as a **Sphere of Influence** to create a finer mesh around the crack.
- Adaptive mesh refinement is not supported. Set the **Use Adaptive Sizing** property (**Sizing** category) of the **Mesh** object to **No** for a successful crack mesh generation.

Example Usage

Arbitrary Crack

In the following example, the hex dominant base mesh inside of the crack buffer zone of an **Arbitrary Crack** is converted to Tetrahedrons using the **Buffer Zone - MPC Contact (Beta)** option.

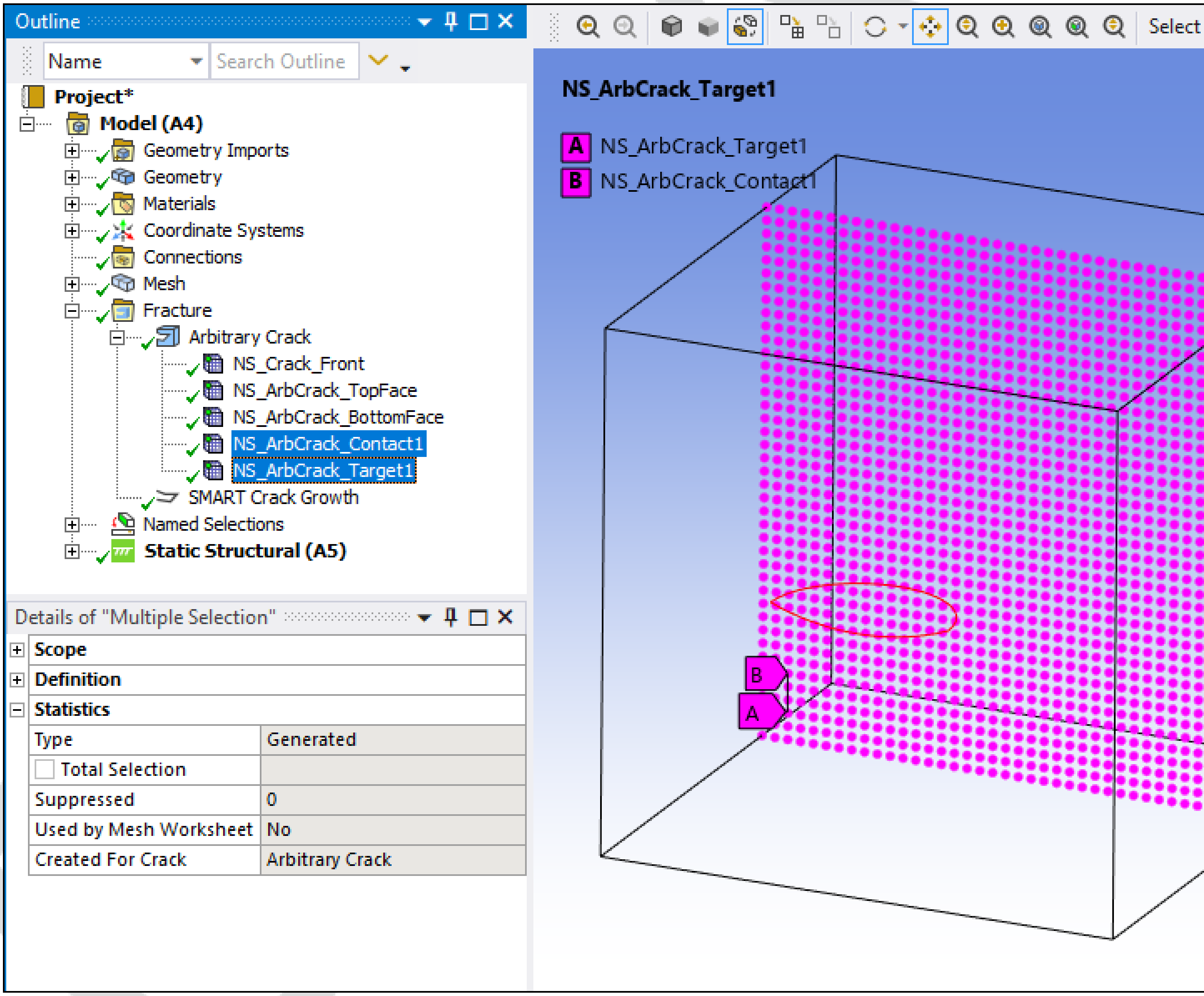




This enables the application to create an MPC bonded contact pair at the interface of the tetrahedrons and a hex-dominant mesh to hold the elements together during the solution.

Note:

There could be some localized stresses at the buffer zone mesh interface where contact is created.

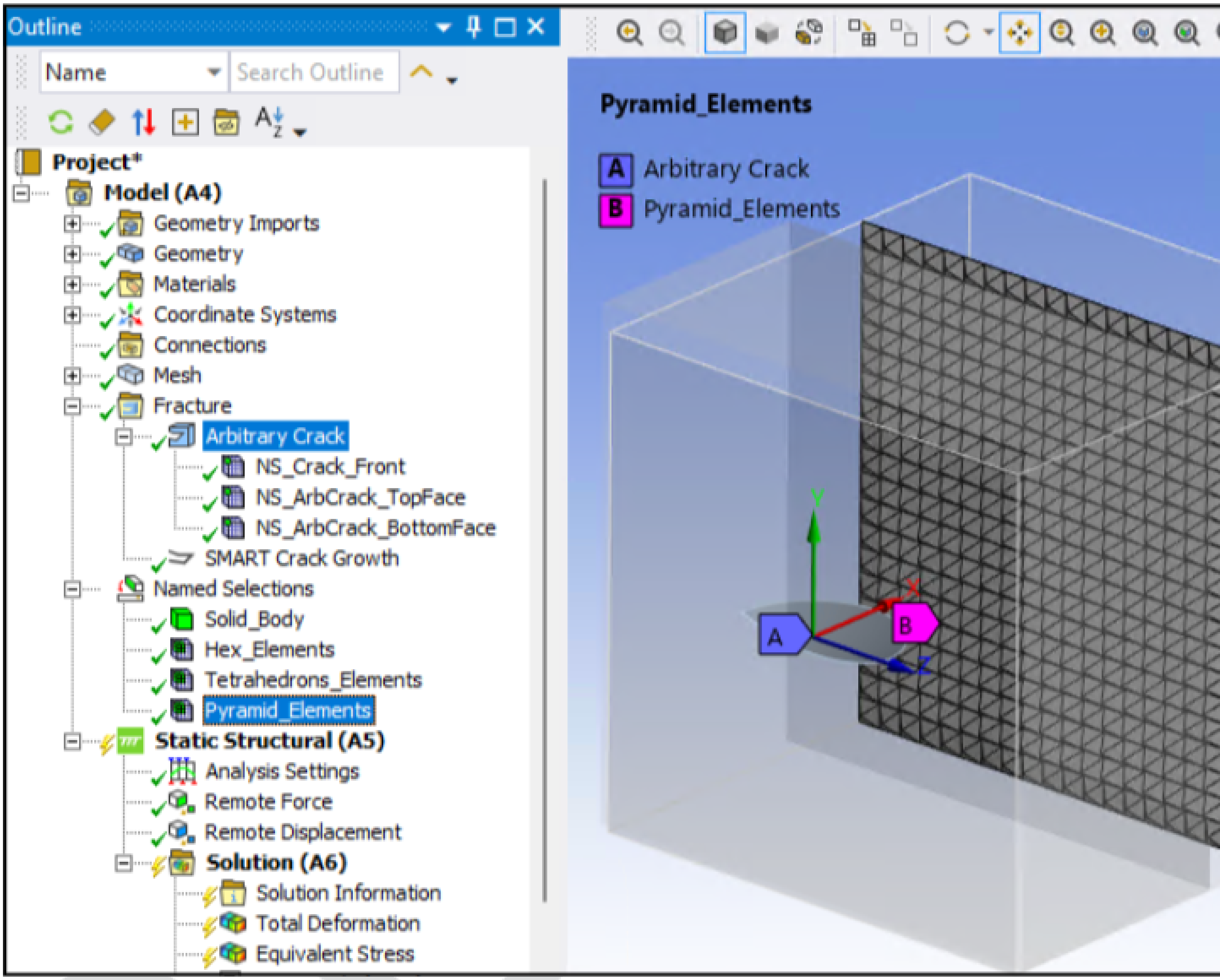


The following example illustrates how the **Buffer Zone – Pyramid Elements (Beta)** setting enables the application to create pyramid elements at the interface of the tetrahedrons and hex-dominant mesh to hold the elements together during the solution.

Note:

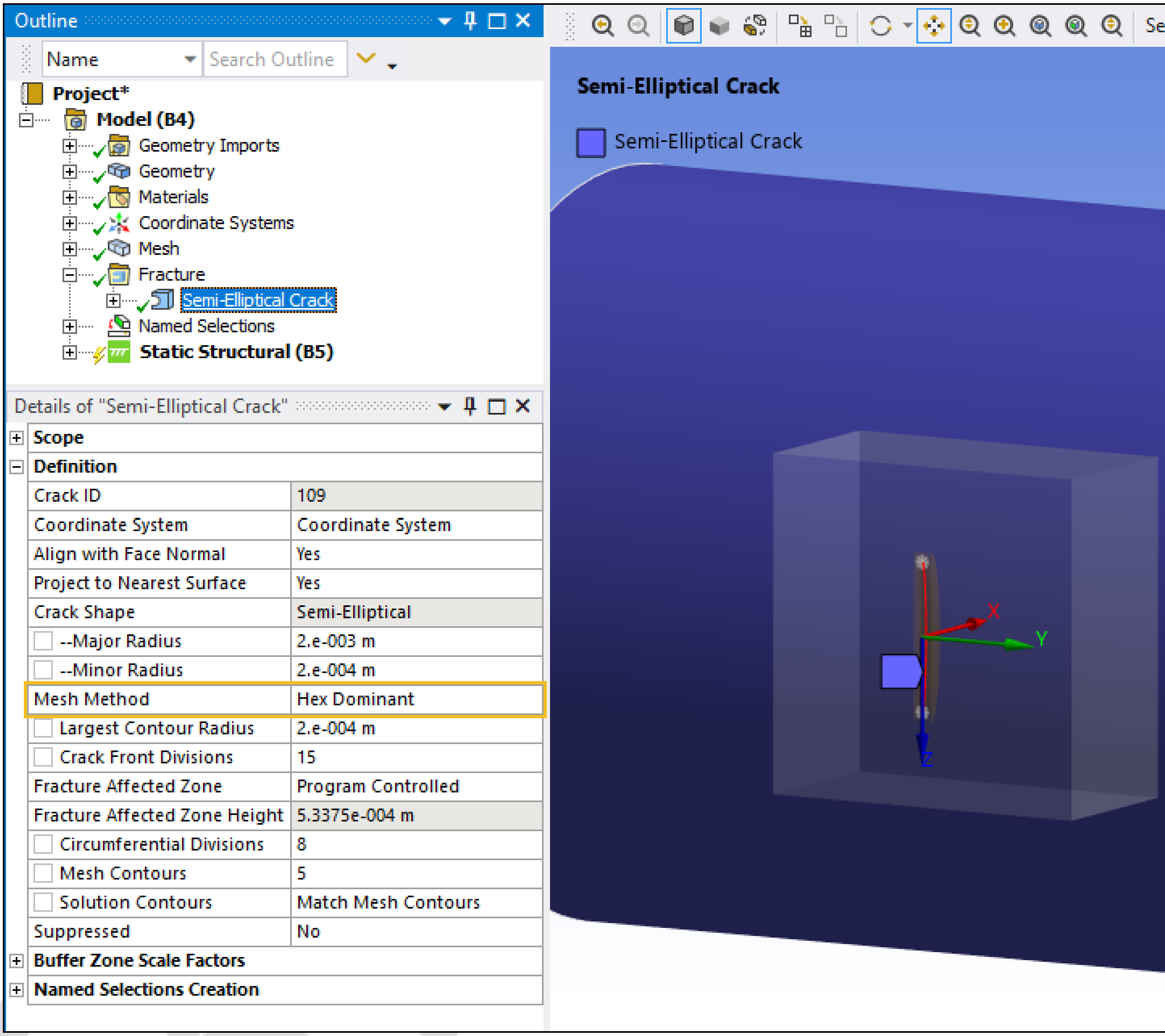
This option is not supported if the base mesh is a linear mesh.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



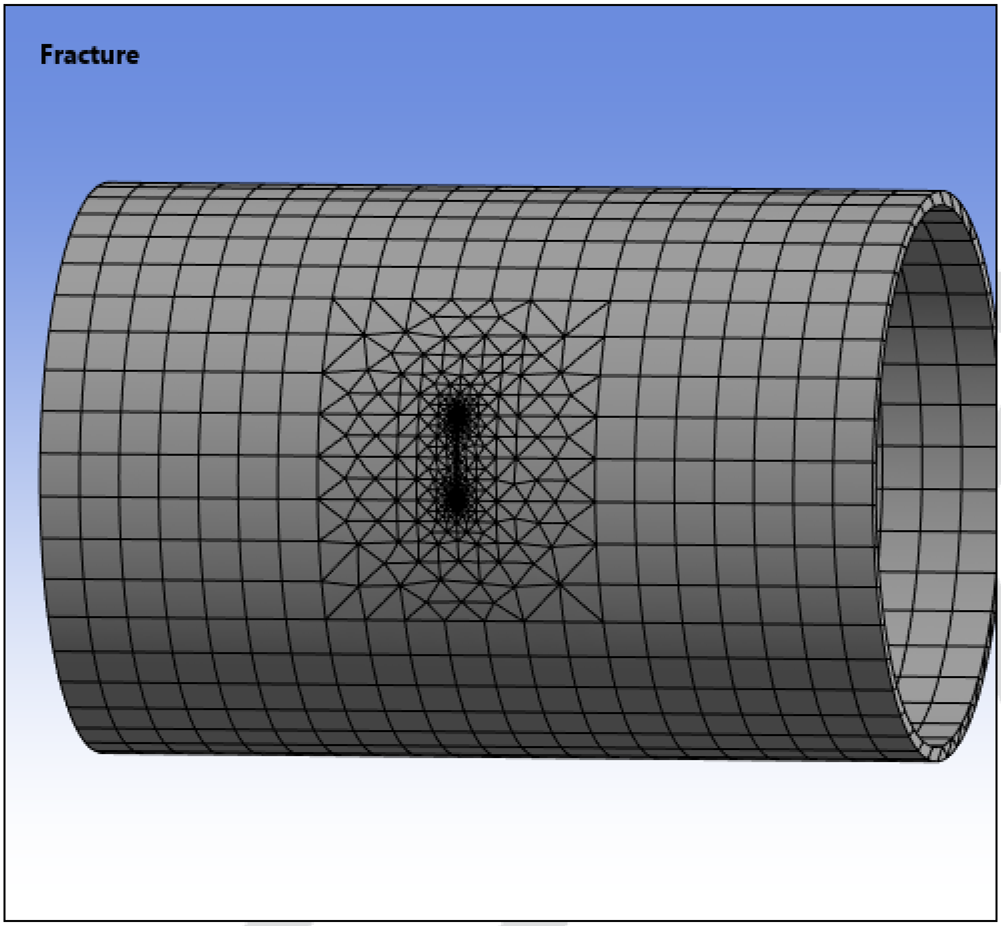
Semi-Elliptical Crack

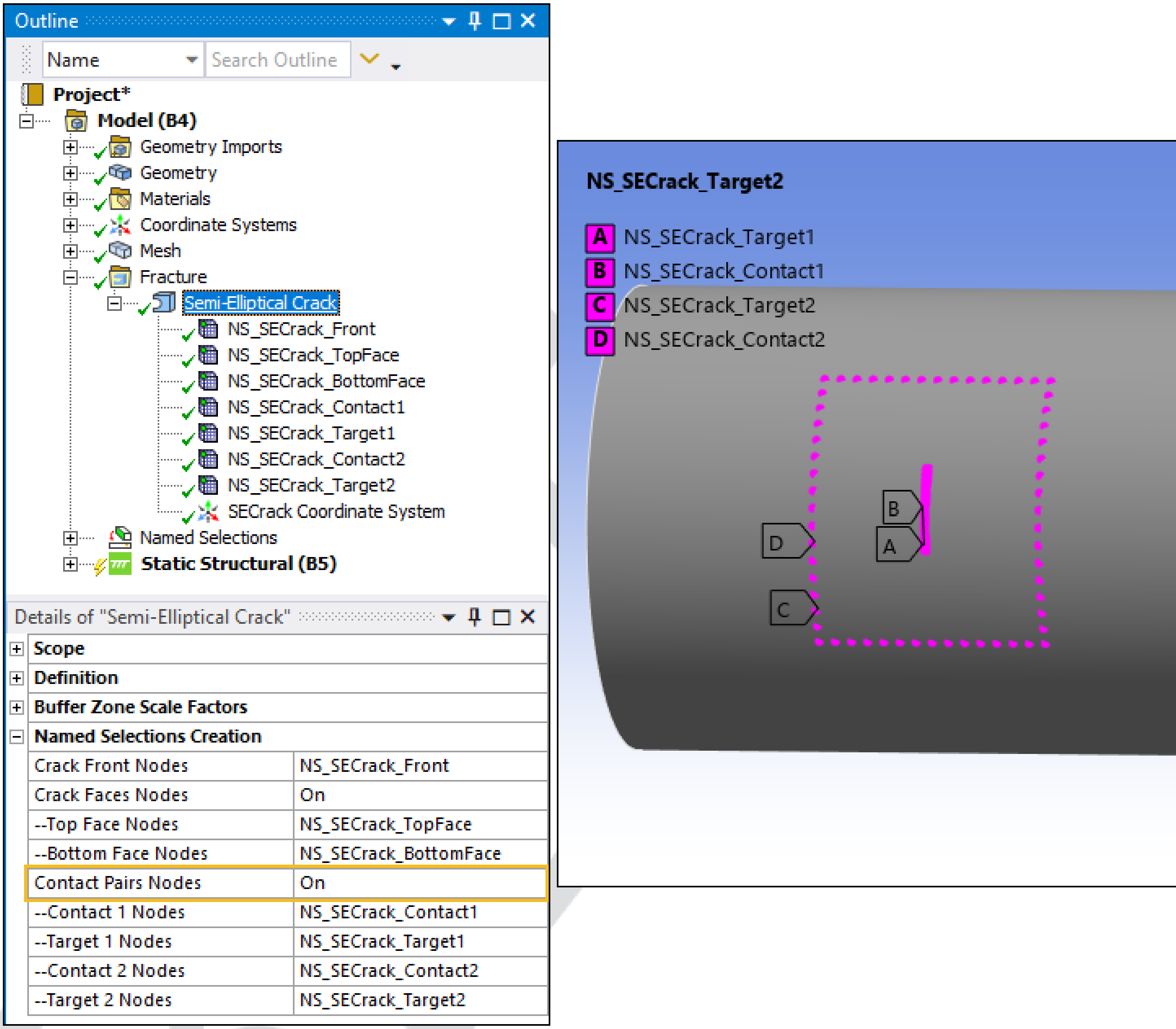
As illustrated for the following **Semi-Elliptical Crack**, for cracks that support the **Hex Dominant** setting for the **Mesh Method** property, the application creates two contact pairs. The first contact is at the **Fracture Affected Zone (FAZ)** interface near the crack and the second contact is at the crack buffer zone interface.



When you set the **Contact Pairs Nodes** property (of the **Named Selections Creation** category) to **On**, the application creates the contact pair target node and contact node components and automatically includes them under the crack object as shown here for the crack mesh and the contact pairs.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

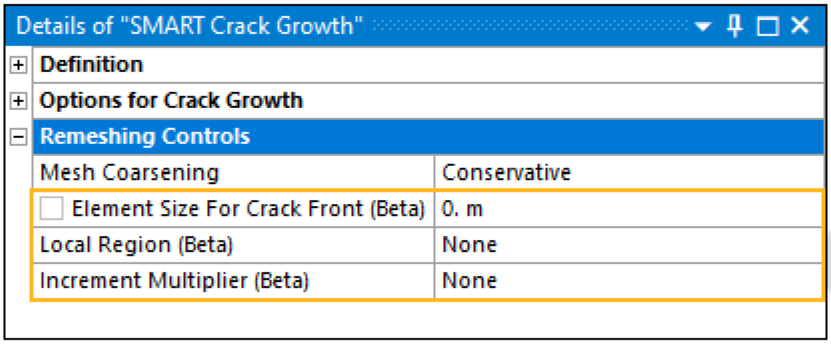

Fracture



7.2. SMART Crack Remeshing Controls

When Beta Options (p. 1) are active, the SMART Crack Growth object supports the following **Remeshing Controls** category properties:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



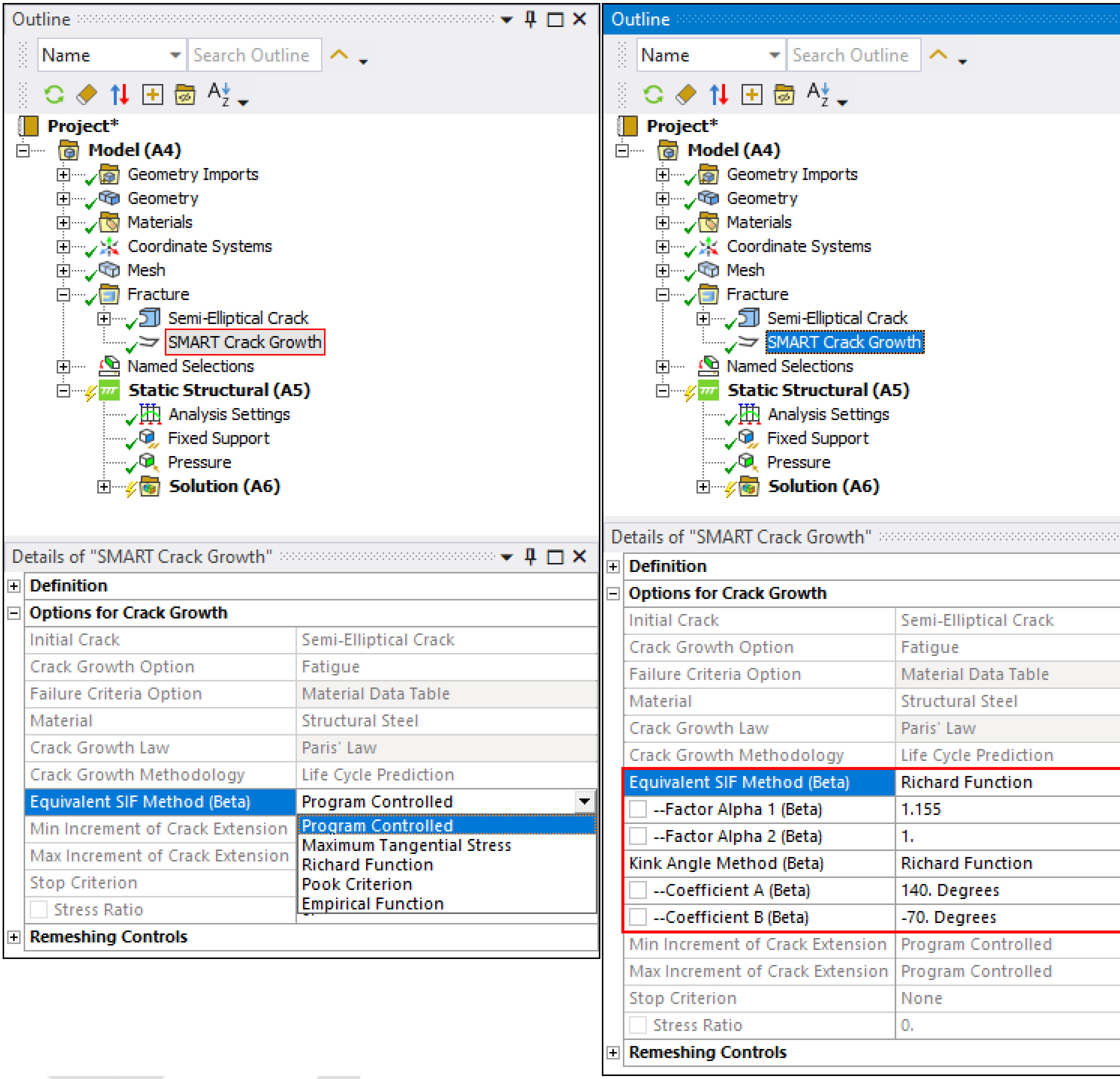
Property	Description
Element Size For Crack Front (Beta)	The default setting is `0.0` or you can specify a double value.
Local Region (Beta)	Options include **None** (default) and **Named Selection**. When you select Named Selection, the following additional properties display: • **Named Selection (Beta)**: Use this property to select a desired Named Selection from the drop-down menu. • **Element Size For Local Region**: The default setting is 0.0 or you can specify a double value.
Increment Multiplier (Beta)	Options include **None** (default), **Solution Time**, **Accumulated Max Crack Extension**, and **Number of Remeshing Steps**. When you select any option *other* than **None**, the following additional properties display: • **Max Multiplier (Beta)**: Enter a value greater than `0`. • **Start Value (Beta)**: Enter a value greater than or equal to `0`. • **End Value (Beta)**: Enter a value greater than `0`.

7.3. SMART Crack Growth - Mode III Stress Intensity Factor (SIF)

For mixed mode fracture problems, you must include the Mode III stress intensity factor- SIFS (K3), to calculate the **Equivalent SIF Factor (SIF)** and **Kink Angle**. Using beta options, Mechanical supports certain Equivalent SIF calculation methods. Specify the Equivalent SIF calculation method and related property definitions/options using the **Equivalent SIF Method** property of the SMART Crack Growth object. As illustrated below, the **Equivalent SIF Method (Beta)** property includes the following options:

- **Program Controlled** (default): The application uses the **Maximum Tangential Stress** calculation method.
- **Maximum Tangential Stress**: This method does not include Stress Intensity Factor K3 from Mode III fracture in the Equivalent SIFS calculation.
- **Richard Function**/**Pook Criterion**/**Empirical Function**: These methods consider the Stress Intensity Factor K3 from Mode III fracture in the Equivalent SIFS calculation.

BETA



Based on the method you select, define the following additional properties:

- **Factor Alpha 1**: Displayed when the Equivalent SIF Method (Beta) is set to Richard Function, or Empirical Function. Specify the positive multiplicative factor for K2 term in the Equivalent Stress Intensity Factor calculation function. The default value is 1.155. This property can be parameterized.

- **Factor Alpha 2**: Displayed when the **Equivalent SIF Method (Beta)** property is set to **Richard Function** or **Empirical Function**. Specify the positive multiplicative factor for K3 term in the Equivalent Stress Intensity Factor calculation function. The default value is 1. This property can be parameterized.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Kink Angle Method (Beta)**: Displayed when the **Equivalent SIF Method (Beta)** property is set to any option other than **Program Controlled**. Use this property to specify the method to be used for kink angle calculation. Options include **Maximum Tangential Stress** (default) and **Richard Function**. This **Richard Function** option is only supported for when the **Equivalent SIF Method (Beta)** property is set to **Richard Function** or **Empirical Function**.

- **Coefficient A**: Displayed when the **Kink Angle Method (Beta)** property is set to **Richard Function**. Specify the coefficient of the first order term in the **Richard Function** for the kink angle calculation. The default value is **140°**. The valid entry range for this property is -180° to 180°. This property can be parameterized.

- **Coefficient B**: Displayed when the **Kink Angle Method (Beta)** property is set to **Richard Function**. Specify the coefficient of the second order term in the **Richard Function** for the kink angle calculation. The default value is **-70°**. The valid range for this property is -180° to 180°. This property can be parameterized.

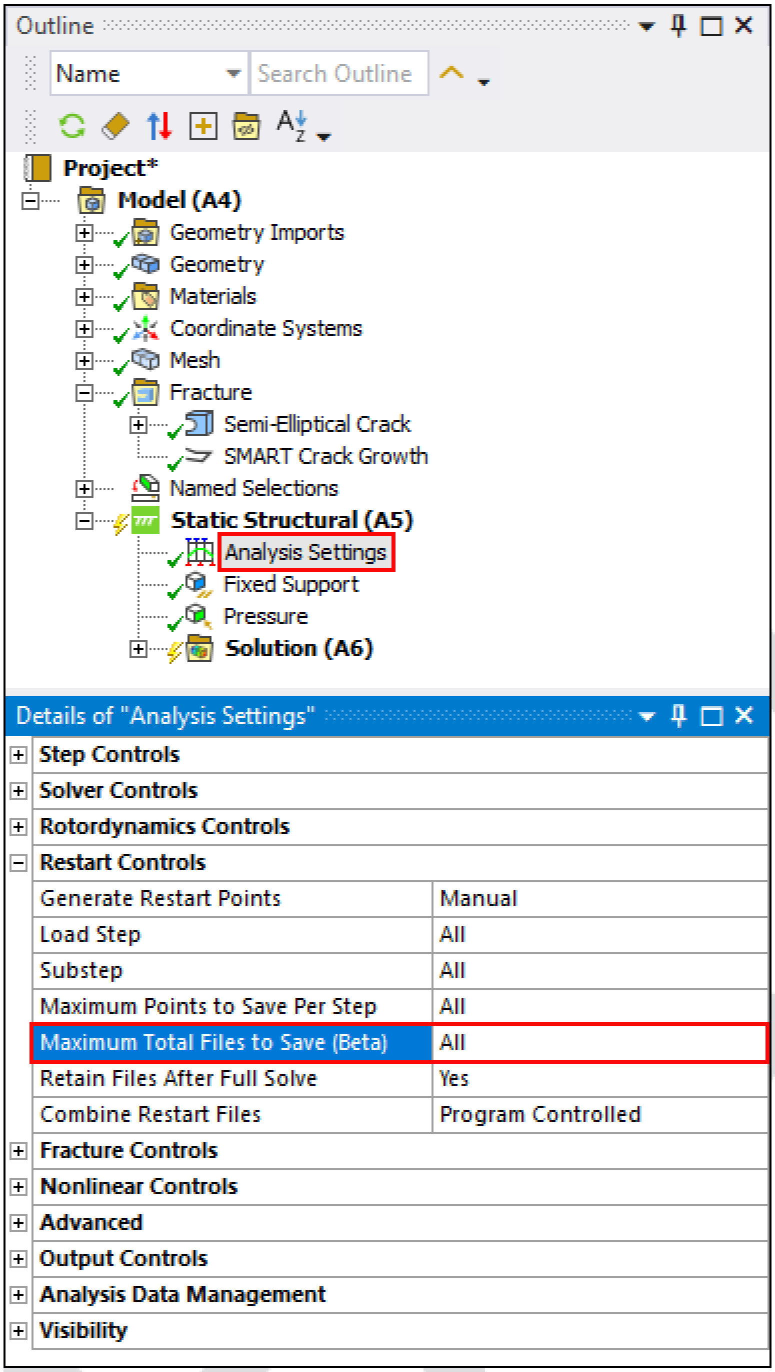
7.4. Saving for Restart Analysis During during SMART Crack Growth

The beta property, **Maximum Total Files to Save (Beta)**, is available in the **Restart Controls** category of the **Analysis Settings** object. This property displays when the **Generate Restart Points** property is set to **Manual** and **Substep** property is set to any option other than **Last**.

This property enables you to specify the maximum total number of restart files that can be saved during a restart analysis. An entry of **0** specifies the option **All**. The maximum number of restart files (`.xnnn`) that can be saved is **999**. The maximum number of remeshing database files (`.rdnn`) for restart analysis that includes a **SMART Crack Growth** object is **99**. To specify a value in the **Maximum Total Files to Save (Beta)** property, you must set the **Maximum Points to Save Per Step** to **All**.

When the specified maximum total number of files is exceeded, the application resets the restart file numbering back to 1 and continuous to write the restart (`.xnnn`) and remeshing database (`.rdnn`) files. The newest files are retained, and the oldest files are overwritten. The value specified applies to all subsequent load steps.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

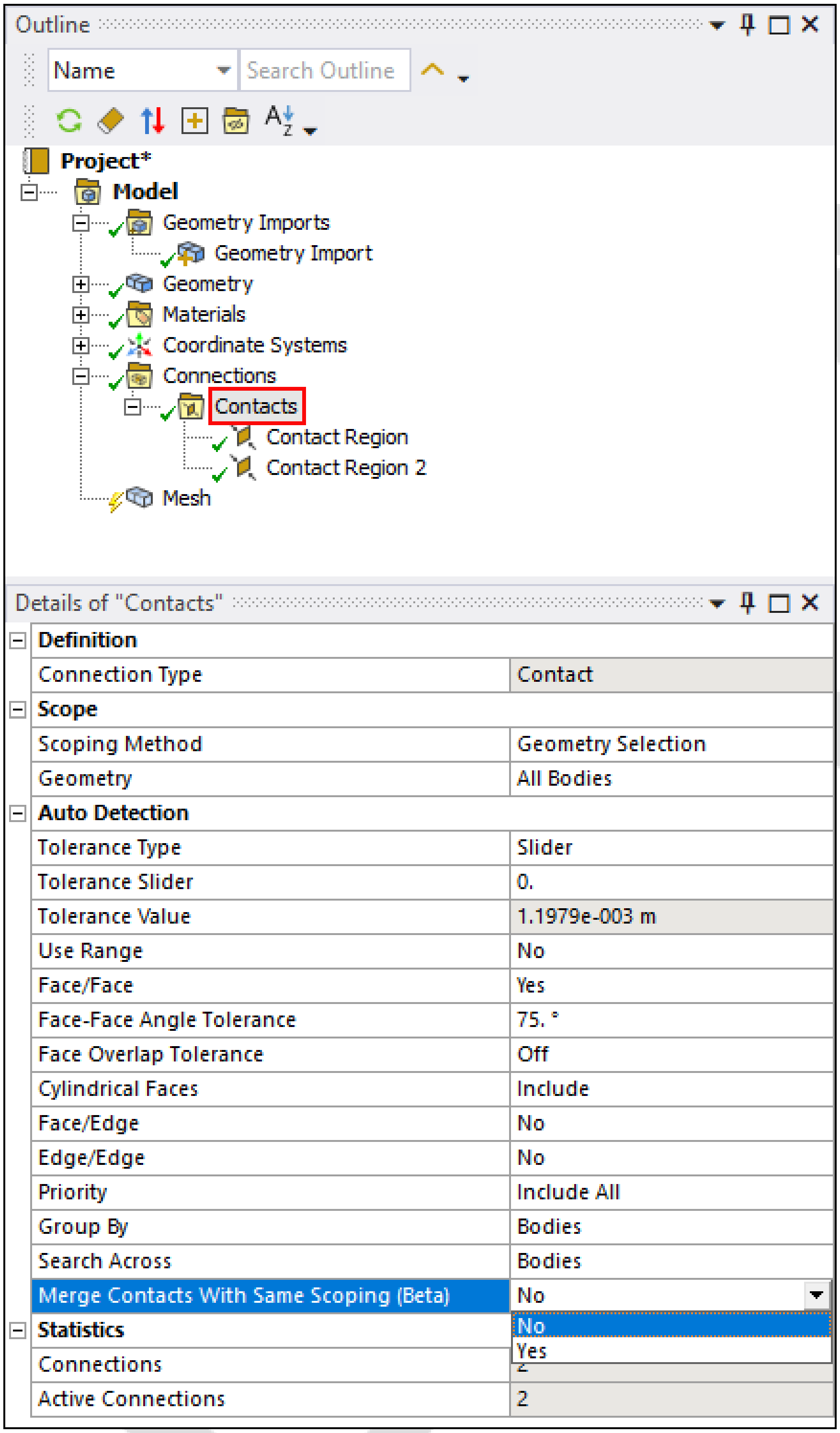


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 8: Merging Contacts with the Same Scoping

When Beta Options (p. 1) are active, the Contact/Connections folder includes the beta property, **Merge Contacts with Same Scoping (Beta)**. Options include **No** (default) and **Yes**. When set to **Yes**, the application automatically merges the already generated/to be generated bonded contacts that have the same scoping with other contacts, on either contact side or target side, when you select the context (right-click) menu option **Create Automatic Connections**.

BETA



Note:

- This feature only works for solid face to solid face contacts.
- You cannot merge contacts scoped to different geometry types like solids and shells.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify the Preference

This property has an associated preference with the same name, **Merge Contacts with Same Scoping (Beta)**, available in the Connections group of the Options dialog. Preference settings include **No** (default) and **Yes**. Setting the preference to **Yes** specifies a new default setting for the above property.

Note:

Changing the preference setting automatically updates the default setting of the property, even in an active session.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 9: Incorporating Contact Large Deflection

For large deflection analyses that include a **Contact Tool** under the Connections folder, this feature makes sure that the application incorporates large deflection at both the connection and analysis levels of your simulation.

Note:

As needed, see the Activate Beta Options (p. 1) section to make the feature available.

Insert a **Contact Tool** from a **Connections** folder. As shown, the Details category **Contact Tool Settings**, and the beta property, **Large Deflection**, become visible. Options for this property include **No** (default) and **Yes**.

When the **Large Deflection** property in the Solver Controls of the **Analysis Settings** is set to **Yes** when this property is also set to **Yes**, the application uses the same program controlled defaults for contacts results under **Contact Tool** as the overall analysis (based on the Solver Controls).

BETA

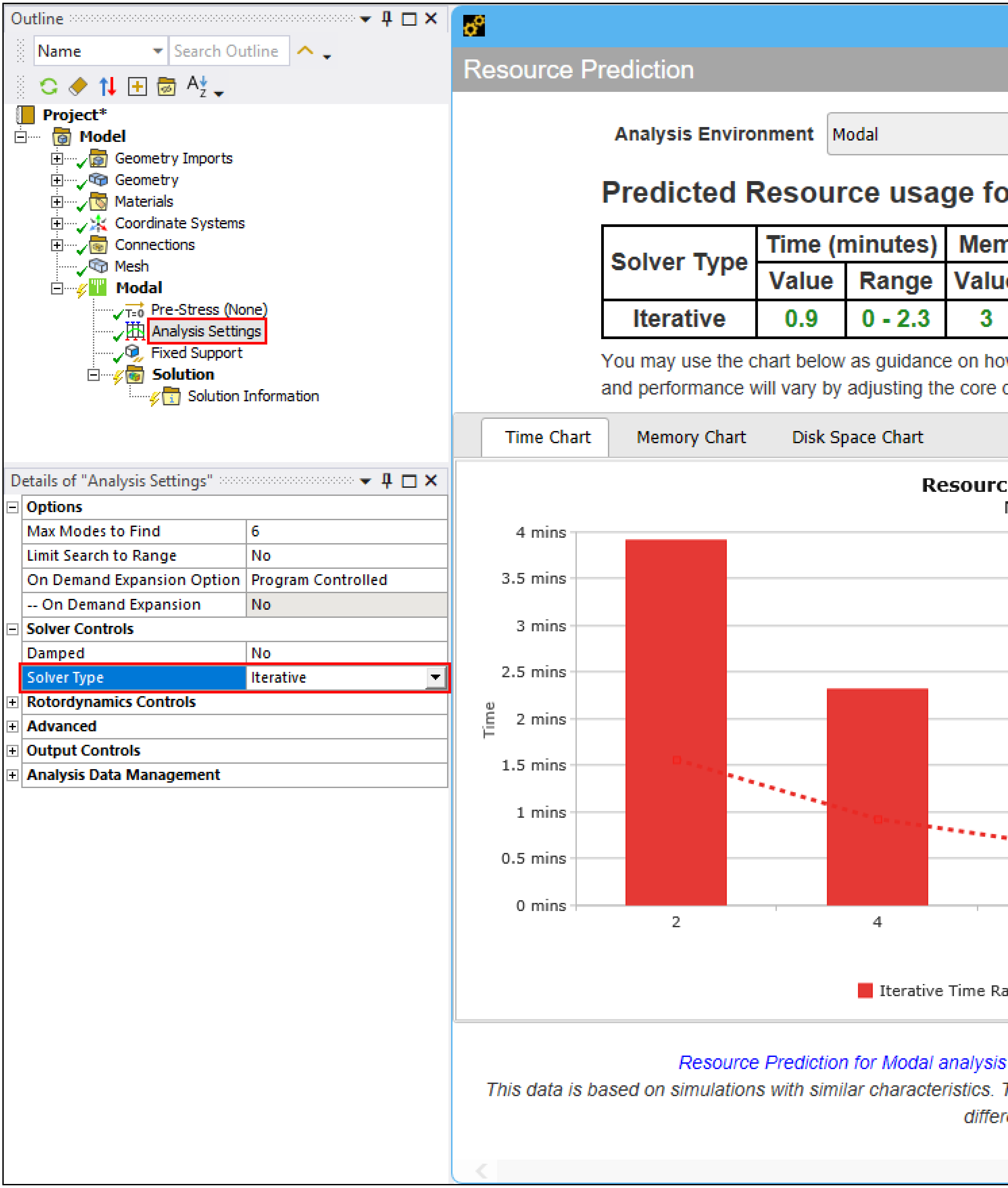
BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 10: Resource Prediction Support for Modal Analysis Iterative Solver

When Beta Options (p. 1) are active, the Resource Prediction feature supports stand-alone Modal analyses using the **Iterative** setting for the Solver Type property. You can set the property manually or it is default selection for the **Program Controlled** option. This setting enables you to estimate the required computational time, memory usage, and disk space for the analysis.

BETA



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 11: Disable Optimized File Processing for External Model Systems

By default, the application organizes and processes data imported from External Data systems using optimized methods to improve performance when importing Abaqus, LS-DYNA, and Mechanical APDL files. As desired, you can use beta options to disable this advanced processing capability and instead use legacy methods.

Note:

This feature does not currently support all source file commands. Some commands may be skipped.

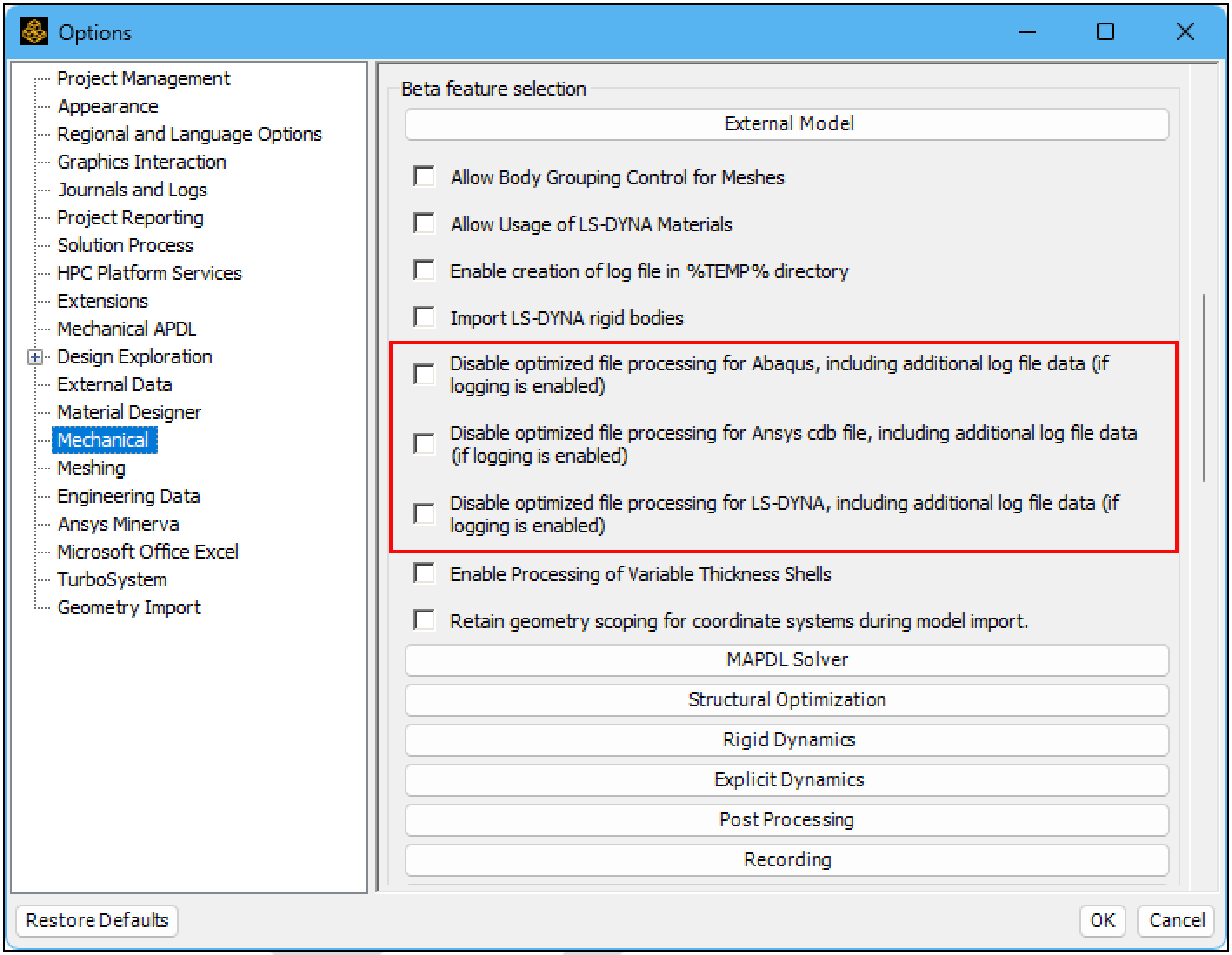
Activate Beta Key and Option

To make the feature available, you must activate two beta options.

1. On the Workbench Project page, select the **Tools** menu and then **Options**. Select the **Appearance** category, scroll down the dialog, select Beta Options (p. 1), and then select **OK**. You need to select **Beta Options** and then close the dialog for the next options to become available.

2. Open the **Tools** > **Options** dialog again and select the **Mechanical** category. Scroll down the dialog to the **External Model** category and select the check box for one or more of the following options based on file type:

BETA



3. Select **OK**. As prompted, close and reopen the Ansys® Workbench™ application.

Log File

An associated property is available for these beta options: **Enable creation of log file %TEMP% directory**, as shown above. When you enable this option, the application automatically creates a detailed log file for each command imported into Mechanical as well as any errors that occur.

Limitations

Note the following:

- For Abaqus, only free format files are supported.
- The **No Grouping** option of the **Body Grouping** property (**Model** cell of Mechanical System > **Mesh Conversion Options**) is not supported.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 12: External Model Browser

The **External Model Browser (Beta)** tool enables you to view the processed and unprocessed keywords for LS-DYNA or ABAQUS input files that you import into Mechanical using an **External Model** system from Workbench or the **Add Model Import** option in Mechanical. From the browser tree, you can select individual keywords and perform actions such as previewing the properties associated with the selected keyword, adding the keyword to the current analysis (LS-DYNA), or adding a Python snippet (ABAQUS).

Important:

This documentation assumes that you know how to import the above file types. See the steps to import mesh files from either Workbench or Mechanical in the Attach Geometry/Mesh section of the *Mechanical User's Guide*.

Go to a section topic:

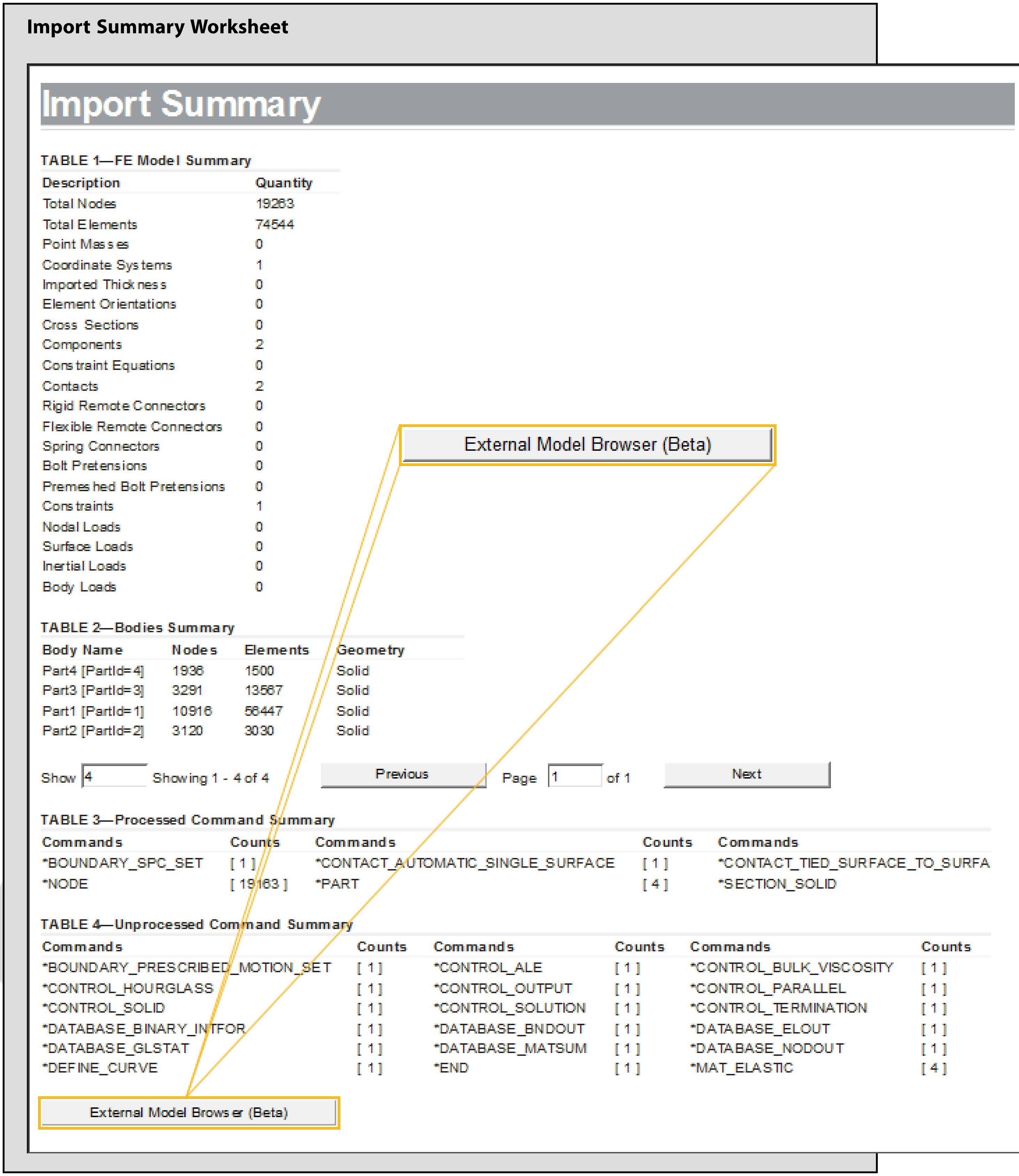
12.1. Displaying the External Model Browser

Once you have turned on Beta Options (p. 1) and imported your file, you display the **External Model Browser (Beta)** by selecting the:

- Import Summary object and then select the **External Model Browser (Beta)** option at the bottom of the page.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Summary Worksheet



Import Summary

TABLE 1—FE Model Summary

Description	Quantity
Total Nodes	19263
Total Elements	74544
Point Masses	0
Coordinate Systems	1
Imported Thickness	0
Element Orientations	0
Cross Sections	0
Components	2
Constraint Equations	0
Contacts	2
Rigid Remote Connectors	0
Flexible Remote Connectors	0
Spring Connectors	0
Bolt Pretensions	0
Premeshed Bolt Pretensions	0
Constraints	1
Nodal Loads	0
Surface Loads	0
Inertial Loads	0
Body Loads	0

TABLE 2—Bodies Summary

Body Name	Nodes	Elements	Geometry
Part4 [PartId=4]	1936	1500	Solid
Part3 [PartId=3]	3291	13567	Solid
Part1 [PartId=1]	10916	56447	Solid
Part2 [PartId=2]	3120	3030	Solid

Show 4 Showing 1 - 4 of 4 Previous Page 1 of 1 Next

TABLE 3—Processed Command Summary

Commands	Counts	Commands	Counts	Commands
*BOUNDARY_SPC_SET	[1]	*CONTACT_AUTOMATIC_SINGLE_SURFACE	[1]	*CONTACT_TIED_SURFACE_TO_SURFA
*NODE	[19163]	*PART	[4]	*SECTION_SOLID

TABLE 4—Unprocessed Command Summary

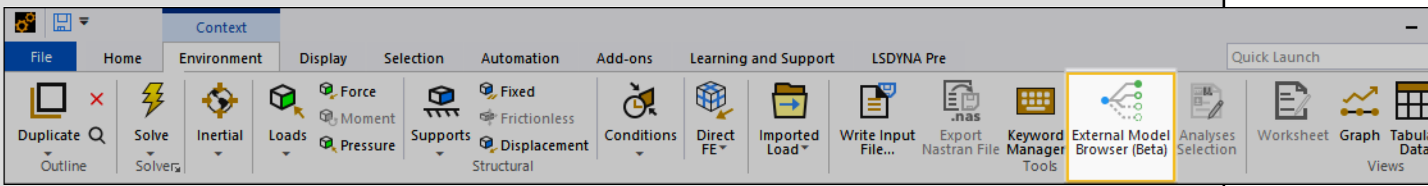
Commands	Counts	Commands	Counts	Commands	Counts
*BOUNDARY_PRESCRIBED_MOTION_SET	[1]	*CONTROL_ALE	[1]	*CONTROL_BULK_VISCOSITY	[1]
*CONTROL_HOURGLASS	[1]	*CONTROL_OUTPUT	[1]	*CONTROL_PARALLEL	[1]
*CONTROL_SOLID	[1]	*CONTROL_SOLUTION	[1]	*CONTROL_TERMINATION	[1]
*DATABASE_BINARY_INTFOR	[1]	*DATABASE_BNDOUT	[1]	*DATABASE_ELOUT	[1]
*DATABASE_GLSTAT	[1]	*DATABASE_MATSUM	[1]	*DATABASE_NODOUT	[1]
*DEFINE_CURVE	[1]	*END	[1]	*MAT_ELASTIC	[4]

External Model Browser (Beta)

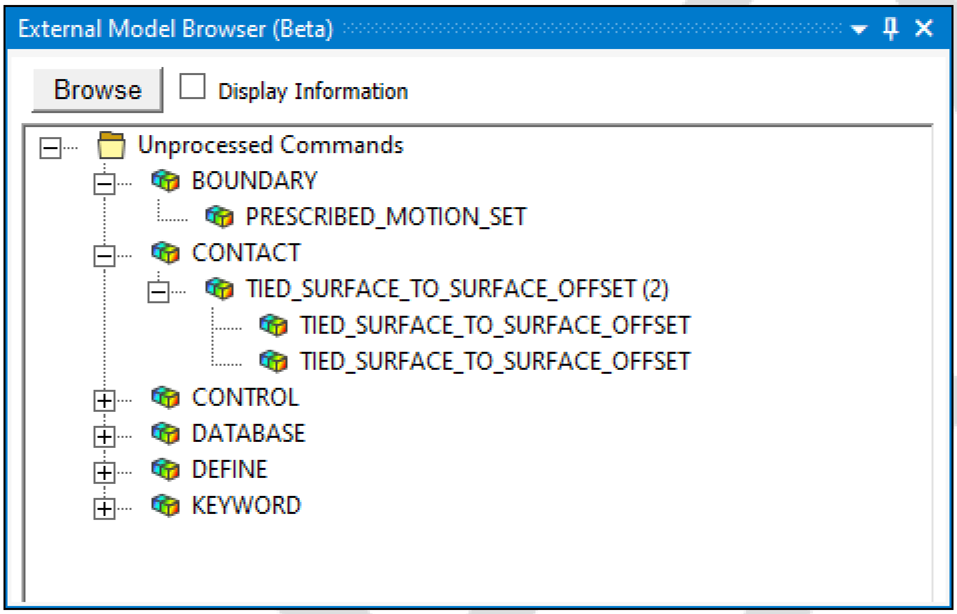
- **External Model Browser (Beta)** option on the Environment Context toolbar.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Environment Context Toolbar



Here is an example of the browser for a LS-DYNA input file.



Note:

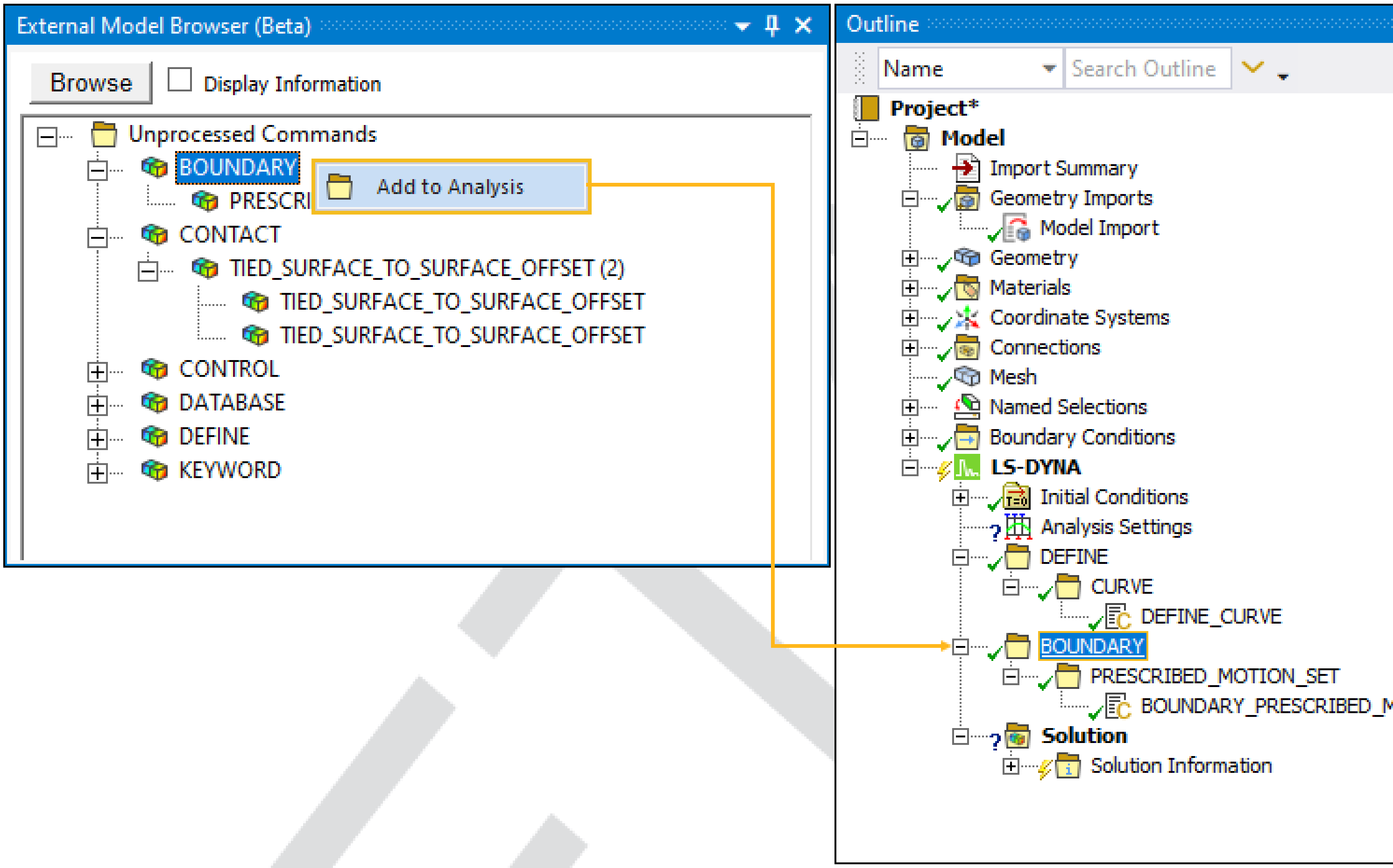
- You can load any input file (LS-DYNA or ABAQUS) using the Browse button for visualization purposes. However, to perform actions like adding keywords to the current analysis, the input file must have been imported using an **External Model** system from Workbench or the **Add Model Import** option in Mechanical.
- If a top-level node is selected, the options to insert keywords or command snippet apply to every keyword under the selected node.

12.2. Working with LS-DYNA Keywords

When importing an LS-DYNA input file, only the unprocessed keywords are shown. You have the option to add the keyword to the current analysis (or create an LS-DYNA analysis if there isn't one).

To add an unprocessed keyword to the project tree, right-click the keyword and select *Add to Analysis*. If there is no LS-DYNA analysis in the project, one is added to the project the first time you try to add a keyword. The keyword is added using the properties associated with it. Right-click a keyword and select Properties to see the properties associated with that keyword.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

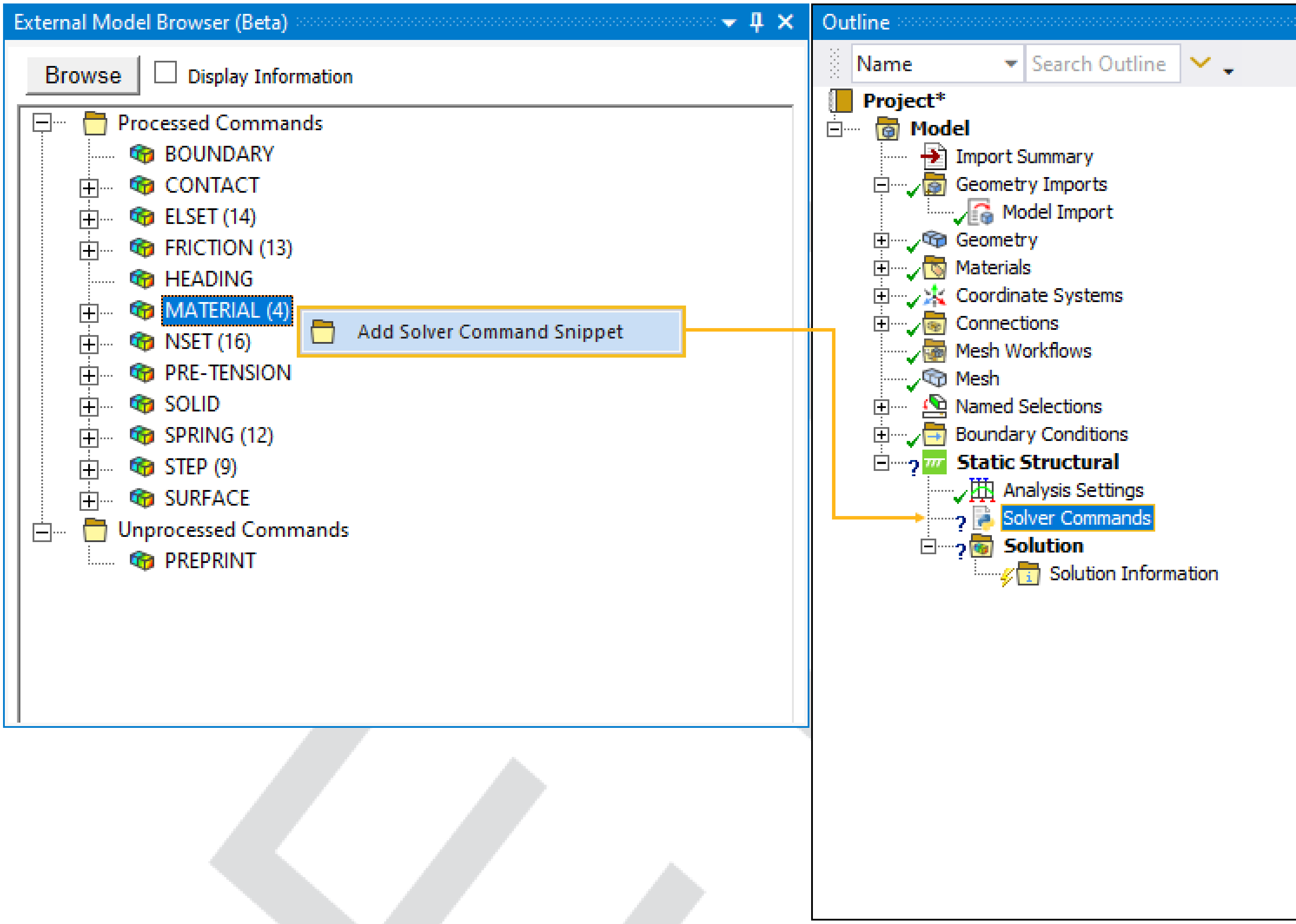


12.3. Working with ABAQUS Keywords

When importing an ABAQUS file, the External Model Browser will show both processed and unprocessed keywords. You have the option to generate a Python snippet (**Solver Commands** object) for any keywords using the *Add Solver Command Snippet* option.

To add a keyword to the project, right-click the keyword and select *Add Solver Command Snippet*. If there is no analysis in the project, a Static Structural analysis is added to the project. The snippet is added to the **Solver Commands** object using the properties associated with the selected keyword. A new **Solver Commands** object is created under the current analysis to write these snippets if one doesn't exist already.

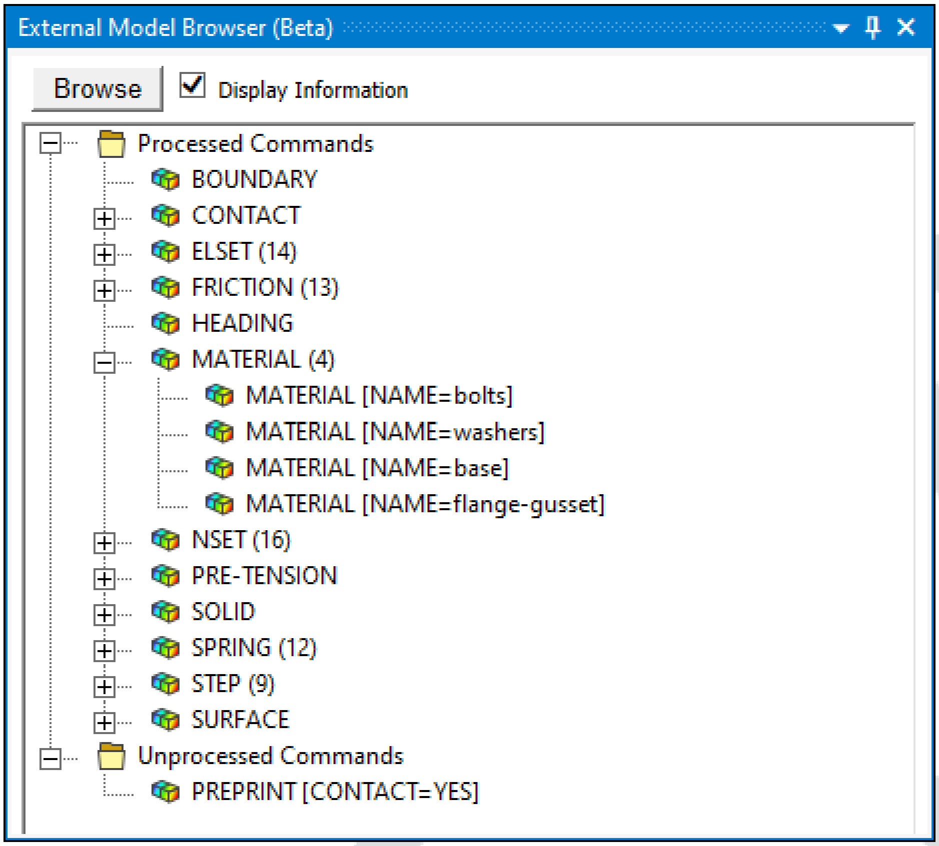
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



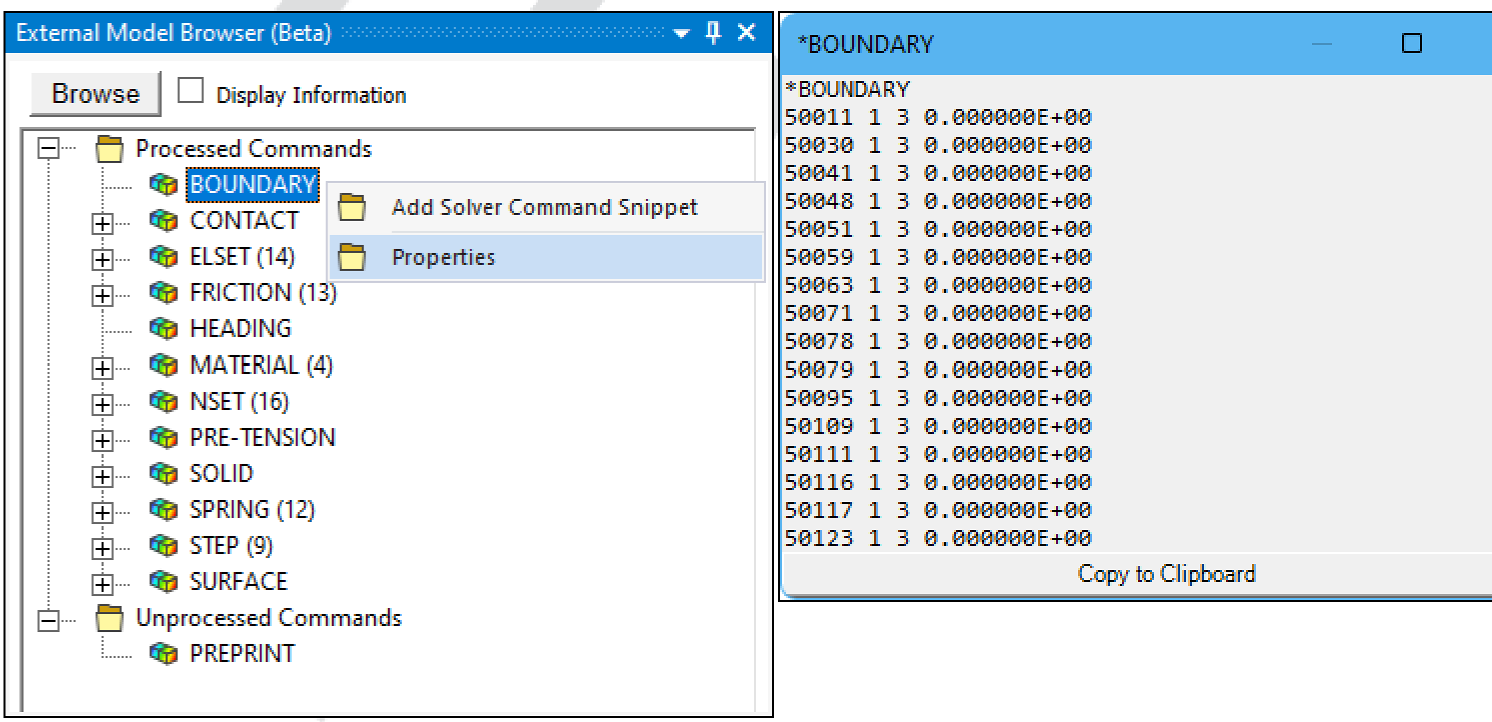
Additional Options

If the **Display Information** box is checked, additional information is displayed for the keywords. This is especially useful to discern between multiple instances of the same keyword.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Right-click a keyword and select **Properties** to see the properties associated with that keyword.

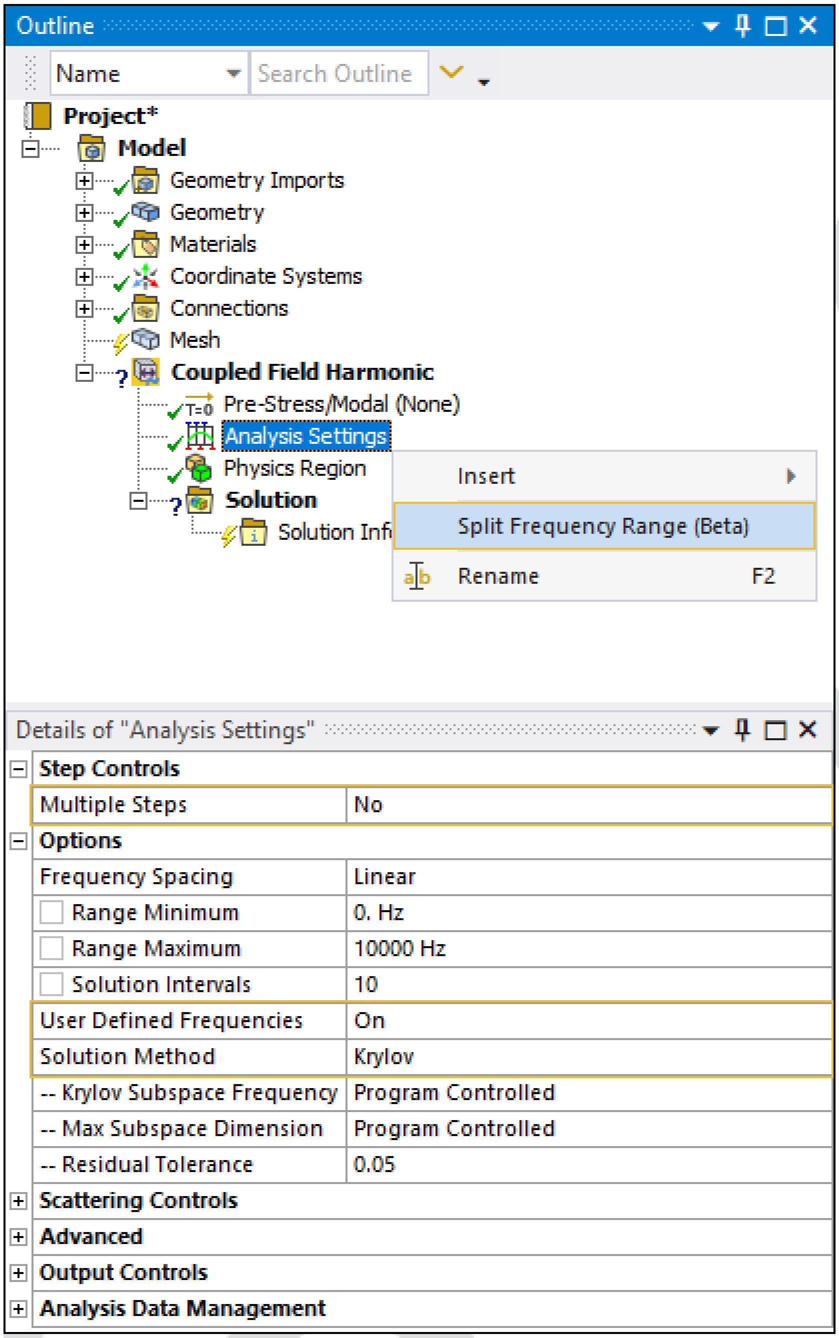


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 13: Improving Harmonic Solutions using Split Frequencies

When you activate Beta Options (p. 1) for a solved pure acoustic Coupled Field Harmonic, pure acoustic Harmonic Acoustics, or a Harmonic Response analysis that has the Solution Method property set to **Krylov**, the Analysis Settings object provides a new context (right-click) menu option: **Split Frequency Range (Beta)**. This option enables you to achieve a more accurate solution, however, it requires additional processing time and also creates additional frequency points.

BETA



Go to a section topic:

- Background (p. 50)
- Required Settings (p. 51)
- Application Example (p. 51)

Background

When using the **Krylov** solution method, the application builds a Krylov Subspace at the middle frequency of a given range (by default). Therefore, when you evaluate a **Krylov Residual Norm Frequency Re-**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sponse result, the result values will be less accurate at frequency points that are farther away from the Krylov Subspace frequency. That is, the wider the frequency range, the less accurate the result becomes.

To improve accuracy, the **Split Frequency Range (Beta)** option requires that you specify a frequency range of not more than 1000 Hz.

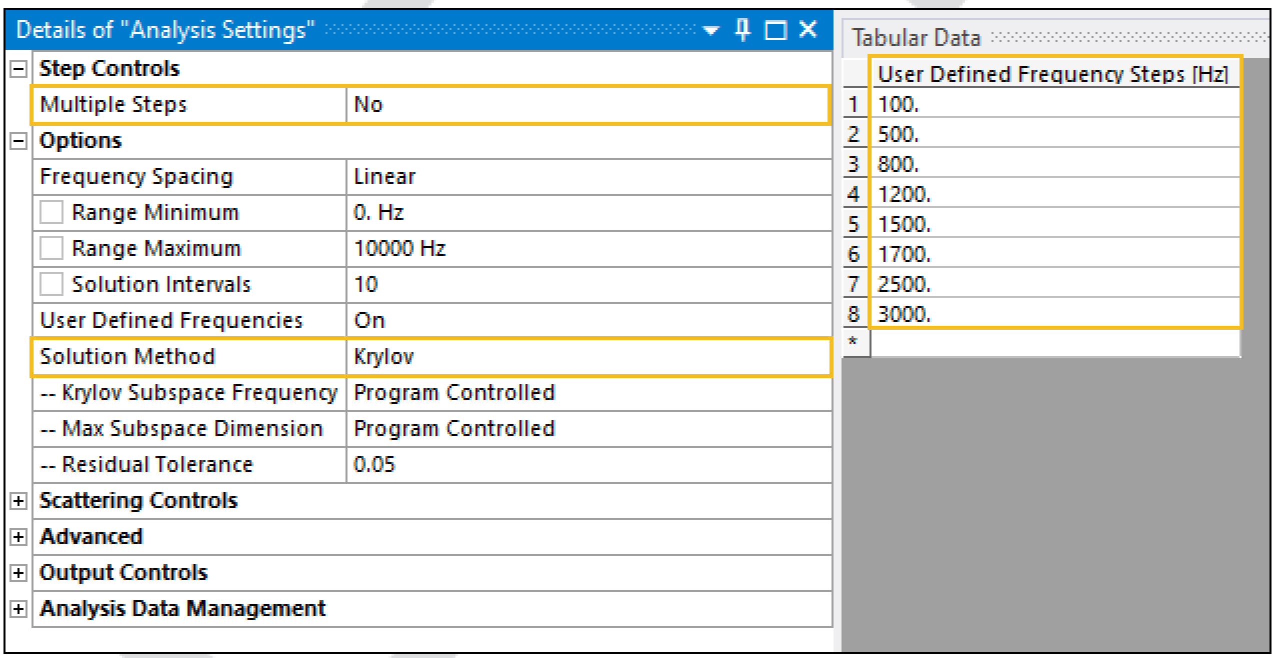
Required Settings

In order to make the **Split Frequency Range (Beta)** option available, you must first specify the following **Analysis Settings** properties:

1. Set the **Multiple Steps** property of the **Step Controls** category is set to **No**.
2. Set the **Solution Method** property is set to **Krylov**.
3. Set the **User Defined Frequencies** property is set to **On**.
4. Specify frequency values in the **User Defined Frequencies Steps** table in the **Tabular Data** window. You must specify a range greater than 1000 Hz.

Application Example

Consider the following example set up:



The user defined frequency range is 2900 Hz. That is, 3000 Hz – 100 Hz..

As illustrated below, once you select the **Split Frequency Range (Beta)** option, the application will automatically change the following **Analysis Settings**:

1. The **Multiple Steps** property is set to **Yes**.
2. The **Multiple Step Type** property is set to **Load Step**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The **Number of Steps** property is set to the upper bound of the user defined frequency range divided by 1000 (which is optimal frequency range). In this case, (3000-100)/1000 which is 3.

4. The **Step Frequency Range Minimum** and **Step Frequency Range Maximum** properties will range for only 1000 Hz and their corresponding user defined frequencies. In this example:

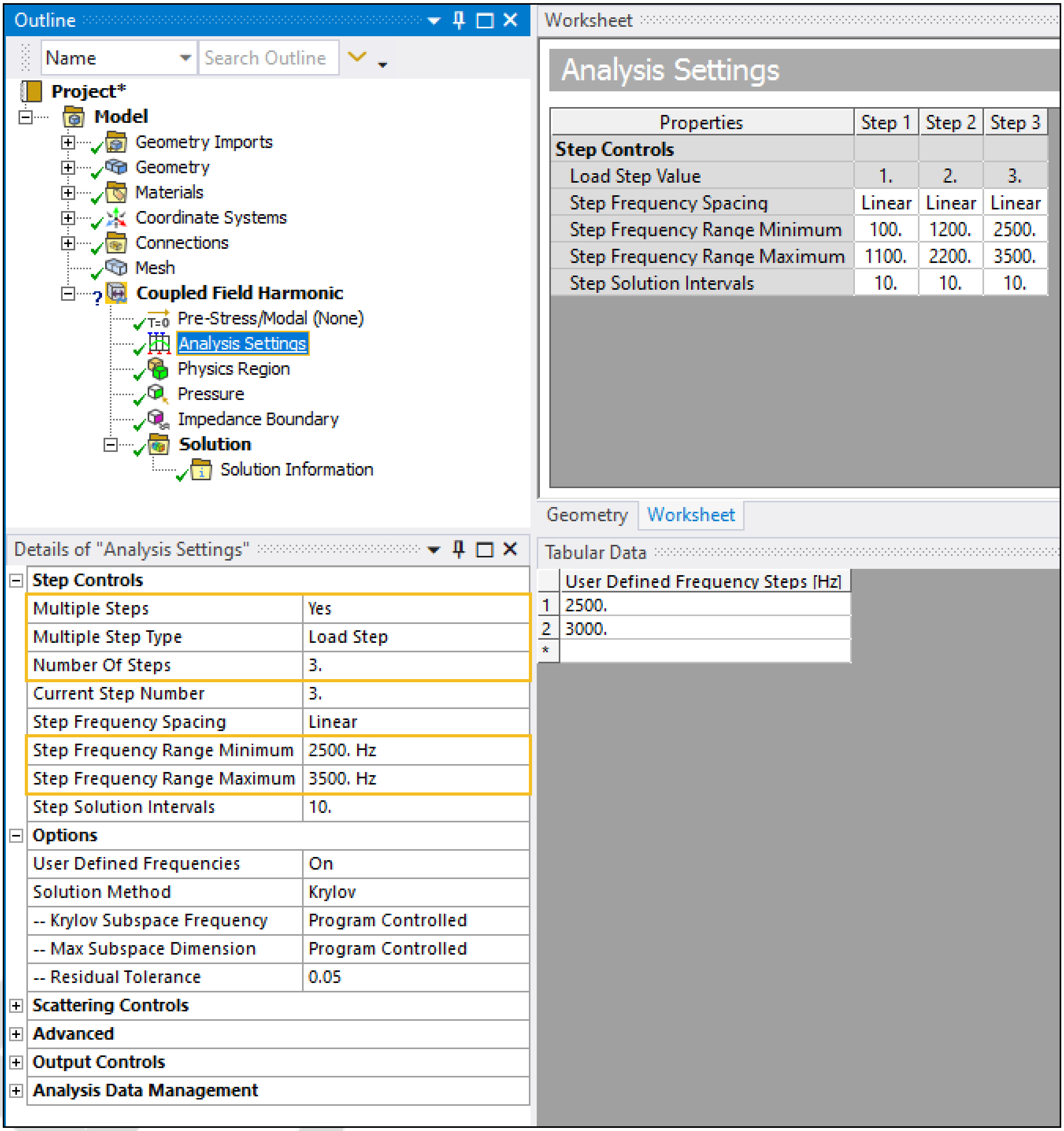
   - 1st Load Step: The Range Minimum = 100 Hz and the Range Maximum = 1100 Hz. The User Defined Frequencies = 100,500,800 Hz.

   - 2nd Load Step: The Range Minimum = 1200 Hz and the Range Maximum = 2200 Hz. User Defined Frequencies = 1200,1500,1700 Hz.

   - 3rd Load Step: The Range Minimum = 2500 Hz and the Range Maximum: 3500 Hz. User Defined Frequencies = 2500 and 3000 Hz.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. 
Outline
Name
Search Outline
Project*
Model
Geometry Imports
Geometry
Materials
Coordinate Systems
Connections
Mesh
Coupled Field Harmonic
Pre-Stress/Modal (None)
Analysis Settings
Physics Region
Pressure
Impedance Boundary
Solution
Solution Information
Worksheet
Analysis Settings
Properties | Step 1 | Step 2 | Step 3
Step Controls
Load Step Value | 1. | 2. | 3.
Step Frequency Spacing | Linear | Linear | Linear
Step Frequency Range Minimum | 100. | 1200. | 2500.
Step Frequency Range Maximum | 1100. | 2200. | 3500.
Step Solution Intervals | 10. | 10. | 10.
Geometry
Worksheet
Details of "Analysis Settings"
Step Controls
Multiple Steps | Yes
Multiple Step Type | Load Step
Number Of Steps | 3.
Current Step Number | 3.
Step Frequency Spacing | Linear
Step Frequency Range Minimum | 2500. Hz
Step Frequency Range Maximum | 3500. Hz
Step Solution Intervals | 10.
Options
User Defined Frequencies | On
Solution Method | Krylov
-- Krylov Subspace Frequency | Program Controlled
-- Max Subspace Dimension | Program Controlled
-- Residual Tolerance | 0.05
Scattering Controls
Advanced
Output Controls
Analysis Data Management
Tabular Data
User Defined Frequency Steps [Hz]
1 2500.
2 3000.
*

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 14: Base Excitations and Loads in a Modal analysis Scaled in a Linked MSUP System

Using beta options, the application makes existing loads available to Modal analysis systems so that you can use them to create load vectors and base excitations that you can then make available to a downstream MSUP system and scale them.

Go to a section topic:

Overview

When beta options are active, you can include the following loading conditions in a Modal analysis, which are not normally supported:

- Acceleration and Acceleration defined as a Base Excitation
- Displacement defined as a Base Excitation
- Standard Earth Gravity
- Pressure
- Force

The Modal analysis also supports the following additional properties when beta options are active:

- **Include Residual Vector**: Available in the **Options** category of the **Analysis Settings** object. When set to **On** the application executes the **RESVEC** command and calculates residual vectors for the Modal analysis. The default setting is **Off**. See the **RESVEC** command documentation in the *Mechanical APDL Command Reference* for additional information.
- **Load Control**: Available in the Pre-Stress object if you also have an upstream Static Structural analysis specified for the Modal system. You use this property to define load generation for prestressed analyses.

The above options enable you to generate load vectors and base excitations which you can then transfer to a downstream Mode Superposition (MSUP) analysis, either a Harmonic Response MSUP or Transient Structural MSUP.

Known Issues and Limitations

Note the following known issues and limitations:

- Having both an unsuppressed Acceleration Base Excitation and a Load Application with Source property set to an upstream Base excitation under an MSUP environment results in a crash.
- Load vectors generated for modal loads with the **Load Vector Assignment** property set to **Program Controlled** and applied by **Surface Effect** may be incorrect.

Load Application (Beta) Object

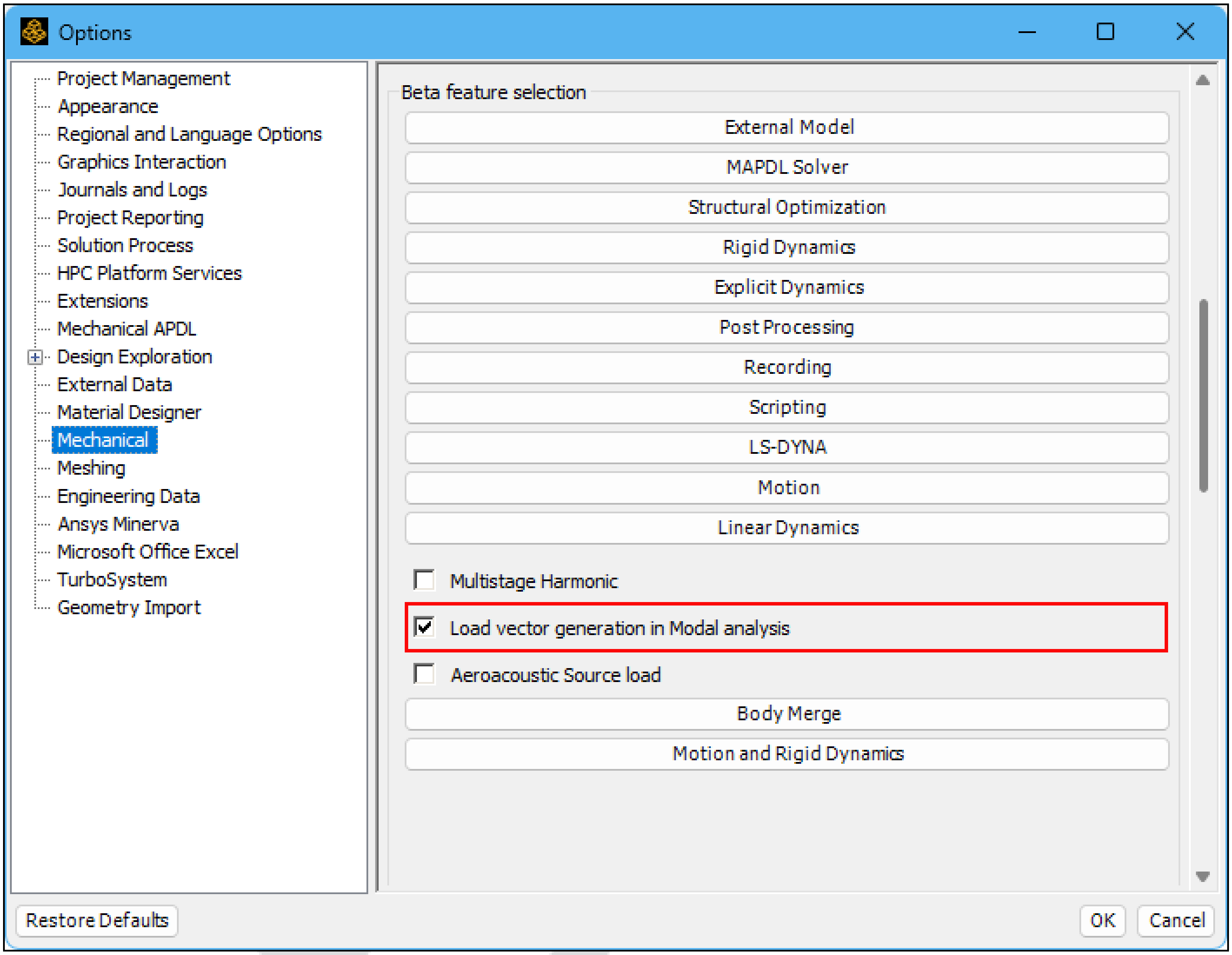
Based on how you configure your upstream system(s), use the beta object, **Load Application (Beta)**, in your downstream Harmonic Response MSUP or Transient Structural MSUP system to transfer and scale load vectors and base excitations generated in the upstream Modal analysis.

Activate Beta Options and Feature Flag

This procedure requires you to close the application in order to make the feature available.

1. Active Beta Options (p. 1). **Beta Options** must be activate in order for the next option to become available.
2. To activate the feature selection option highlighted below, select the **Tools** drop-down menu, and then Options. This displays the dialog shown below. Select the Mechanical option from the tree. Scroll to the **Linear Dynamics** group and select **Load vector generation in Modal analysis**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. As prompted, close and reopen the Ansys® Workbench™ application.

Application

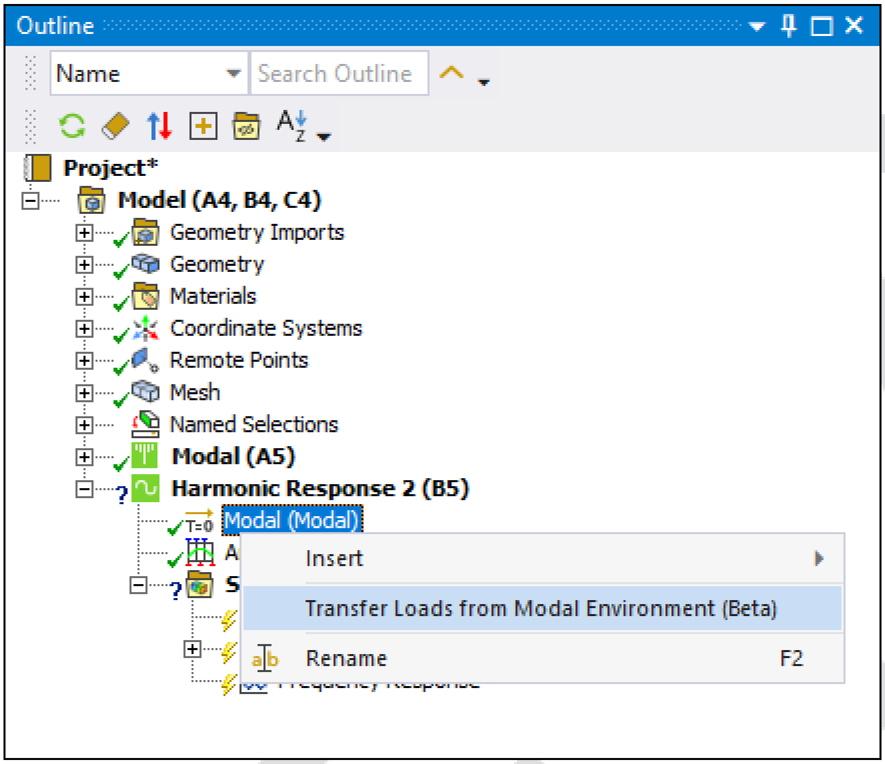
This procedure assumes that you have properly prepared the simulation to the point where you can now insert the **Load Application (Beta)** object into a downstream Harmonic Response MSUP or Transient Structural MSUP analysis. That is, you have properly configured and solved your Modal or prestressed Modal analysis and inserted the desired downstream MSUP analysis.

Tip:

Based on your downstream analysis type, you can use the initial specification object to automatically generate **Load Application (Beta)** objects. A Harmonic Response analysis includes a Modal initial specification object and the Transient Structural analysis includes an Initial Conditions object which includes a **Modal** object you use to select the upstream system. Right-click the associated object and select **Transfer Loads form Modal Environment (Beta)**. The application automatically creates one **Load Application (Beta)** object for each base excitation or load with **Program Controlled Load Vector Assignment** defined in the upstream

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

system as well as one **Load Application (Beta)** object for each load vector number assigned to upstream loads for **Manual Controlled Load Vector Assignment**. See below (p. 60) for additional information.

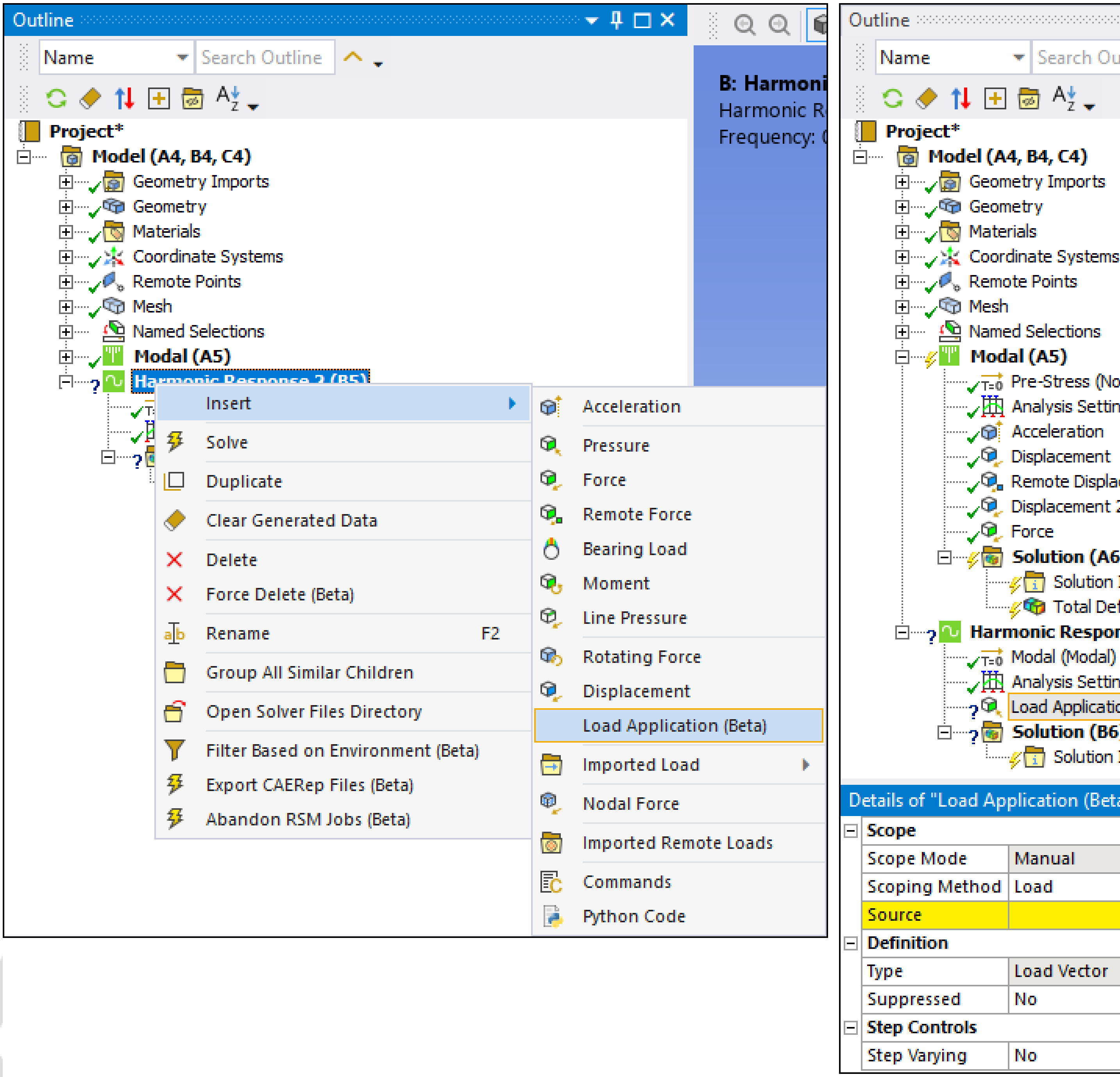


To specify a **Load Application (Beta)** object:

1. Right-click the environment object of the downstream and select **Insert** > **Load Application (Beta)**. The application inserts the object.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Specify the **Scoping Method** property as either **Load** or **Load Vector**. Based on your selection, specify one of the following additional properties:

   - **Source**: Select a desired load from the drop-down list.
   - **Load Vector Number**: Enter the desired load vector number. This value refers to the load vector numbers specified in the upstream load objects.

3. Specify the **Absolute Result** property for base excitation source loads. Options include **Yes** (default) and **No**. Set this property to **No** if you do not want to include enforced motion.

   Important:

   If you apply more than one base excitation, the **Absolute Result** property for each must have the same setting, either **Yes** or **No**.

4. Define the **Magnitude** property. Magnitude properties for scaling displacement base excitations have a length unit and scaling acceleration base excitations have acceleration unit. They are otherwise unitless.

Note:

See the Object Reference - Details Pane Properties (p. 60) topic below for descriptions of each property available for the **Load Application (Beta)** object.

Automatically Defined Load Application (Beta) Objects

The application examines each upstream base excitation and load to determine how to define the properties of the associated downstream **Load Application (Beta)** object. Note the following:

- For an upstream base excitation, the application automatically sets the **Type** property of the downstream **Load Application (Beta)** object to **Base Excitation**.
- For an upstream loading condition whose **Load Vector Assignment** property is set to **Program Controlled**, the application automatically sets the **Scoping Method** property of the downstream **Load Application (Beta)** object to **Load**.
- For an upstream loading condition whose **Load Vector Assignment** property is set to **Manual**, the application automatically sets the **Scoping Method** property of the downstream **Load Application (Beta)** objects to **Load Vector** and then assigns the associated **Load Vector Number** to the objects.

  Important:

  If more than one of the loading conditions uses the same load vector number, the application only creates one Load Application (Beta) object for that particular load vector number, not multiple objects.

Object Reference - Details Pane Properties

The Details pane for **Load Application (Beta)** object includes the following properties.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
BETA

Category	Property/Options/Description
Scope	**Scope Mode**: Read-only property that describes whether the object is created manually or using the content (right-click) menu option **Transfer Loads from Modal Environment (Beta)** of the initial specification object (described above under the Tip (p. 57)). **Scoping Method**: This property specifies whether the Load Application scales a base excitation or load defined or load components with the same load vector number. Options include Load (default) and Load Vector. **Source**: Visible when the **Scoping Method** is set to **Load**. This field provides a drop-down list of valid loads included in the upstream Modal analysis. Valid loads include base excitations and loads with **Load Vector Assignment** property set to **Program Controlled**. **Load Vector Number**: Visible when the **Scoping Method** is set to **Load Vector**. Specify a load vector associated specified in the upstream load.
Defin- i- tion	**Type**: Read-only field that describes whether the upstream Source load generated a base excitation or load vectors when **Scoping Method** is **Load**. **Absolute Result**: For Base Excitation Type only. This option enables you to include enforced motion with or without base motion. Options include **Yes** (default) and **No**. **Magnitude**: This property defines the scaling for the upstream loads or base excitations. • When you set the **Scoping Method** property to **Load Vector**, the application scales all upstream Components that have an assigned load vector number, created either manually or using the **Program Controlled** option, that is equal to the value of the **Load Vector Number** property of the **Load Application (Beta)** object. • When you set the **Scoping Method** property to **Source**, the application scales the non-zero real parts of the load when it is a base excitation, or when you define the load using one of the following options in the upstream system: – **Vector** – **Vector: Real – Imaginary** – **Normal To** – **Normal To: Real – Imaginary** • **Magnitude - Real**: Defines the scaling for the non-zero real part of the source load when it is a base excitation or defined by **Vector**, **Vector: Real - Imaginary**, **Normal To**, or **Normal To: Real - Imaginary** in the MSUP Harmonic Response system. • **Magnitude - Imag**: Defines the scaling for the non-zero imaginary part of the source load when it is a base excitation or defined by **Vector**, **Vector: Real - Imaginary**, **Normal To**, or **Normal To: Real - Imaginary** in the MSUP Harmonic Response system. • **X Component**: Defines the scaling for the non-zero real part of the X Component of the source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Transient Structural system.

Category	Property/Options/Description
	• **X Component - Real**: Defines the scaling for the non-zero real part of the X Component of the source load if the load is defined by Components or Components: Real - Imaginary in an MSUP Harmonic Response system. • **X Component - Imag**: Defines the scaling for the non-zero imaginary part of the X Component of the source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Harmonic Response system. • **Y Component**: Defines the scaling for the non-zero real part of the Y Component of the source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Transient Structural system. • **Y Component - Real**: Defines the scaling for the non-zero real part of the Y Component of the source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Harmonic Response system. • **Y Component - Imag**: Defines the scaling for the non-zero imaginary part of the Y Component of the source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Harmonic Response system. • **Z Component**: Defines the scaling for the non-zero real part of the Z Component of the source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Transient Structural system. • **Z Component - Real**: Defines the scaling for the non-zero real part of the Z Component of the Source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Harmonic Response system. • **Z Component - Imag**: Defines the scaling for the non-zero imaginary part of the Z Component of the Source load if the load is defined by **Components** or **Components: Real - Imaginary** in the MSUP Harmonic Response system. **Suppressed**: Exclude the object in the analysis. Options include **No** (default) and **Yes**.
Step Controls (Multiple Step Harmonic Response Analyses Only)	**Step Varying**: Options include **No** (default) and **Yes**. When you select **No**, the **Load Application (Beta)** object is applicable at all defined steps. When set to **Yes**, only the load step selected in either the **RPM Selection** or **Step Selection** properties described below is applicable. **RPM Selection**: This property displays when the **Multiple Step Type** property is set to **RPM**. Select an **RPM Selection** value from the drop-down menu. **Step Selection**: This property displays when the **Multiple Step Type** property of the **Analysis Settings** object is set to Load Step. Specify a **Step Selection** value from the values available in the **Load Step Value** property of the **Analysis Settings** object to use for the **Load Application (Beta)** object.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 15: Coupled Field Analyses using Composite Materials

Now, when you have **Beta Options** activated, you can link Ansys Composite PrepPost (ACP) systems to a Coupled Field analysis. See the Basic Workflow section of the *ACP User's Guide* for the steps to configure this analysis combination.

Note:

The composite body imported into Mechanical needs to correspond to a **Physics Region** that is specified as structural only (**Structural** property set to **Yes**).

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 16: Aeroacoustic Source Imported Loading Condition

To study sound propagation of aerodynamically generated noise, Mechanical enables you to import an **Aeroacoustic Source** loading condition into either a Coupled Field Harmonics or Harmonic Acoustics analysis system. You create this loading condition in Ansys® Fluent®, and then import it into a supported downstream Mechanical system, to solve for sound propagation.

Go to a section topic:

Assumptions

Note the following:

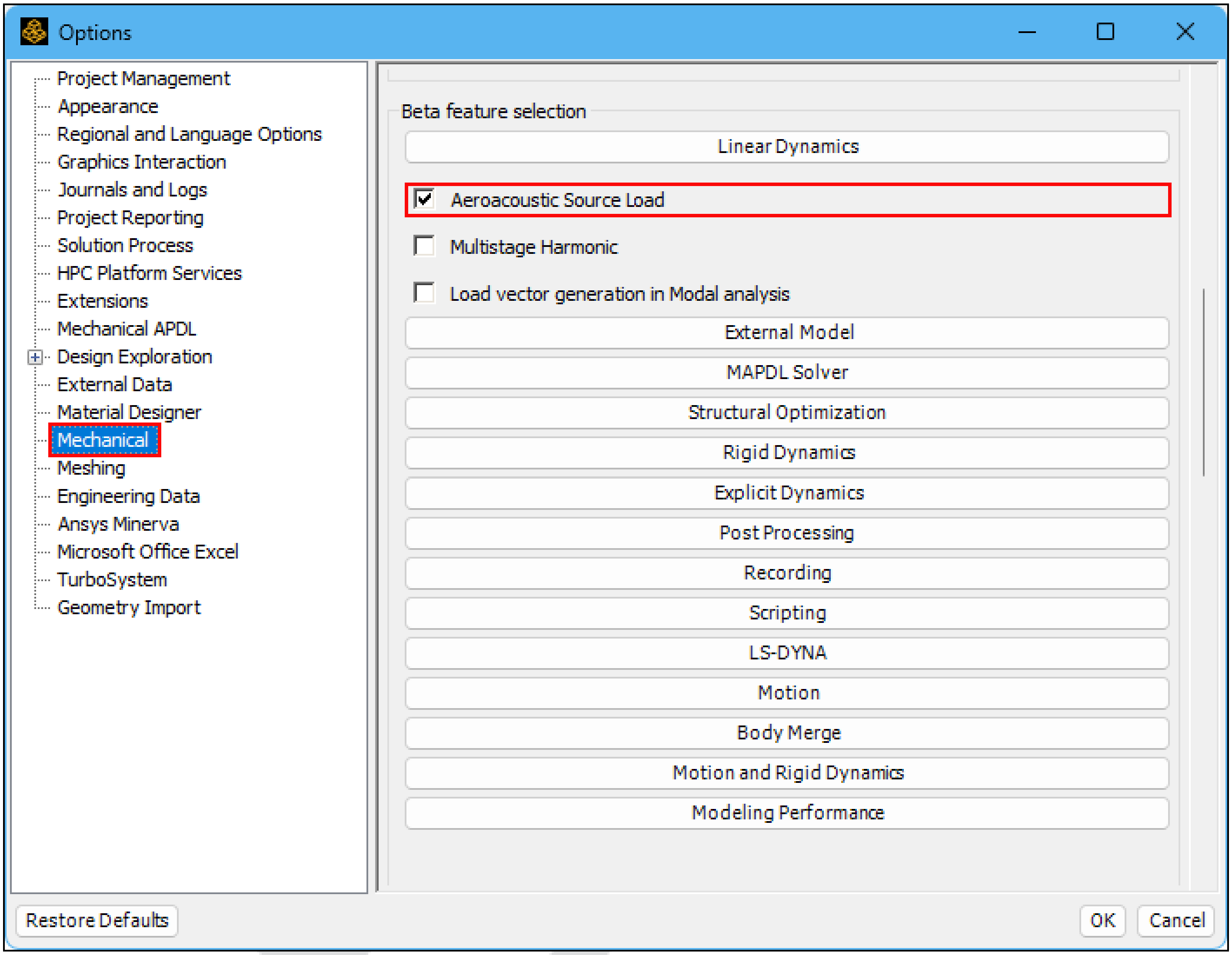
- You must assign the appropriate acoustics material to the bodies of model.
- You are required to create the required source file.
- This feature is only applicable for low Mach number flows.
- There may be modelling errors due to the Lighthill aeroacoustics analogy of splitting the compressible flow variables into the two parts (flow and acoustics) and handling them separately with the two approximate mathematical models connection through a source term in the wave equation.

Activate Beta Options and Feature Flag

This procedure requires you to close the application in order to make the feature available.

1. Activate Beta Options (p. 1). You need to select **Beta Options** and then close the dialog for the next option to become available.
2. To activate the feature selection option highlighted below, select the **Tools** drop-down menu, and then Options. This displays the dialog shown below. Select the Mechanical option from the tree. Scroll to the **Linear Dynamics** group and select **Aeroacoustic Source Load**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



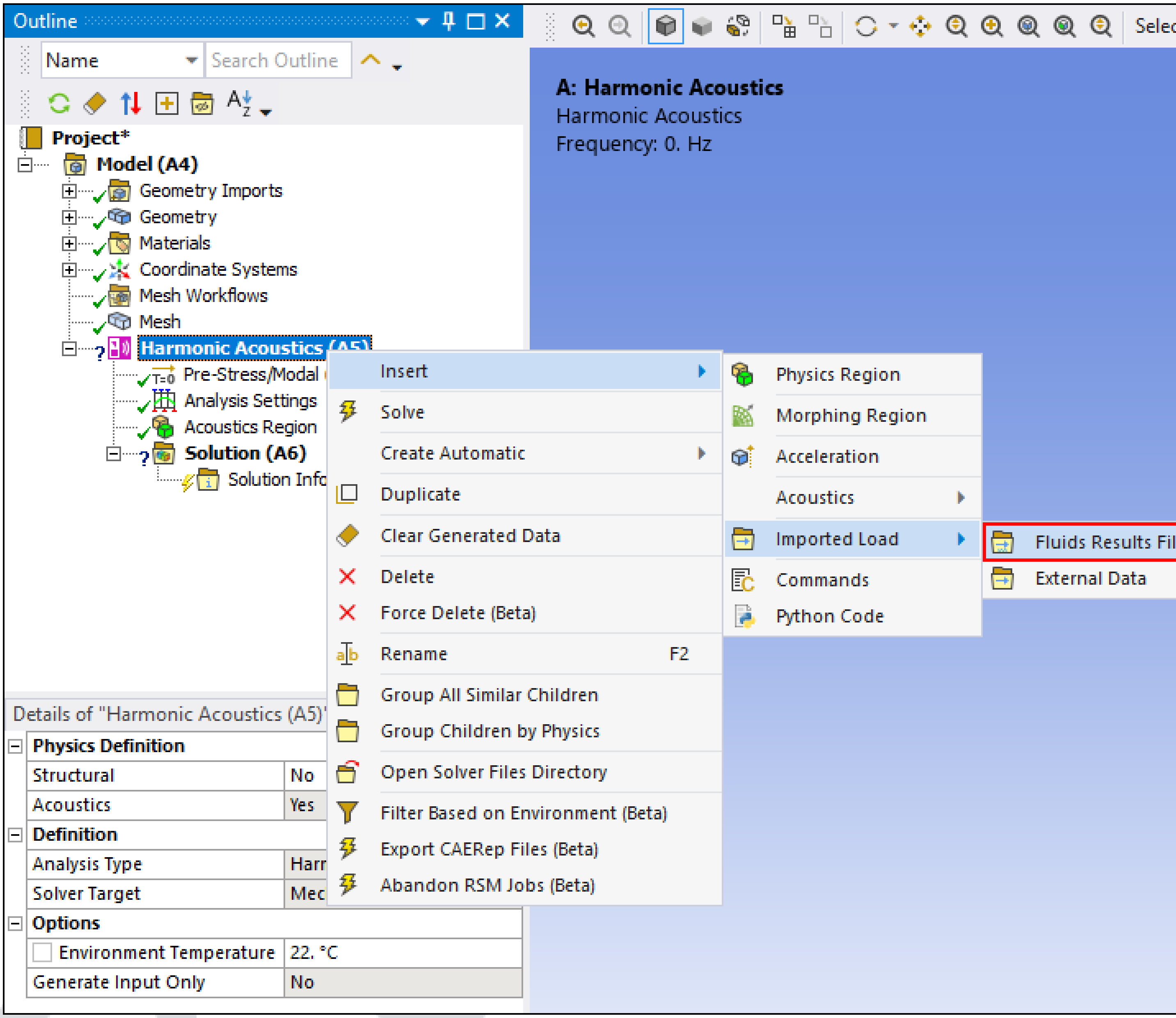
3. As prompted, close and reopen the Ansys® Workbench™ application.

Application

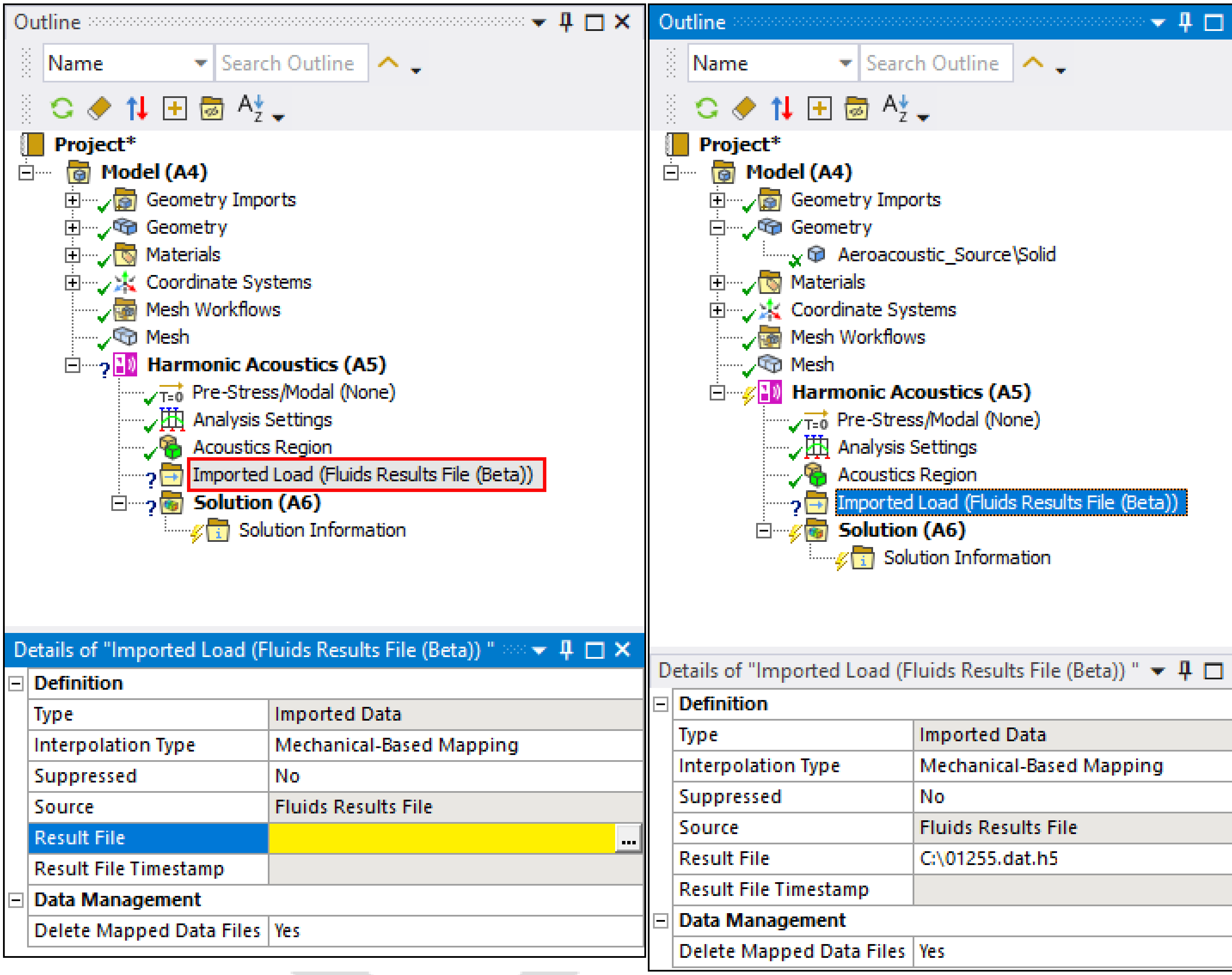
This procedure uses a Harmonic Acoustics analysis to illustrated the steps to import an **Aeroacoustic Source** loading condition.

1. Right-click the Environment object in the Outline and select **Insert** > **Imported Load** > **Fluids Results (Beta)**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

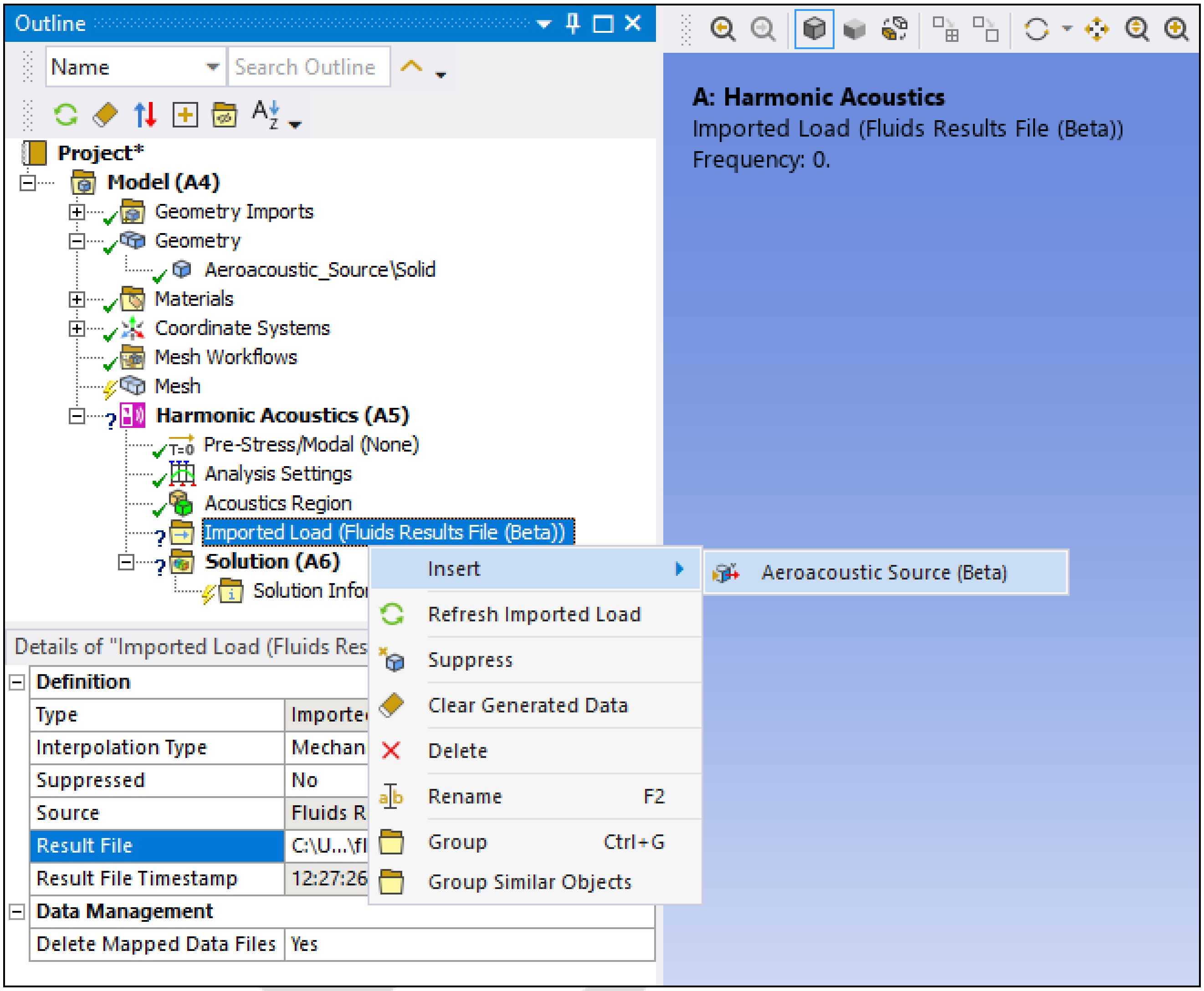


2. Using the **Result File** property, navigate to and select your `*.dat.h5` file created Fluent. The `*.cas.h5` file must also be available in the same folder as the `*.dat.h5`.

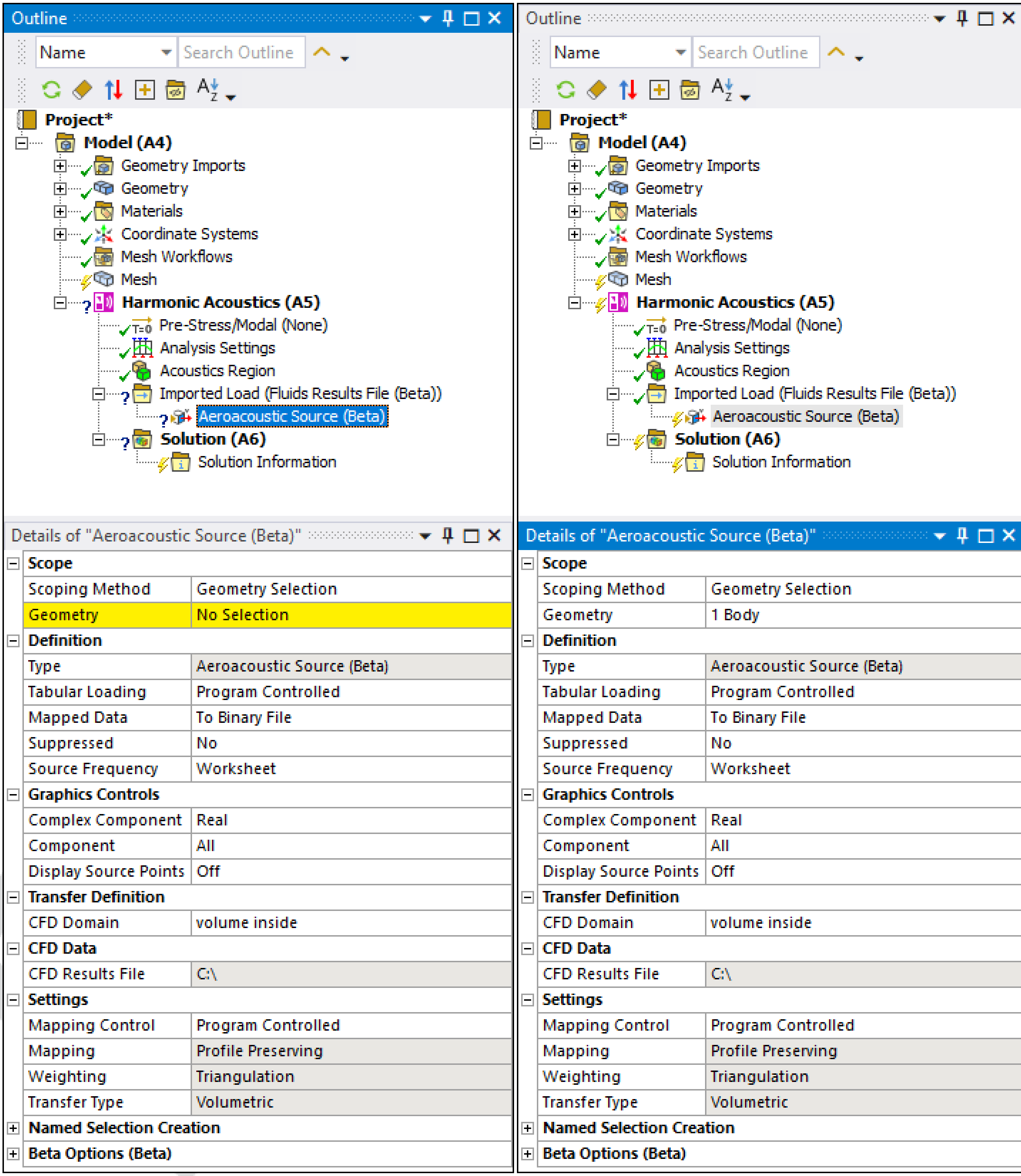


3. Right-click the **Imported Load (Fluids Results File (Beta))** object and select **Aeroacoustic Source (Beta)**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Select the desired acoustic bodies (only) using the **Geometry** property and click **Apply**.



5. Right-click the **Aeroacoustic Source (Beta)** object and select **Import Load**.

6. Complete any additional property specifications. Note that:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You apply this load using the using the data processing framework (DPF) to generate a binary file: the **Mapped Data** property set to **To Binary File**. See the Load Mapping Workflow Specification section for the steps to complete any required specifications.
- This feature supports source frequency filtering and source frequency interpolation.
- You can specify one or more CFD domains/zones using the **CFD Domain** property.
- The divergence of the Lighthill tensor (a vector quantity) is transferred from the CFD cell center to the acoustic finite element node using the Point Cloud algorithm. Therefore, only the **Triangulation** and **Distance Based Average** mapping is supported.

Source Term Computation in Ansys Fluent

Aerodynamically generated noise for low Mach number flows can be predicted using the Lighthill's analogy. This offers an efficient alternative for the direct method which is computationally expensive. Lighthill developed the following wave equation using the continuity and momentum equations of motions:

$$\frac{\partial^2 \rho'}{\partial t^2} - c_0^2 \nabla^2 \rho' = \nabla \bullet \left(\nabla \bullet T_{ij} \right)$$

Where the Lighthill Tensor (T_{ij}) is:

$$T_{ij} = \rho v_i v_j + \tau_{ij} + c_0^2 \left(\rho - \rho_0 \right) \delta_{ij}$$

And the variables are:

ρ = Fluid density.

v = Flow velocity.

ρ_0 = Destiny at rest.

c_0 = Speed of sound at rest.

T_{ij} = Lighthill tensor.

τ_{ij} = Reynolds stress tensor.

For low Mach number, incompressible flow, the Lighthill Tensor can be approximated as only the Reynolds stress term:

$$T_{ij} = \rho v_i v_j$$

The divergence of this Lighthill Tensor also known as Aeroacoustic Source can be applied as the sound source term to solve for the sound propagation in Harmonic Acoustic wave equation in Mechanical. The divergence of this Lighthill Tensor can be obtained using the named expressions in Fluent. Then, a Discrete Fourier Transform is performed to transform the results in the frequency domain. These transformed results can be written to a Case and Data (extensions?) file that can be imported in to Mechanical.

See the following sections of the *Fluent User's Guide* for more information:

BETA

- Creating and Using Expressions
- Zone-Specific Sampling Options Dialog Box
- Reading and Writing Case and Data Files Together

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 17: Creating Spot Weld Groups

Prerequisites

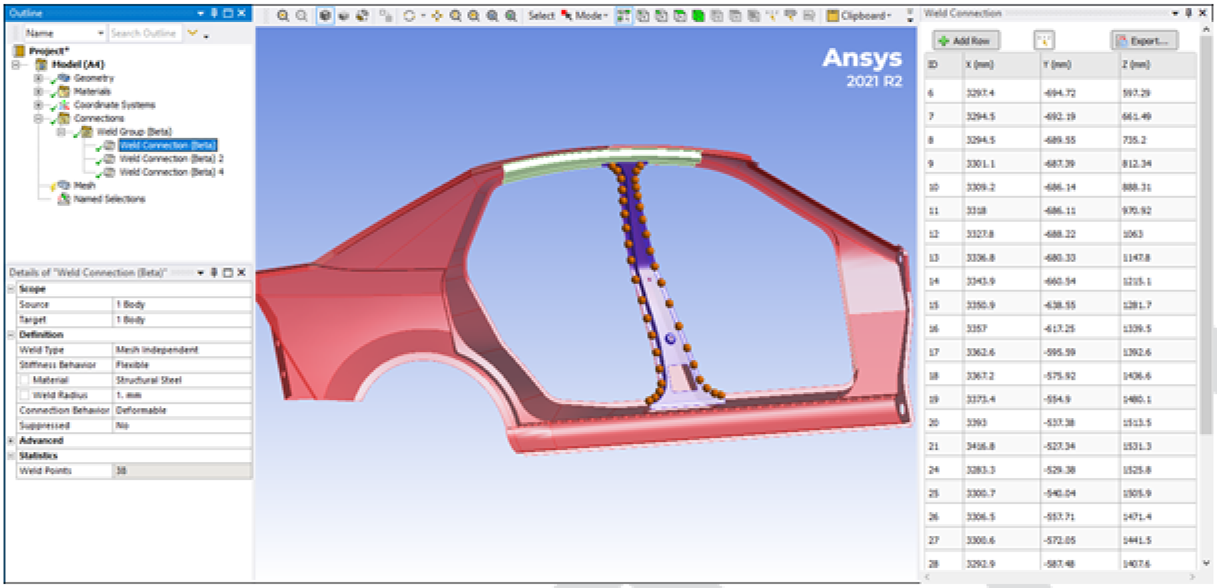
This documentation assumes that you understand the concept and use of the application's existing **Spot Weld** feature. The **Weld Group** (Beta) feature is like the existing **Spot Weld** control, but has a few notable differences, including:

- The Spot Weld feature requires a Spot Weld for each weld. The Weld Group (Beta) groups numerous spot welds by the parts/bodies they connect.
- The Spot Weld feature constructs "mesh dependent welds." The Weld Group (Beta) provides an option to construct the welds as mesh dependent or as mesh independent (as discussed in the Understanding Mesh Dependent and Mesh Independent Definitions topic).
- The Spot Weld feature requires that you define the spot welds in SpaceClaim or DesignModeler. Or, that you at least create the spot weld vertices to be used (before entering Mechanical). There is no current method to create a Spot Weld in Mechanical.
- Currently, you cannot import spot welds defined in SpaceClaim or DesignModeler into Mechanical as a group, however, there is an existing script that imports point locations.
- Activate Beta Options (p. 1) to turn on the feature.

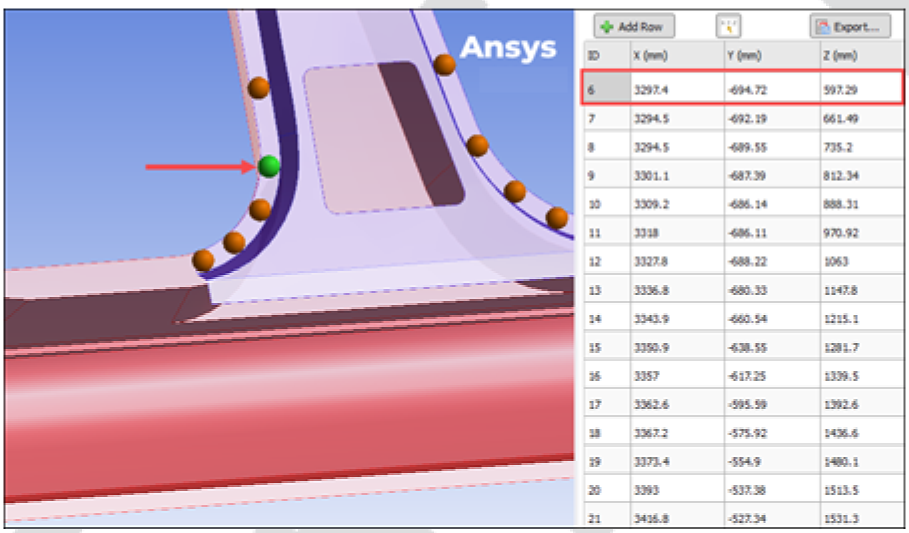
Overview

The **Weld Group** connection feature (folder) enables you to manage a group of spot welds that connect multiple surface body parts. This folder provides properties to either import predefined spot welds or to manually create individual spot welds. An example of an imported group of spot welds is illustrated below. Once imported, the application automatically creates **Weld Connection** objects. The data for the **Weld Connection** object is displayed in the Details pane. The data for each spot weld is displayed in the Weld Connection Worksheet. Data includes the source and target bodies of the weld and properties relating to the set of welds in the Details view and an identifier and a coordinate location for each spot weld in the Worksheet.

When you select a **Weld Connection** object, the application highlights the corresponding spot welds for the given connection in the **Geometry** window, as illustrated below.



In addition, the Source body displays in red, the Target body displays in blue, and an orange ball displays each spot weld. Selecting a row in the Weld Connection Worksheet highlights the individual spot welds in green.



Understanding Mesh Dependent and Mesh Independent Definitions

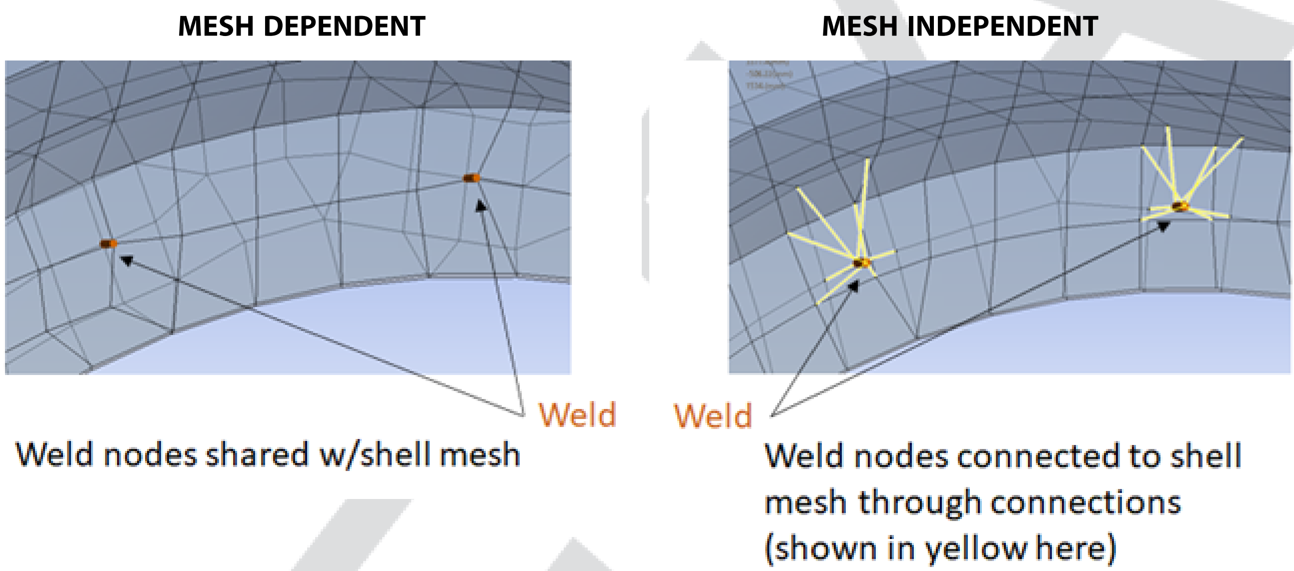
The spot welds are assigned a type, either Mesh Dependent or Mesh Independent (default). These settings are specified by the Weld Type property using the Weld Connection objects. The Weld Type, along with the setting for Stiffness Behavior determines how the spot welds will be modeled when generating the mesh.

In the following example, you can see:

- The weld element is modeled the same for mesh dependent and mesh independent weld types. Using the Stiffness Behavior property, you specify the element as a Flexible (beam), Stiff Beam or Rigid (beta).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For each mesh dependent weld, nodes are inserted during mesh process so that the weld element connects directly to the shell mesh on the Source and Target bodies. That is, each node of the weld element is shared with the shell elements.
- For mesh independent welds, the shell mesh and weld elements are generated independently, and the weld element nodes are not shared with the shell mesh. The Connection Behavior property controls how the nodes of the weld elements are tied to the shell mesh. Options for this property include Deformable and Rigid. The connection between the weld elements and the shells is displayed as a yellow spider web.



Simply put, for mesh dependent welds, the nodes of the spot weld elements are directly connected to the shell mesh. For mesh independent welds, the spot weld elements are connected to the shell mesh either through contact or rigid connections based on property settings.

Spot Weld File Format

Prior to importing your spot weld file, you must ensure that the file you use follows the conventions used by Dyna or Primer, as shown in this example:

```
$         10        20        30        40        50        60        70        80        90        100
$         < weld ID>          < X coord>< Y coord>< Z coord>          < Part 1 >< Part 2 >< Part 3 >< Part 4 >
SPOTWELD           1        POINT 2173.4331-644.90894 283.17007     PARTS     64000     42000
SPOTWELD           2        POINT 2197.3462-640.45935 337.95139     PARTS     64000     42000
```

As illustrated, the spot weld files have the following properties:

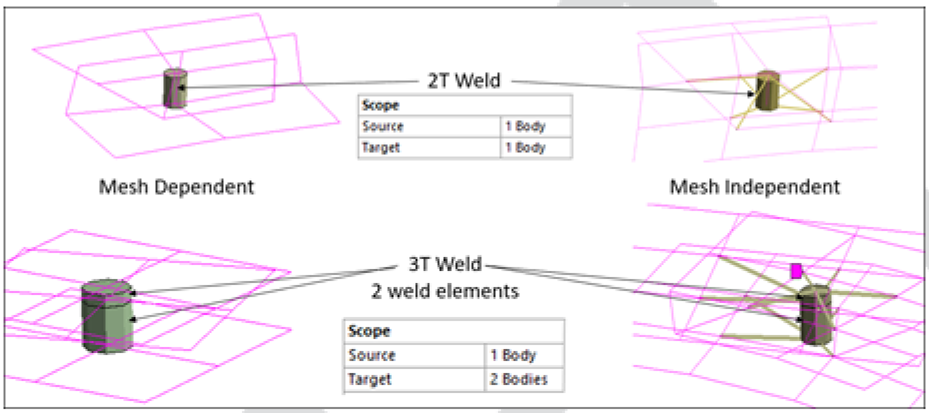
- Asci file where text is separated by spaces (*.csv type format)
- Each line represents a spot weld, and:
  - The first entry should be “SPOTWELD.”
  - The second entry is the id of the weld.

  - The third entry is the X position of the weld.
  - The fourth entry is the Y position of the weld.
  - The fifth entry the Z position of the weld.
  - Part identifiers are optional. If part ids are desired, the 6th entry should specify “PARTS” and then the seventh to nineth entries indicate the parts of the weld.

- If parts are used the part ID in the weld file need to be encoded in the names of the bodies. For example, if the weld file contains part IDs (54325,54625,54525), the body names would need to appear as follows, where the body names include the part IDs at the end of the body name.

Weld Thickness

Only two thickness (2T) and three thickness (3T) welds are supported:

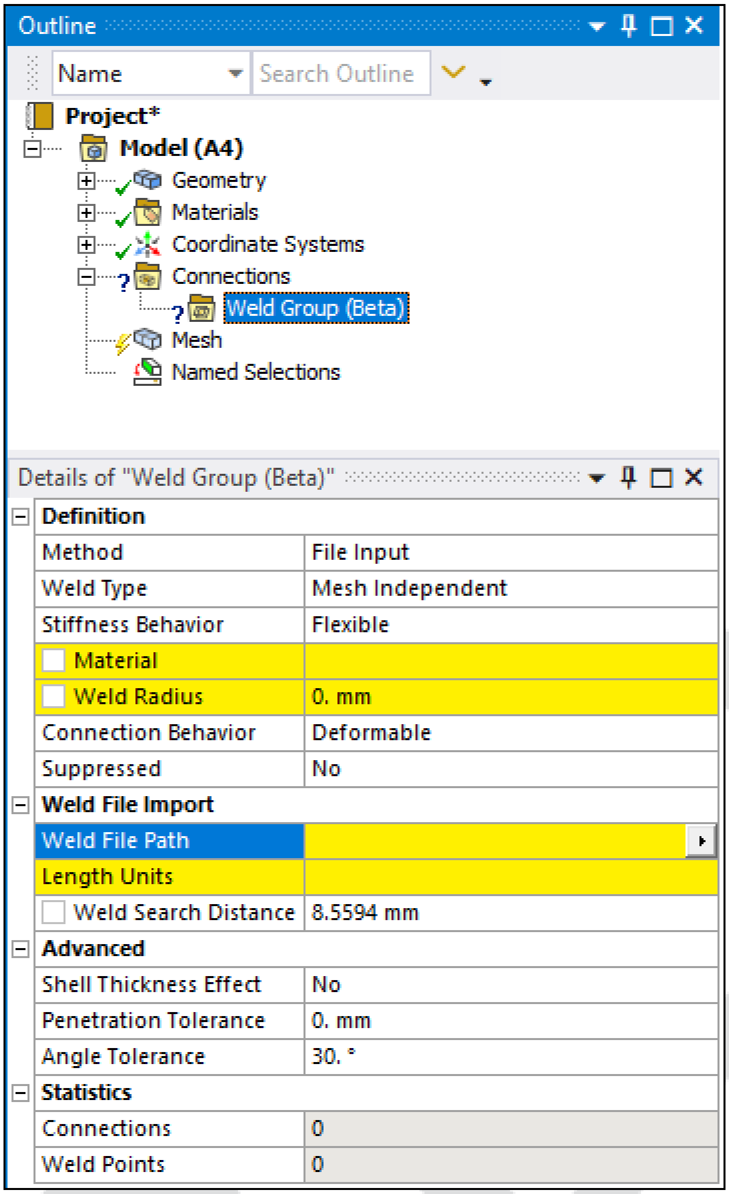


Importing Weld Connections

To import Weld Group Connections:

1. Highlight the **Connections** object and then select the **Weld Group** option from the **Connect** group of the **Connections** Context tab to insert a **Weld Group** object. An example of the object’s Details is show here. Many properties have default values.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



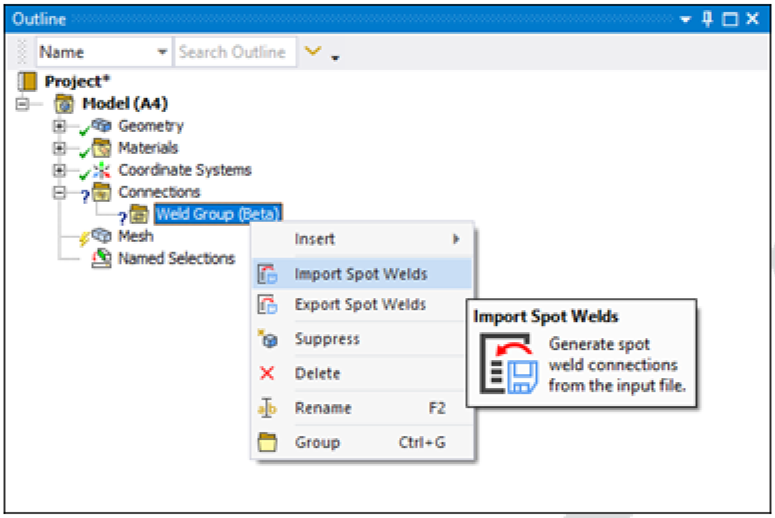
2. Specify highlighted properties. Properties include:

   - Material
   - Weld Radius
   - Length Units

3. Select the entry field of the **Weld File Path** property and the arrow menu display option. A dialog opens.

4. Navigate to and select the file you wish to import from the Open dialog.

5. Right-click the **Weld Group** object and select the **Import Spot Welds**.



6. Select each **Weld Connection** object and review all property definitions as well as the data provided in the **Weld Connection** window. Perform corrections as needed.

Manually Creating Weld Group Connections

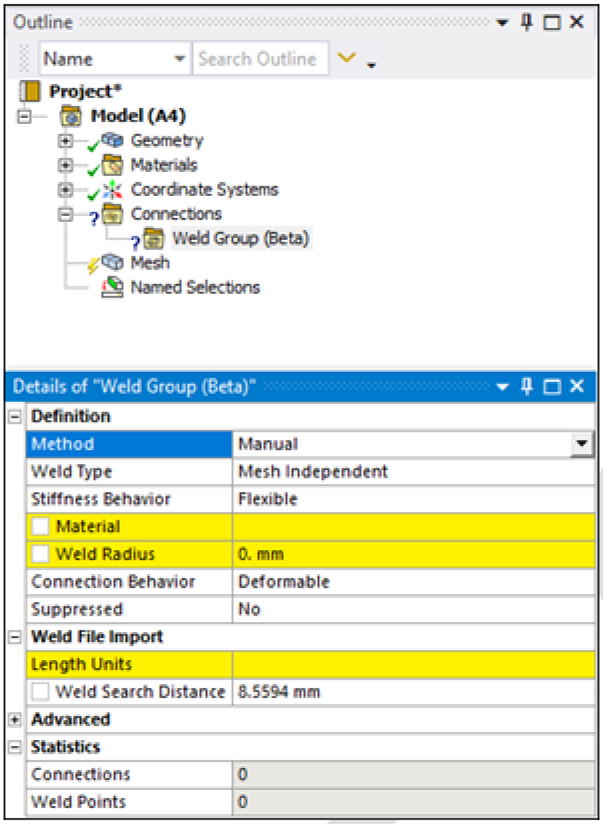
To create Weld Connections manually:

1. Highlight the Connections object and then select the Weld Group option from the Connect group of the Connections Context tab to insert a Weld Group object. An example of the object's Details is show here. Many properties have default values.

2. Set the **Method** property to **Manual**.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Specify highlighted properties. Properties include:

   - Material
   - Weld Radius
   - Length Units

4. Right-click the **Weld Group** object and select **Insert** > **Weld Connection**.

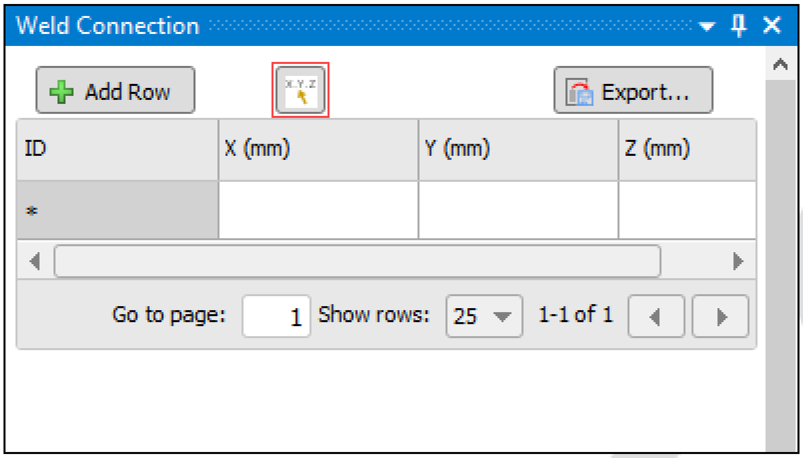
5. Specify the desired surface bodies for the **Source** and **Target** properties.

6. Modify object properties as needed.

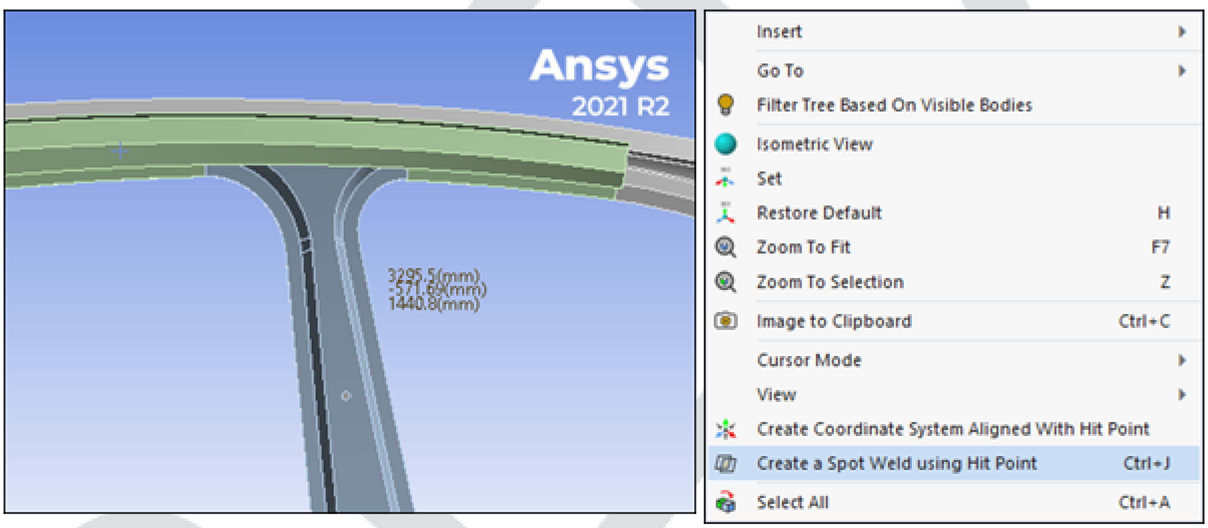
7. When inserted, the Weld Connection window displays automatically. The Hit Point Coordinate option, highlighted below, is active by default. This option enables you to set a coordinate location. As show in the next step, the option displays the exterior coordinates of the model as the cursor is moved across the model.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. Coordinate labels appear where you place the cursor. Select the locations for your spot weld, right-click, and select the option Create Spot Weld using Hit Point. The data table automatically populates each coordinates each time a spot weld is specified.



This feature is functional on faces only. It is not functional on edges or line bodies.

9. Continue creating sport welds as desired.

Weld Group Object Properties

The properties of the **Weld Group** folder specify values for all **Weld Connection** child objects.

Category	Property/Options/Descriptions
Definition	**Method**: Specify to import spot weld from a text file or to create them manually. Options include **File Input** (default) and **Manual**. **Stiffness Behavior**: Depending upon the **Weld Type** setting, select a (element) stiffness behavior, including: • **Flexible** (default)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Property/Options/Descriptions
	• **Stiff beams** • **Rigid (beta)** **Material**: Specify desired material for all (child object) weld connections. **Weld Radius**: When the **Stiffness Behavior** is set to **Flexible** or **Stiff Beam**, beam elements are created during mesh generation. The beams automatically receive a circular cross section defined for them where the radius is set by this property. In other words, this **Weld Radius** is the circular radius representing the weld. **Connection Behavior**: This property displays when the **Weld Type** property is set to **Mesh Independent**. The **Connection Behavior** (for the yellow lines) is defined as either: • **Deformable** (default) • **Rigid** **Snap To Edge Tolerance**: This property displays when the **Weld Type** property is set to **Mesh Dependent**. The application uses this tolerance (+/-) value during the meshing process to place or "snap" a weld point to a boundary (edge) as opposed to placing the weld point on a face that is close to the boundary (closer than defined tolerance). This helps makes sure that small edges are not created. **Suppressed**: Options include **Yes** or **No** (default). This property enables you to remove (and re-include) the object from analysis processes.
Advanced	**Shell Thickness Effect**: This property enables you to include the thickness of the surface body during contact calculations. Instead of contact being detected on the face of the surface body, contact will be detected at a distance that is half the thickness from the face. **Penetration Tolerance**: Specify a desired **Penetration Tolerance** value. **Angle Tolerance**: This property enables you to specify a tolerance (+/-) value that the desired/ideal orthogonal angle of each spot weld may deviate. The default value is `30°`.
Statistics	**Connections**: Total number of **Weld Connection** objects included in group. **Weld Points**: Total number of spot weld connections included in group.

Weld Connection Object Properties

The properties of the **Weld Connection** object specify values for the associated grouping of spot welds.

Category	Property/Options/Descriptions
Scope	**Source**: Display/select the source side surface body for the weld connection. This geometry can be manually selected or automatically generated.

Category	Property/Options/Descriptions
	Target: Display/select the target side surface body for the weld connection. This element can be manually set or automatically generated.
Definition	**Method**: Specify to import spot weld from a text file or to create them manually. Options include **File Input** (default) and **Manual**. **Stiffness Behavior**: Depending upon the **Weld Type** setting, select a (element) stiffness behavior, including: • **Flexible** (default) • **Stiff beams** • **Rigid (beta)** **Material**: Specify desired material for all (child object) weld connections. **Weld Radius**: When the **Stiffness Behavior** is set to **Flexible** or **Stiff Beam**, beam elements are created during mesh generation. The beams automatically receive a circular cross section defined for them where the radius is set by this property. In other words, this **Weld Radius** is the circular radius representing the weld. **Connection Behavior**: This property displays when the **Weld Type** property is set to **Mesh Independent**. The **Connection Behavior** (for the yellow lines) is defined as either: • **Deformable** (default) • **Rigid** **Snap To Edge Tolerance**: This property displays when the **Weld Type** property is set to **Mesh Dependent**. The application uses this tolerance (+/-) value during the meshing process to place or "snap" a weld point to a boundary (edge) as opposed to placing the weld point on a face that is close to the boundary (closer than defined tolerance). This helps makes sure that small edges are not created. **Suppressed**: Options include **Yes** or **No** (default). This property enables you to remove (and re-include) the object from analysis processes.
Advanced	**Shell Thickness Effect**: This property enables you to include the thickness of the surface body during contact calculations. Instead of contact being detected on the face of the surface body, contact will be detected at a distance that is half the thickness from the face. **Penetration Tolerance**: Specify a desired **Penetration Tolerance** value. **Angle Tolerance**: This property enables you to specify a tolerance (+/-) value that the desired/ideal orthogonal angle of each spot weld may deviate. The default value is `30°`. **Snap To Edge Tolerance**: This property displays when the **Weld Type** property is set to **Mesh Dependent**. The application uses this tolerance (+/-) value during the meshing process to place or "snap" a weld point to a boundary (edge) as

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Property/Options/Descriptions
	opposed to placing the weld point on a face that is close to the boundary (closer than defined tolerance). This helps makes sure that small edges are not created.
Statistics	**Weld Points**: Total number of spot weld connections for the object.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 18: Specifying Energy Convergence Criteria

For Static Structural and Transient Structural analyses, Mechanical includes a beta feature that enables you to specify an **Energy Convergence** criteria property. This new property is available in the **Nonlinear Controls** category of the **Analysis Settings** object. When you include this criterion, the solver requires fewer iterations to achieve a moderately accurate solution as compared to the **Force Convergence** criterion. Property options include:

- **Remove** (default)
- **Program Controlled**
- **On**

When set to **On**, the following additional properties become available:

- **Value**: Default value is **Calculated By Solver** which is equal to an entry of zero (0).
- **Tolerance**: Default value is 0.5%.
- **Minimum Reference**: Default value is 1 J.

See the Nonlinear Controls for Steady-State, Static, and Transient Analyses section of the *Mechanical User's Guide* for descriptions of the these properties.

Note:

Like most convergence criteria, all convergence plots include designations where any bisections, converged substeps, converged steps, or remesh points occur.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 19: Creating Local Volume Min/Max Probes

Mechanical includes a beta feature that enables you to display probe labels for the minimum and maximum result values of all active parts. The application currently includes a feature that enables you to display min/max probe labels for the results on exterior faces of the model.

Note:

As needed, see the Activate Beta Options (p. 1) section to make the feature available.

Select an evaluated result object, right-click *in* the **Geometry** window, and then select one of the following options:

- **Create Local Volume Max Probes (Beta)**: Display probe labels for the largest (Max) result values within the local range for all active parts.
- **Create Local Volume Min Probes (Beta)**: Display probe labels for the smallest (Min) result values within the local range for all active parts.

These options support results on the entirety of active parts. Any local min/max value contained in an active part is detected.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 20: Evaluate Gasket Line Pressure Result

When Beta Options (p. 1) are active, you can now create a **Gasket Line Pressure** result. This result creates line pressure values for a normalized sealing load acting on a gasket, per unit length. This enables you to detect fluid leaks across the area of the gasket. This result maps the forces to an ordered array of points along an imaginary curve at the center line of the gasket. This curve is then presented in graphical and tabular form.

This result requires a corresponding scoping beta feature: a Construction Geometry Path specified as an **Offset**. This path creates multiple quasi parallel curves that you apply to the gasket surface.

Using the defined offset path, that includes user-defined discretization points as well as sub-paths, you can integrate pressures and determine loading along an imaginary center line of the gasket surface.

Once you evaluate a **Gasket Line Pressure** result, the application presents the force loads along the center line as well as the pressures detected along each sub-path.

You can use the Chart feature to compare multiple gasket line pressure results.

Define an Offset Path

In order to create an Offset Path, you must have a circular geometry.

1. Using the **Construction Geometry** option, **Path**, specify the **Path Type** property as **Offset**.
2. Define the geometries of the path:
   a. Under the **Reference** category of the Details pane, select a desired edge to define the **Reference Geometry** property.
   b. Under the **Offset Guide** category of the Details pane, select the interior edge of the body to define the **Offset Geometry** property.
3. The application automatically creates sub-paths in between the selected edges using the values of the **Discretization Points** (the default setting is **47**) and **Number of Subpaths** (the default setting is **3**) properties. You can change these values as desired.

Define Gasket Line Pressure Result

In order to solve this result, you need to first define loading along the center line of the gasket body. This can be done by creating a beam and scoping the beam to a Bolt Pretension load.

Specify the result:

1. From the **Solution** object, open the **Gasket** drop-down menu and select **Gasket Line Pressure (Beta)**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the **Scoping Method** property to **Path**.

3. Set the **Path** property to the offset path created above. Make sure that the **Geometry** property displays a single body that contains the offset path.

4. Verify a meaningful **Display Time** property setting. It should not be set to zero.

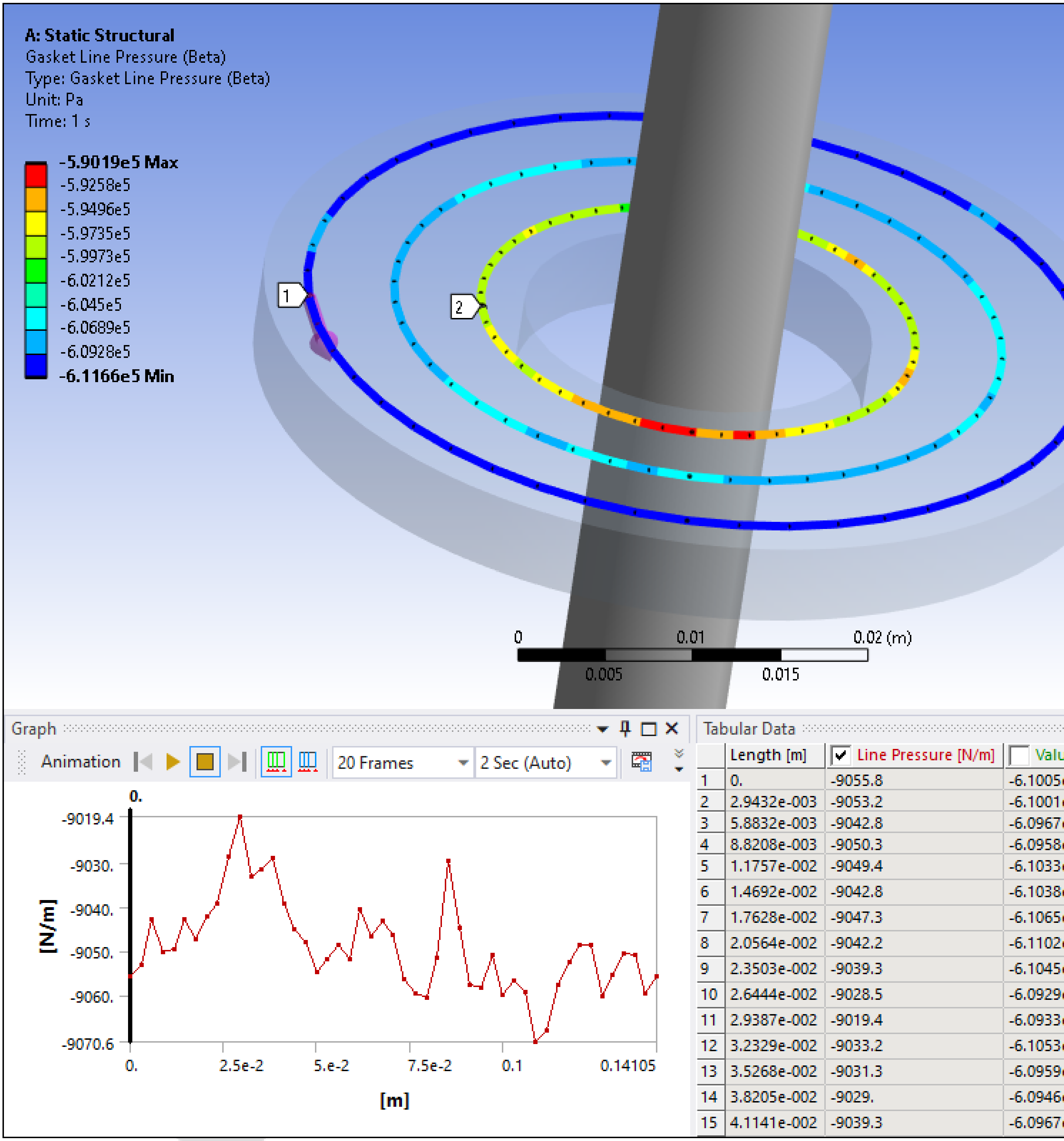
5. Solve the analysis.

Evaluate Result Values

Here is an example of a **Gasket Line Pressure** result. Note the available values. The **Line Pressure** column displays the force created along the center line and each Value columns present the pressure values along each sub-path. The **Graphics** pane displays the force values at each discretization point.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	Length [m]	Line Pressure [N/m]	Valu
1	0.	-9055.8	-6.1005
2	2.9432e-003	-9053.2	-6.1001
3	5.8832e-003	-9042.8	-6.0967
4	8.8208e-003	-9050.3	-6.0958
5	1.1757e-002	-9049.4	-6.1033
6	1.4692e-002	-9042.8	-6.1038
7	1.7628e-002	-9047.3	-6.1065
8	2.0564e-002	-9042.2	-6.1102
9	2.3503e-002	-9039.3	-6.1045
10	2.6444e-002	-9028.5	-6.0929
11	2.9387e-002	-9019.4	-6.0933
12	3.2329e-002	-9033.2	-6.1053
13	3.5268e-002	-9031.3	-6.0959
14	3.8205e-002	-9029.	-6.0946
15	4.1141e-002	-9039.3	-6.0967

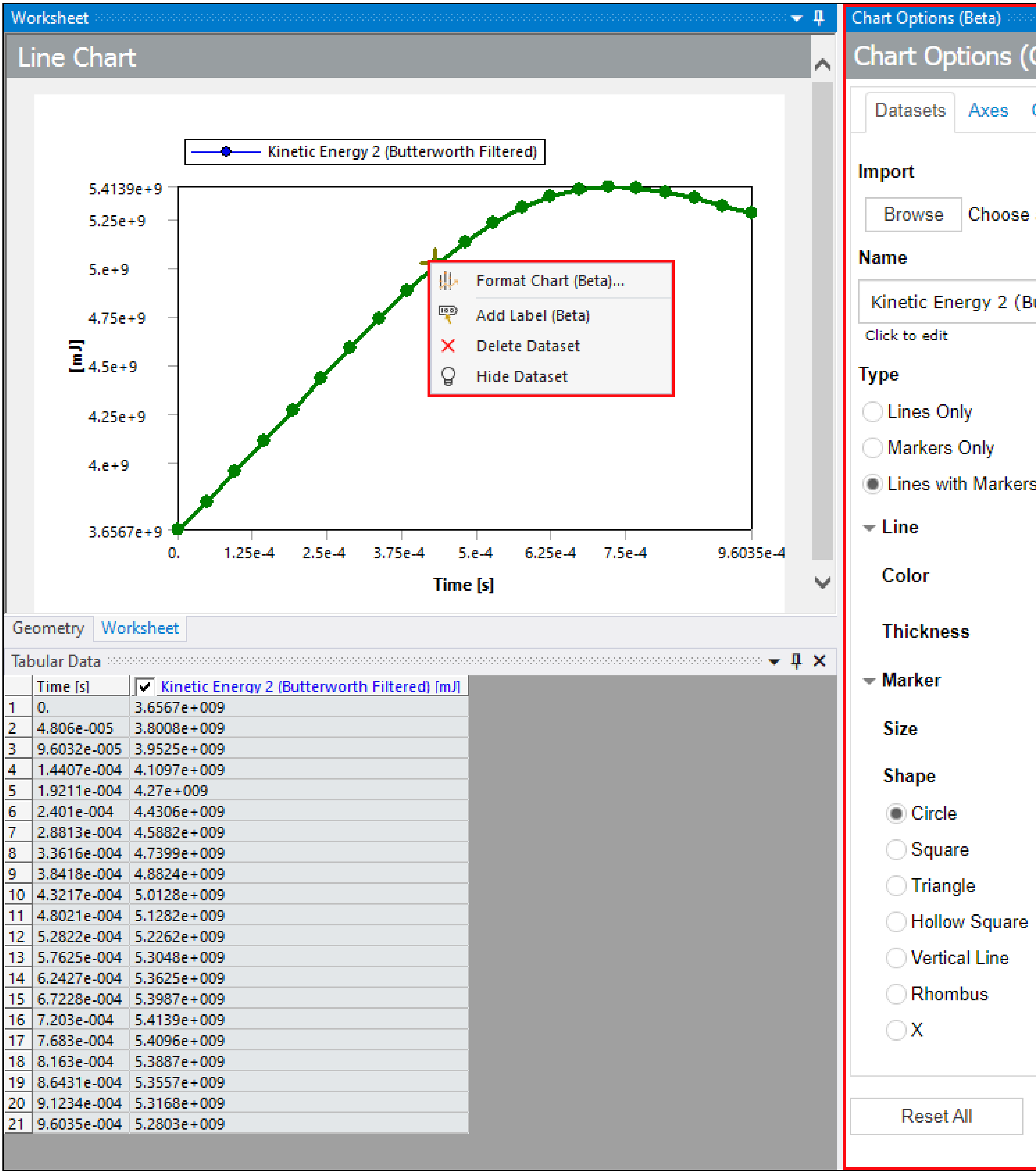
BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 21: Line Chart Options (LS-DYNA Only)

For LS-DYNA analyses, the **Line Chart** feature has a variety of useful Beta Options (p. 1) that enable you to create and work with chart data. New context menu options enable you to label and toggle the display of data included in the chart. In addition, a new **Chart Options** window includes features to import additional datasets and perform various operations and apply filters. This pane also provides options to change the display of your data in the chart and the ability to modify the properties of the chart axes.

BETA



Got to a section topic:

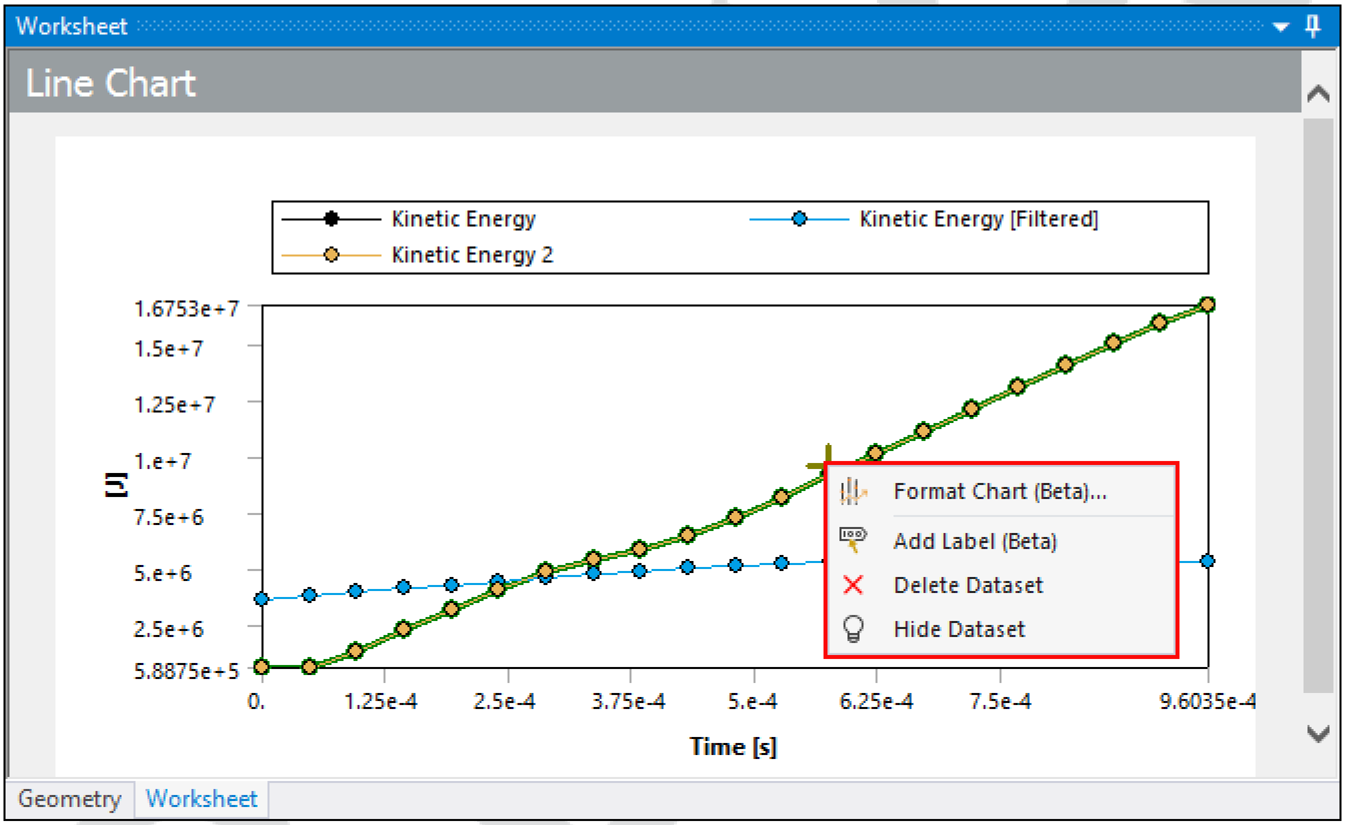
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- [Line Chart Context Menu Options (p. 95)](#)
- [Chart Options Window (p. 96)](#)

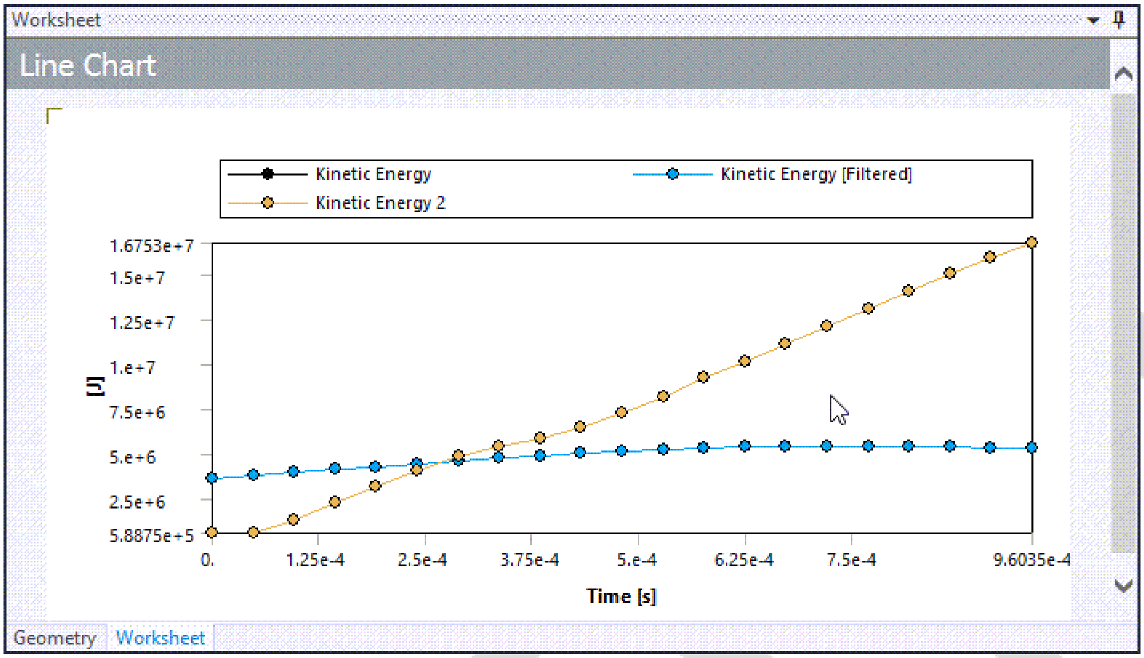
Also see the Line Chart Results (LS-DYNA Only) section in the *Mechanical User's Guide*.

Line Chart Context Menu Options

When you specify a **Line Chart** object, the application displays a graph of the plotted data in the **Worksheet**. The graph includes the following context (right-click) menu options:



- **Format Chart (Beta)**: This option opens the **Chart Options (Beta)** pane.
- **Add Label (Beta)**: Create a label showing the X-Y coordinate of a point selected on the plotted line of a dataset. You can drag and drop labels anywhere within the chart. Once inserted, right clicking a label presents the context menu option **Delete Label (Beta)** in order to remove labels. Also note that you can drag and drop labels anywhere within the chart.



Note:

If you change the unit system of your analysis, the application automatically removes all labels from the chart.

- **Delete Dataset**: Delete a selected dataset.
- **Hide Dataset**: Hide a selected dataset. When you hide a dataset, the context menu then includes the option **Show All Datasets**. Select this option to redisplay any hidden datasets.

Chart Options Window

The **Chart Options** window includes the following tabs. All tabs include the **Reset All** option to reset your specifications.

Datasets Tab

The **Datasets** tab provides the following options:

- **Import**: Browse to and select a new or additional dataset to include in the line chart. The feature supports importing one dataset at a time.
- **Name**: Change the default name given to the dataset(s).
- **Type**: Select the desired display style of the plotted lines and markers used in the chart. You can change the size, color, and shapes of lines/markers.

Axes Tab

The **Axes** tab provides the options **X Axis** and **Y Axis**. Use these option to:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Display grid lines in the chart.
- Display chart data in log10 scale.
- Manually specify the range of the Minimum and/or Maximum values displayed in the chart.
- Edit the labels of either axis displayed on the chart. When you create a line chart in Mechanical, the application applies defaults. When you import datasets, they may already include X-Y axis labels. Either way, you can specify a new and unique label as desired.

Operations Tab

The **Operations** tab provides the following options:

Operation

To specify an **Operation**:

1. Select the **Operation** option button (also know as radio button) and select the desired **Operation** from the drop-down menu. Options include:

   - **None** (default)
   - **Integration**
   - **Differentiation**
   - **Summation**
   - **Subtraction**
   - **Multiplication**
   - **Division**
   - **Scale**
   - **Offset**
   - **Flip XY**

2. Use the **Apply To** options to select the desired dataset(s) on which to apply the operation. Options include:

   - **Current Pick [Name of dataset]** (default): Apply the operation to the currently selected dataset.
   - **All Datasets**: Apply the operation to all of the datasets included in the line chart.
   - **Multi-select**: This option provides a drop-down menu you use to select multiple datasets.

3. Based on the above selection, use either the **Output Dataset Name** or **Output Dataset Name Suffix** field to name your results.

   For the **Current Pick** option, enter a new name in the **Output Dataset Name** field or you can use the application assigned default which is the name of the dataset appended with the name of the operation in parentheses.

   To name the results of operations using the **All Datasets** or **Multi-select** options, enter your desired string in the **Output Dataset Name Suffix** field. The application appends this string to each affected dataset's name. If the operation produces a single output, such as the summation of multiple datasets, the application uses your string as the name of the resulting dataset.

4. Once you have completed defining the operation, select the **Apply** button. The application plots the new dataset on the chart. The application performs the computation on the y-axis for all datasets you have selected in the chart. Each operation uses the x-axis of the first dataset as reference. Other datasets are linearly interpolated.

Note:

Input for the operations can differ. Certain operations require at least one dataset, while others, such as subtraction and division, must have exactly two datasets as input.

Output for the operations may also differ. Certain operations, such as multiplication, can accommodate more than one dataset as input, but it returns only a single dataset as a result. Whereas other operations produce one output for each associated dataset.

Note:

When using the **Division** operation, there are instances when it divides by zero (0) and fails to generate a new dataset.

Filter

To specify a **Filter**:

1. Select the **Filter** option button (also know as radio button).
2. Select the desired **Filter** option from the drop-down menu. Options include:
   - **None** (default): Reset any previous filters and redisplay the plot of the original dataset.
   - **Butterworth**: Apply a Butterworth filter to create and plot a new dataset.
   - **Butterworth LSPP**: Apply a Butterworth LSPP filter to create and plot a new dataset.
   - **SAE**: Apply a SAE filter to create and plot a new dataset.
3. When you specify a **Filter** type option, the **Cutoff Frequency (Hz)** property also displays. Use this property to define a cutoff frequency value for the applied filter. The default setting is **0.1**.
4. Use the **Apply To** options to select the desired dataset(s) on which to apply the filter. Options include:
   - **Current Pick [Name of dataset]** (default): Apply the operation to the currently selected dataset.
   - **All Datasets**: Apply the operation to all of the datasets included in the line chart.
   - **Multi-select**: This option provides a drop-down menu you use to select multiple datasets.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Based on the above selection, use either the **Output Dataset Name** or **Output Dataset Name Suffix** field to name your results.

   For the **Current Pick** option, enter a new name in the **Output Dataset Name** field or you can use the application assigned default which is the name of the dataset appended with the name of the filter in parentheses.

   To name the results of a filter using the **All Datasets** or **Multi-select** options, enter your desired string in the **Output Dataset Name Suffix** field. The application appends this string to each affected dataset's name.

6. Once you have completed defining the filter, select the **Apply** button.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

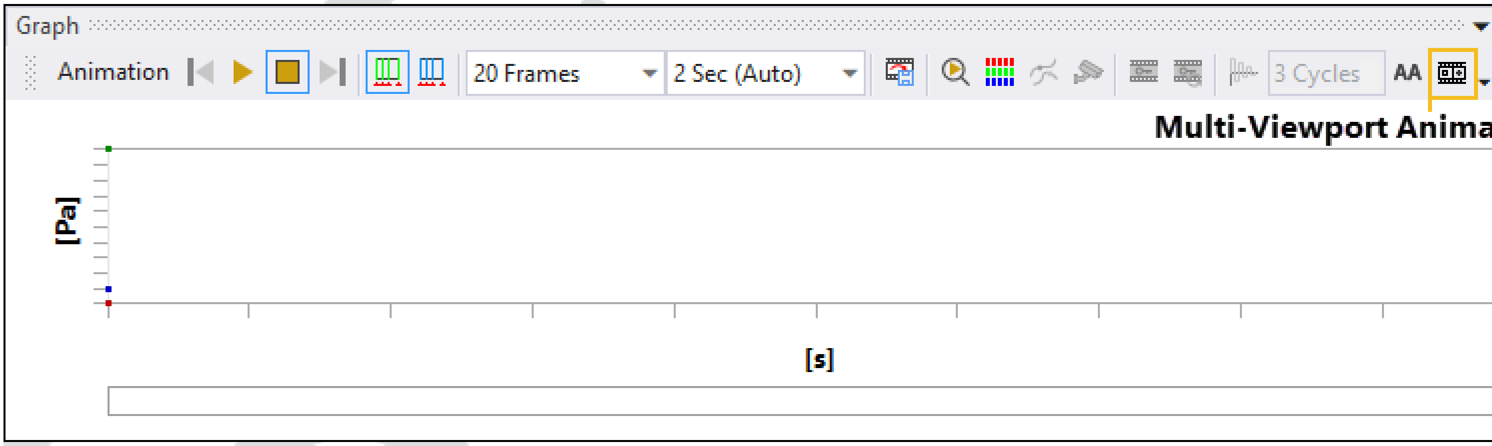
Chapter 22: Animating Results in Multiple Viewports

If you are using a Windows system that supports OpenGL 4.3, and the **Beta Option** is turned on, you can animate (multiple) results in multiple **Viewports**. The **Viewports** feature enables you to split the Geometry window into multiple windows, up to four, and select independent result objects in each window.

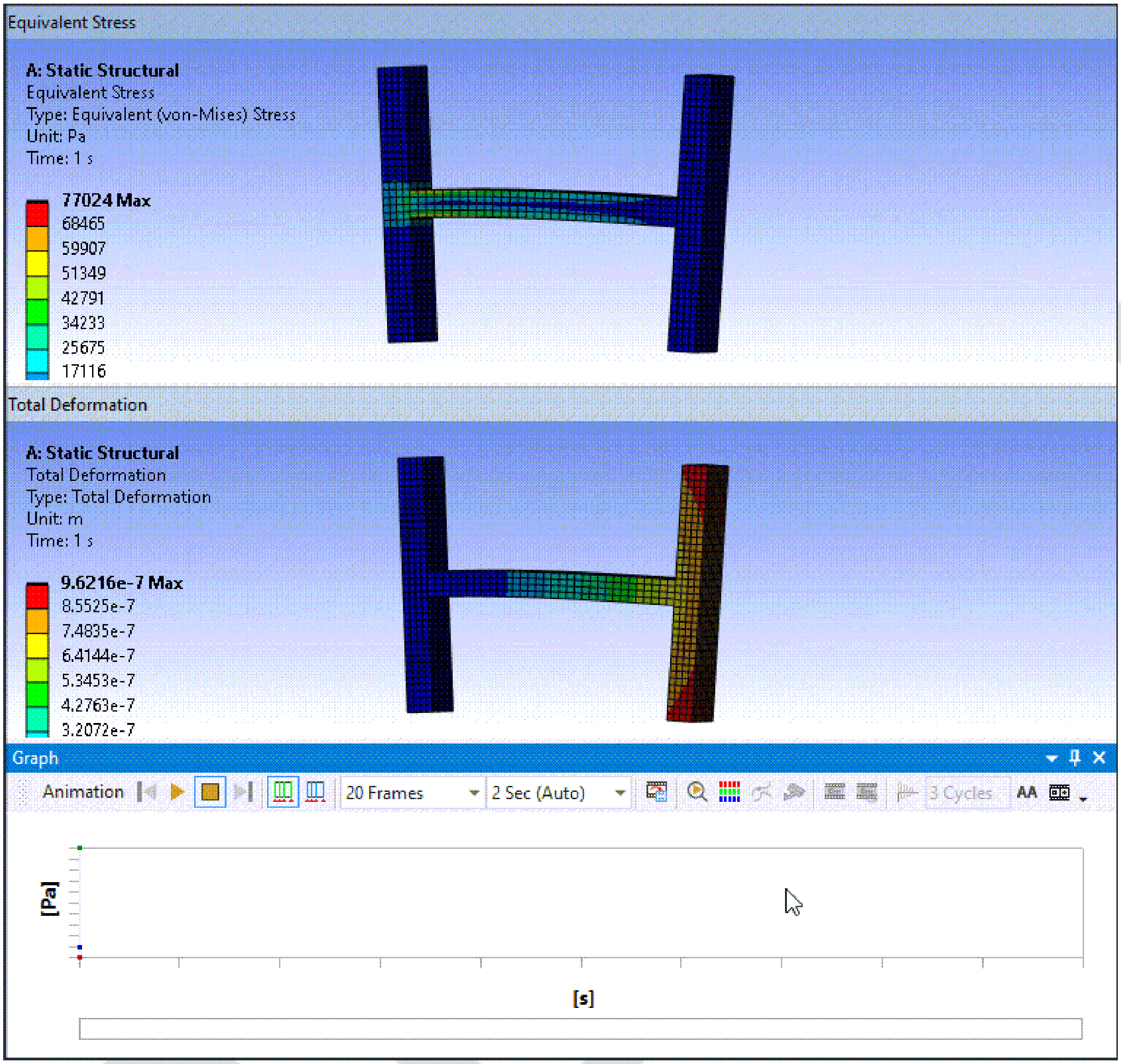
> **Note:**
>
> The results you select must support **Accelerated Animation**.

> **Note:**
>
> As needed, see the Activate Beta Options (p. 1) section to make the feature available.

Once you have displayed your evaluated results in multiple viewports, select the **Multi-Viewport Animation** option on the **Animation** toolbar to begin the animation.



Here is an example of the feature in use.



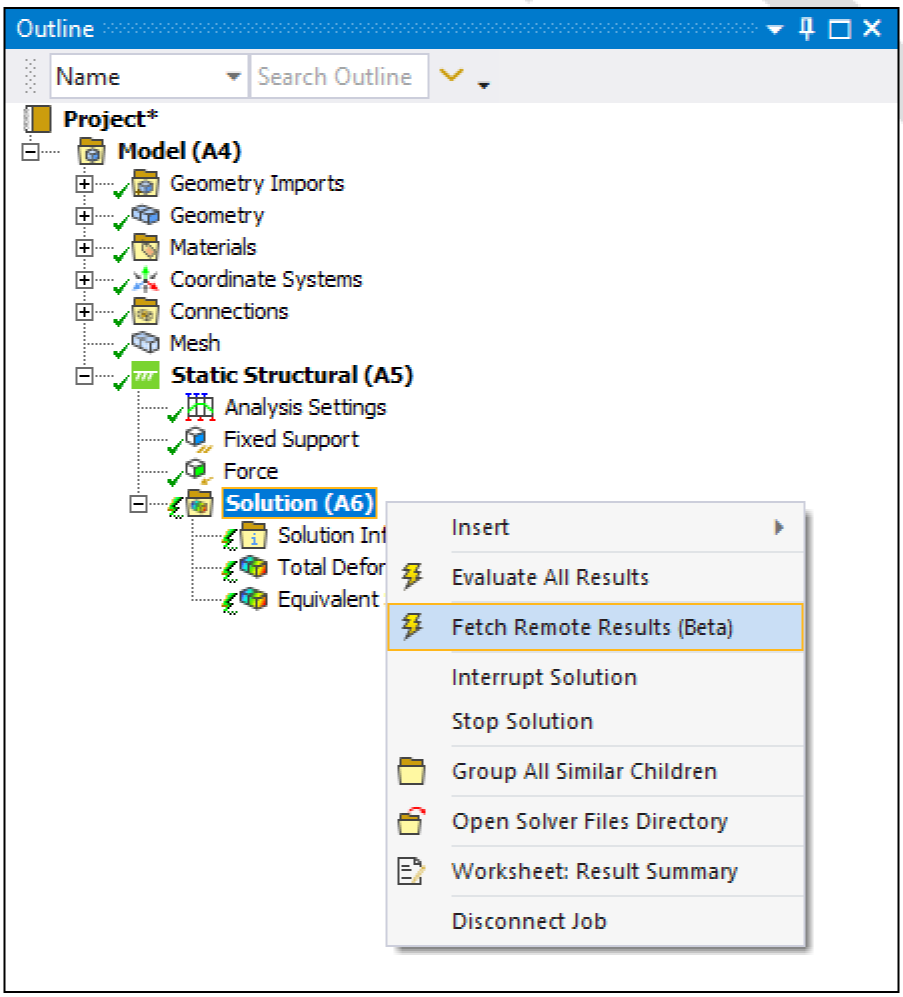
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 23: Fetching an RSM Result

Summary

Currently, when you are 1) performing a multistep solution, 2) using the Mechanical APDL Solver, 3) using the Remote Solve Manager (RSM), there is a beta option that enables you to retrieve load step-based results during the solution process.

Once you have properly configured your remote resource using the RSM Configuration Application, enabled the Remote Post feature, queued your solution in Mechanical (and waited for it to start), the **Solution** object provides the context (right-click) menu option: **Fetch Remote Results** (illustrated below). This option enables you to display result content for the most recently completed load step while the solution is running.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compatibility Requirements

Review the following compatibility requirements for the use of this feature. The Remote Post system:

- Requires enabling the beta key and feature flag prior to starting a solution (see next section).
- Will only work on cluster setups that support the IBM Platform MPI for inter-node communication.
- Does not encrypt or authenticate internal data connections. May not be appropriate for use on unsecured networks.
- Selects a random port when initiating a connection between the server and Mechanical. Network firewalls that block certain ports may interfere with functionality.
- Does not support Probes or Toolbox results.

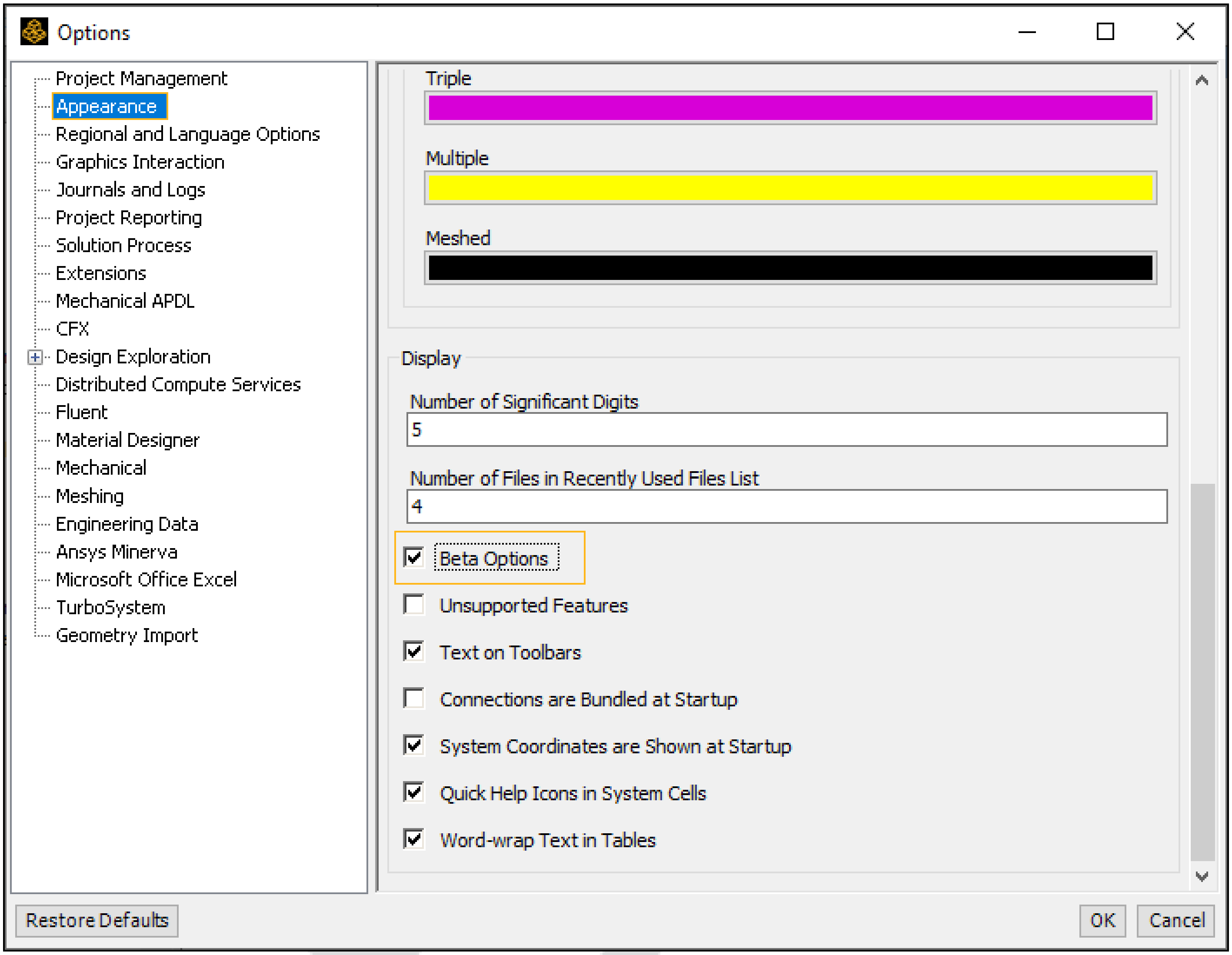
Activate Beta Key and Option

Activate this feature from the Command Prompt. At the command prompt, type **`RunWB2.exe`** and include the **`'-k Mechanical.RemotePostRSM'`** argument. This turns on the feature flag and opens the Workbench application.

On the Workbench Project page, select the **Tools** menu and then **Options**. Select the **Appearance** category, scroll down the dialog, select **Beta Options**, and select **OK**.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

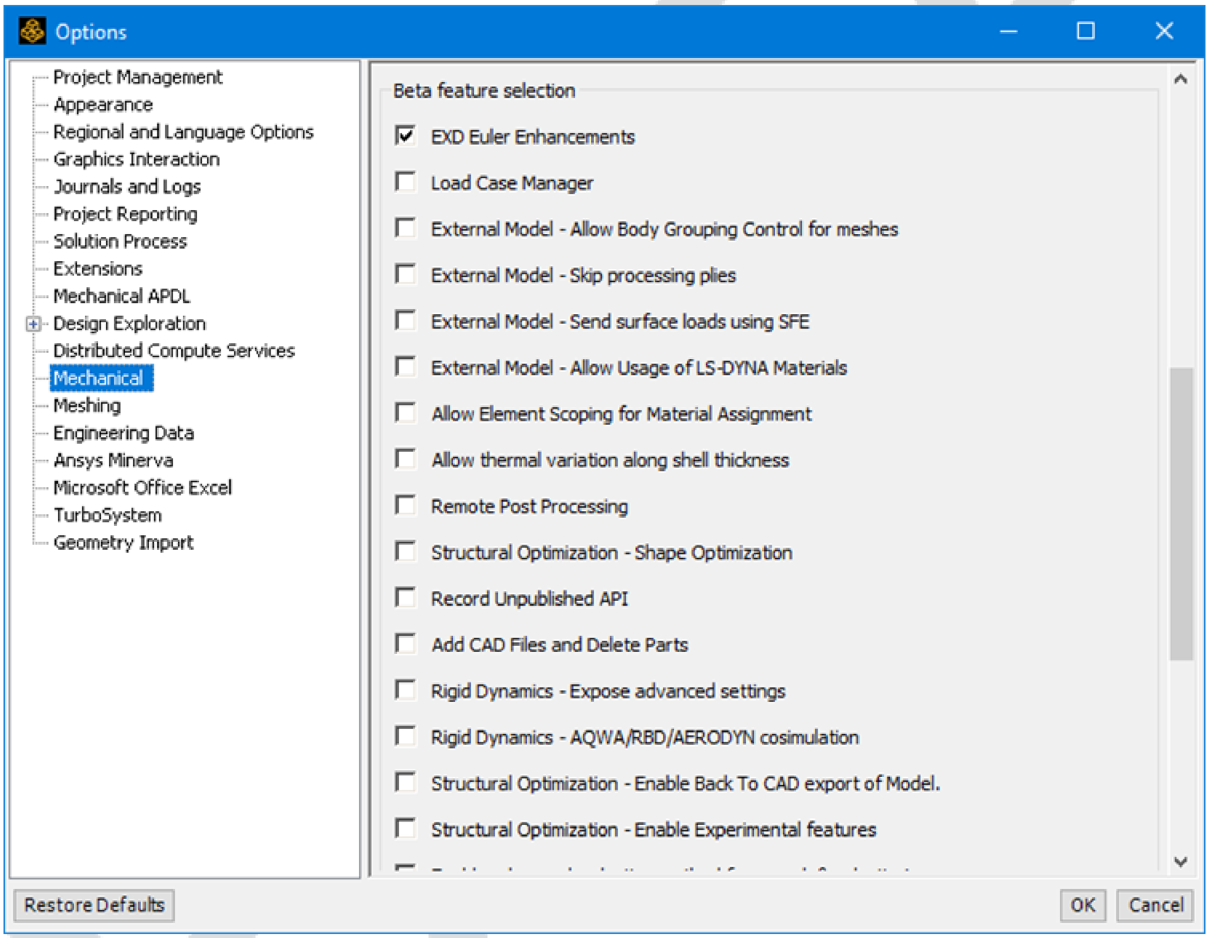

Options
Project Management
Appearance
Regional and Language Options
Graphics Interaction
Journals and Logs
Project Reporting
Solution Process
Extensions
Mechanical APDL
CFX
Design Exploration
Distributed Compute Services
Fluent
Material Designer
Mechanical
Meshing
Engineering Data
Ansys Minerva
Microsoft Office Excel
TurboSystem
Geometry Import
Triple
Multiple
Meshed
Display
Number of Significant Digits
5
Number of Files in Recently Used Files List
4
Beta Options
Unsupported Features
Text on Toolbars
Connections are Bundled at Startup
System Coordinates are Shown at Startup
Quick Help Icons in System Cells
Word-wrap Text in Tables
Restore Defaults
OK
Cancel

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 24: Explicit Dynamics Blast Analysis in Mechanical

An initial prototype exposing the Autodyn blast analysis functionality in the Explicit Dynamics analysis system is available through the EXD Euler Enhancements beta feature flag. To set this flag in Workbench, open the **Tools** → **Options** dialog. Check the box next to the option in the **Mechanical** tab.



The functionality exposed under this beta feature for release 2025 R1 is as follows:

- Two different domain types for 2D Euler analyses:
  - 1D wedge domain for performing a 1D blast
  - 2D grid domain
- Exposure of the Detonation Point in 2D
- Output of data files (with file extension `.fil`) of the results of a 2D axisymmetric MME analysis to be used for remapping into a 3D MME analysis

- Remapping from a 2D data file (extension `.fil`) into a 3D MME analysis in Mechanical exposed via an ACT extension

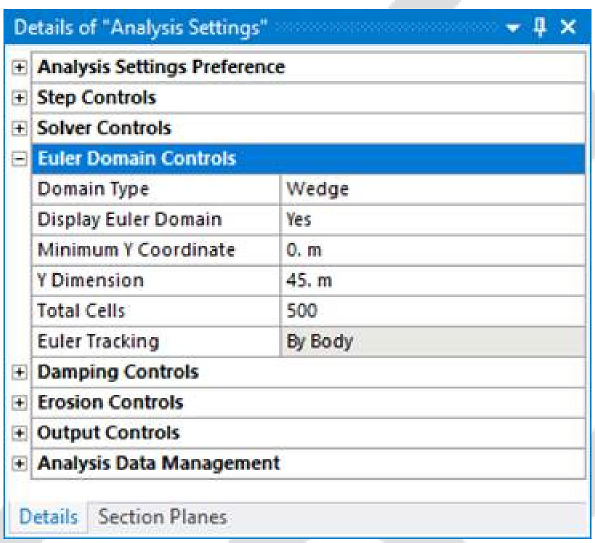
For more information about the above functionality and EulerRemap object limitations, see the following sections:

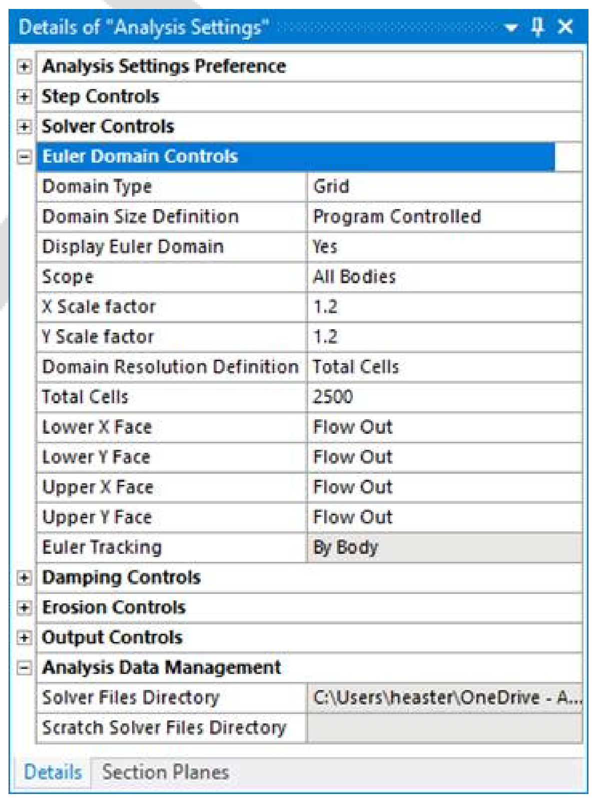
24.1. Domain Types for 2D Multi Material Euler Solver

The 2D Multi Material Euler (MME) solver from the Ansys Autodyn solver has been exposed in the Mechanical interface. Two domain types can be created:

- A wedge for performing 1D blast analyses
- A grid (box) for performing 2D MME analyses

The domain type is determined by the Euler Domain Controls in the analysis settings.





If the **Domain Type** is set to Wedge, the virtual MME domain created is a 1D wedge. For how to set up a 1D wedge analysis, see Guidance on Setting up a 1D Wedge Analysis in Mechanical (p. 109).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the **Domain Type** is set to Grid, the default virtual MME behaviour is activated: the virtual MME domain created is a box and the controls work the same as for 3D virtual Euler MME domains with full Euler-Lagrange coupling turned on in the Explicit Dynamics solver.

The option to output data files containing the information needed to remap the results from a 2D axisymmetric MME domain into a subsequent analysis has been added to the Output Controls. The option is **Output Remap File** and can be set to Off, On or Program Controlled. If set to Off, no data files will be produced. If set to On, data files will be produced containing the data from any 2D MME domains, at the same frequency as the results files. If set to Program Controlled, the data files will be produced if there is a virtual MME domain of type Wedge, but will not be produced if there is no virtual MME domain or if there is a virtual domain of type Grid.

The file name of the data files is of the form `remap_{cycle}.fil`, where *{cycle}* is the cycle number at which the data file was produced. The data files can be used to initialize an 3D MME analysis in Mechanical using the provided ACT extension (see Using the Euler Remap ACT Extension (p. 111)). They are also of the correct file format to be used to fill Multi Material Euler and Euler Ideal Gas parts in the Autodyn Component System.

24.2. Guidance on Setting up a 1D Wedge Analysis in Mechanical

First create an Explicit Dynamics system in the Workbench Project Schematic. Open the Properties for the Geometry cell and set the **Analysis Type** under **Advanced Geometry Options** to 2D, and create or import an appropriate geometry. For simple 1D wedge analyses consisting of a sphere of explosive surrounded by the ambient material, a sample geometry has been provided with in `%AWP_ROOT212%\aisol \Samples\AUTODYN`. To use the sample geometry, import the file `1D_wedge_geometry_for_blast.agdb`. To change the radius of the explosive material, or the blast radius, modify the parameters ExplosiveMaterialRadius or BlastRadius in DesignModeler.

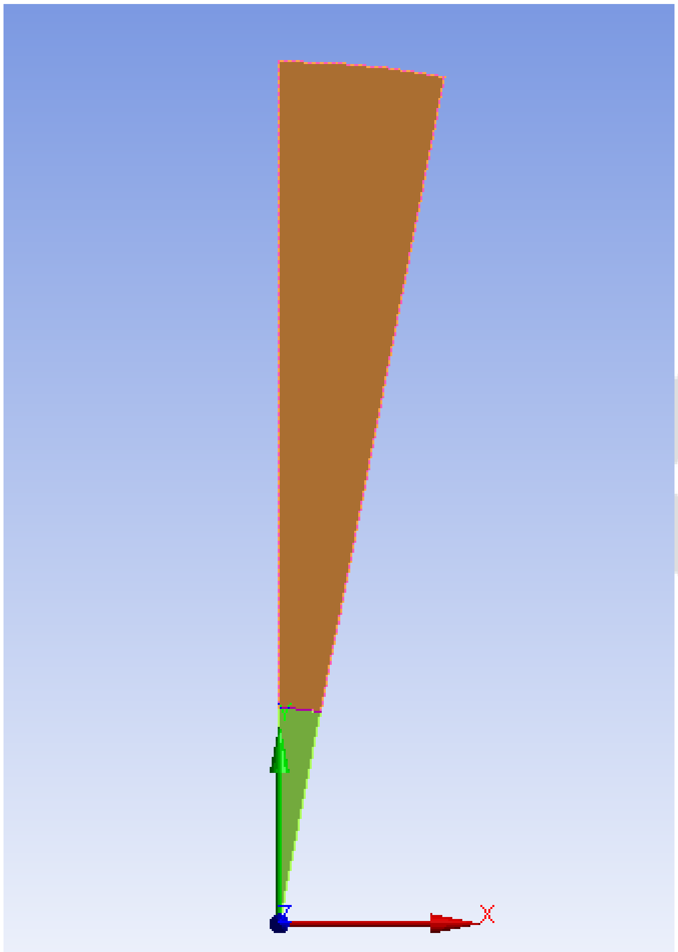
If you need to create your own geometry, it is important to understand that the geometry of the virtual MME wedge domain that will be created in the Explicit Dynamics solver. This is because the created bodies designed to fill this virtual MME domain must cover the virtual MME domain completely (else there will be void regions).

The wedge domain created in the solver will begin at the global origin and will run along the axis of symmetry (see the following note (p. 109)). The wedge has an angle of 10 degrees. The bodies with **Reference Frame** set to Eulerian (Virtual) must cover this region in space. An example of an appropriate geometry is shown in Figure 24.1: Appropriate Geometry for a 1D Wedge Analysis (p. 110).

Note:

For users familiar with the Autodyn user interface, in Autodyn 2D axisymmetric models, the axis of symmetry lies along the global X-axis. This is contrary to 2D axisymmetric models in Mechanical, where the axis of symmetry lies along the global Y-axis. Therefore, all Explicit Dynamics 2D axisymmetric models created in Mechanical must be set up assuming the axis of symmetry is along the global Y-axis. This is appropriately handled in the Explicit Dynamics solver by performing a coordinate transformation so that the axis of symmetry is mapped onto the global X-axis.

Figure 24.1: Appropriate Geometry for a 1D Wedge Analysis



The geometry that makes up the wedge consists of two bodies: one to model the explosive (the smaller green colored body), and the other to model the ambient material in which the explosion is taking place (the larger brown colored body). Be sure that these two bodies do not overlap; otherwise the fill into the Eulerian domain may not be as expected. The coordinate system displayed is the Global Coordinate System. Note how the wedge is aligned with the global Y axis, and that the two bodies together effectively form a sector of a circle centered at the global origin with a 10 degree angle at the center of the sector. The shape shown here will also be the shape of the MME domain.

Once the geometry has been created or imported, the remainder of the model can be set up in Mechanical.

1. For a 1D wedge analysis, the **2D Behavior** of the Geometry must be set to axisymmetric. The bodies need to have the **Reference Frame** set to Eulerian (Virtual).

2. You should delete all objects under the Connections folder, including Contact Regions and Body Interaction objects, since the analysis will be purely Eulerian.

3. Generate the mesh. The default mesh settings should be sufficient.

4. Insert a Detonation Point at X = 0, Y = 0.

5. Set the Analysis Settings:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Step Controls**:
  - **Time Step Safety Factor** = 0.6666.
- **Solver Controls**:
  - **Detonation Burn Type** = Program Controlled (for 2D analyses, this defaults to Direct) or Direct.
- **Euler Domain Controls**:
  - **Domain Type** = Wedge.
  - **Minimum Y Coordinate** = 0. M.
  - **Y Dimension** - set to blast radius.
  - **Total Cells** - set the total number of cells along the wedge. The resulting cell size will be the Y Dimension divided by the Total Cells. Note that the more cells, the more accurate the simulation, but the longer the run-time. It is recommended that the cell size is such that there are at least 10 cells across the radius of the explosive material.

    Note:

    No grading of the mesh has been exposed at release 2025 R1. This is being considered for a future release.

- **Output Controls**:
  - **Output Remap File** – Set this to Program Controlled or On to trigger the output of data files (`.fil` file format) to be used to remap the solution into a 3D analysis. The frequency of the data files produced is the same frequency as the result file output. The files produced are named `remap_{cycle}.fil`, where *cycle* is the cycle number that the data file is produced.

    You can set **Output Remap Files** to On for other Eulerian domain types if the **2D Behavior** is set to Axisymmetric - otherwise no remap files will be output (Program Controlled regulates this behavior as well).

    If there are no Euler domains present in the model, setting **Output Remap Files** to On is ignored (a warning message is generated).

Prior to using any generated data files for remapping, you should inspect the results in order to ensure that the blast wave has not reached the extents of the wedge at the cycle where the results will be transferred during the remapping. This can be done by inserting User Defined results for variables such as PRESSURE or INT_ENERGYALL, or by adding Pressure or Energy trackers.

24.3. Using the Euler Remap ACT Extension

The Euler Remap ACT Extension included with the standard Ansys installation provides a means to remap results produced in a 1D wedge blast analysis into a 3D analysis containing an Euler Multi Material virtual domain in an Explicit Dynamics analysis in Mechanical. The extension can only be used when the EXD Euler Enhancements beta feature flag is switched on.

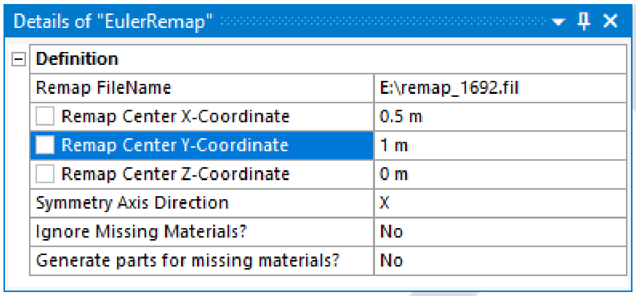
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The EulerRemap extension must be loaded before it can be used. Select **Extensions** → **Manage Extensions...** in Workbench and select the check box next to the extension to load it. Then add a 3D Explicit Dynamics system to the project and create or import your geometry, and define the materials used in Engineering Data. Once the extension is loaded and the materials and geometry have been fully defined, the extension is ready to be used in Mechanical.

Note:

The extension will currently only work for 3D analyses.

To remap the data from a data file (`.fil` file), set up the analysis so that a virtual Euler domain will be created. Then right-click the Explicit Dynamics system in the Project and insert an EulerRemap object. The EulerRemap object has the following fields:



- **Remap FileName** – Choose the data file to use for the remapping. The data file could be from a 2D MME analysis performed in an Explicit Dynamics system, or from an analysis performed in an Autodyn component system.
- **Remap Center X/Y/Z-Coordinate** – Enter the coordinates in the 3D analysis where the origin of the 2D analysis should be mapped.
- **Symmetry Axis Direction** – Choose which axis in the 3D model aligns to the axis of symmetry from the 2D axisymmetric analysis.
- **Ignore Missing Materials?**
  - Yes – Any materials that are in the data file that do not exists in the 3D model (meaning, they are not defined under the Materials object in Mechanical and they are also not assigned to a body) will not be remapped.
  - No – If there are any materials in the data file that do not exist in the 3D model (meaning, they are not defined under the Materials object in Mechanical and they are not assigned to a body), the 3D model will issue an error message and will not run.
- **Generate parts for missing materials?**
  - Yes – If any materials defined under the Materials object in the 3D analysis are not assigned to a body in Mechanical, the extension will automatically create a construction geometry part for that material. See the notes (p. 113) on automatically generated parts.
  - No – Do not automatically generate parts for missing materials. See the notes (p. 113) on automatically generated parts.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes on Automatically Generated Parts

When **Generate parts for missing materials?** is set to Yes, construction geometry parts are created for existing materials in the project that are not assigned to a body. This ensures that these materials can be successfully remapped into the 3D model, allowing them to be post-processed in the 3D model in Mechanical. The autogenerated parts are created when you click **Solve**. Each part created is a cube located at the remap center. The dimensions of the cube are chosen such that the part should have no impact on the results of the analysis.

The parts are created using Construction Geometry Solids and are given a name of the format AutoGenerated_DummyBody_{*material_name*}, where {*material_name*} is the name of the material for which the dummy part is created. The parts created from the construction geometry solid are set to Eulerian (Virtual). Additionally, a Named Selection for each dummy part is created to assist with the scoping of results to these parts for Post Processing.

24.4. Limitations

- EulerRemap object limitations:
  - Only one EulerRemap object is permitted per analysis
  - The object can be used in 3D analyses only
- Any objects generated by the EulerRemap object at solve time (for instance auto generated parts) do not get deleted when you use Clear Generated Data.

BETA

BETA

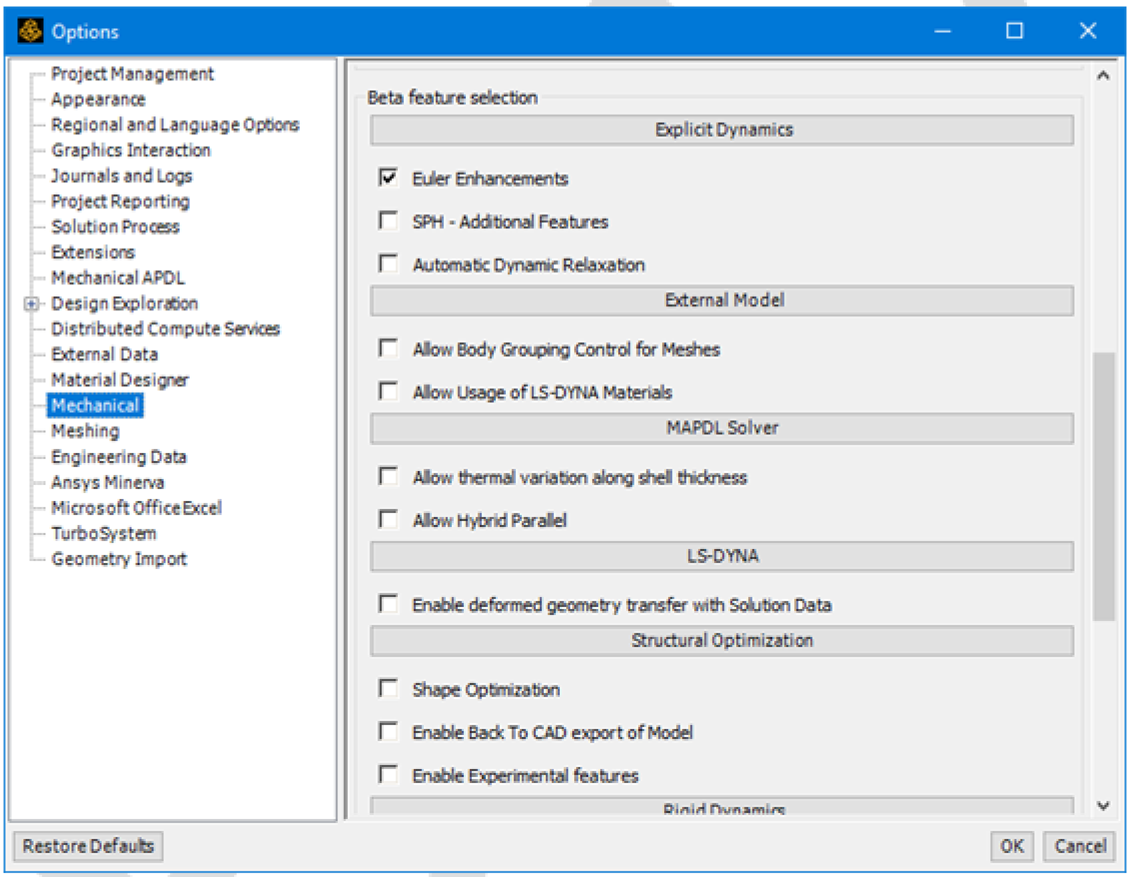
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 25: Virtual Eulerian Surface Bodies

An initial prototype, which exposes the use of (Virtual) Eulerian surface bodies in the Explicit Dynamics analysis system, is available through the Euler Enhancements beta feature flag. To set up this flag in Workbench, follow these steps:

1. Open the **Tools** > **Options** dialog.

2. Check the box next to the option in the **Mechanical** tab.



- Two different (Virtual) Eulerian surface body options are supported for mapping material and initial conditions into 3D Euler:
  - Volume filling of the Eulerian domain, using closed single connected surface bodies
  - Directional filling of the Eulerian domain, using surface bodies with a defined filling direction

Mapping of Surface Bodies with Euler Reference Frame to Virtual Euler Domain

- The standard mesh, which is generated on surface bodies marked with Eulerian (virtual) reference frame, is only used to represent the geometry of the surface body during initialization of the model for the solver. The material and initial conditions defined on surface bodies, which are marked as an Eulerian reference frame, are mapped to the Euler domain. The mesh associated with the original surface body is then deleted, prior to the solve. A unique material is created for each surface body that is mapped into the Euler domain for post processing.

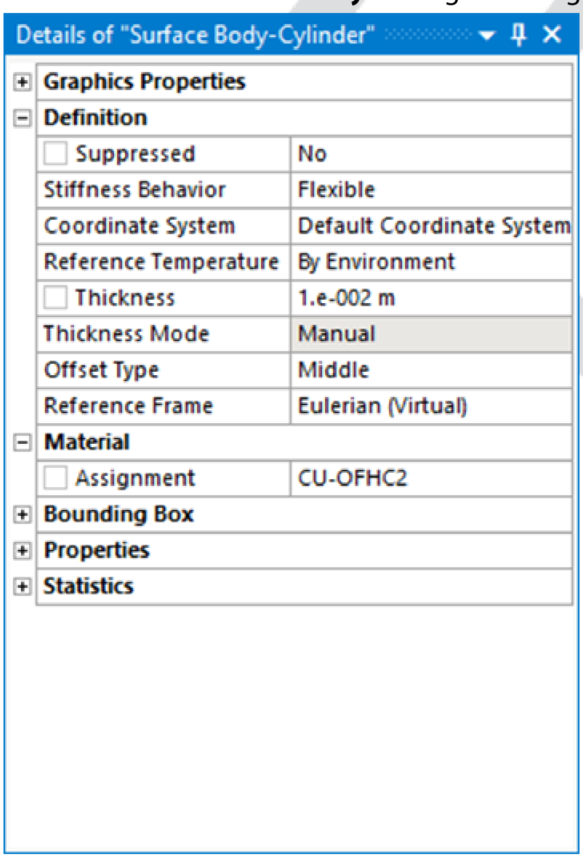
- If multiple solid and surface bodies marked as **Eulerian** overlap, the body higher in the **Outline** view will take precedence. Therefore, the material assigned to the region of overlap will correspond with the material assigned to the first Eulerian body.

For more information about the Eulerian surface body options and the limitations, see the following sections:

25.1. Closed Single Connected Euler Surface Bodies

The domain of the 3D Multi Material Euler (MME) solver can be filled with material and initial conditions using a closed single connected surface body. The interior of the closed surface body inside the Eulerian domain will be filled with the material and initial conditions of that surface body.

A surface body is defined as a 3D (virtual) Euler body, when the Reference Frame of that surface body is switched in the **Geometry** setting from Lagrange to Eulerian (Virtual).



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Not only does this definition trigger the creation of the virtual MME domain, but the interior volume of the closed surface body is used to fill the virtual MME domain with the material and initial condition assigned to it.

The surface must be single connected. This means that the surface consists of faces which only connect a maximum of two faces along their edges, and no T-joints or multiple joints are allowed.

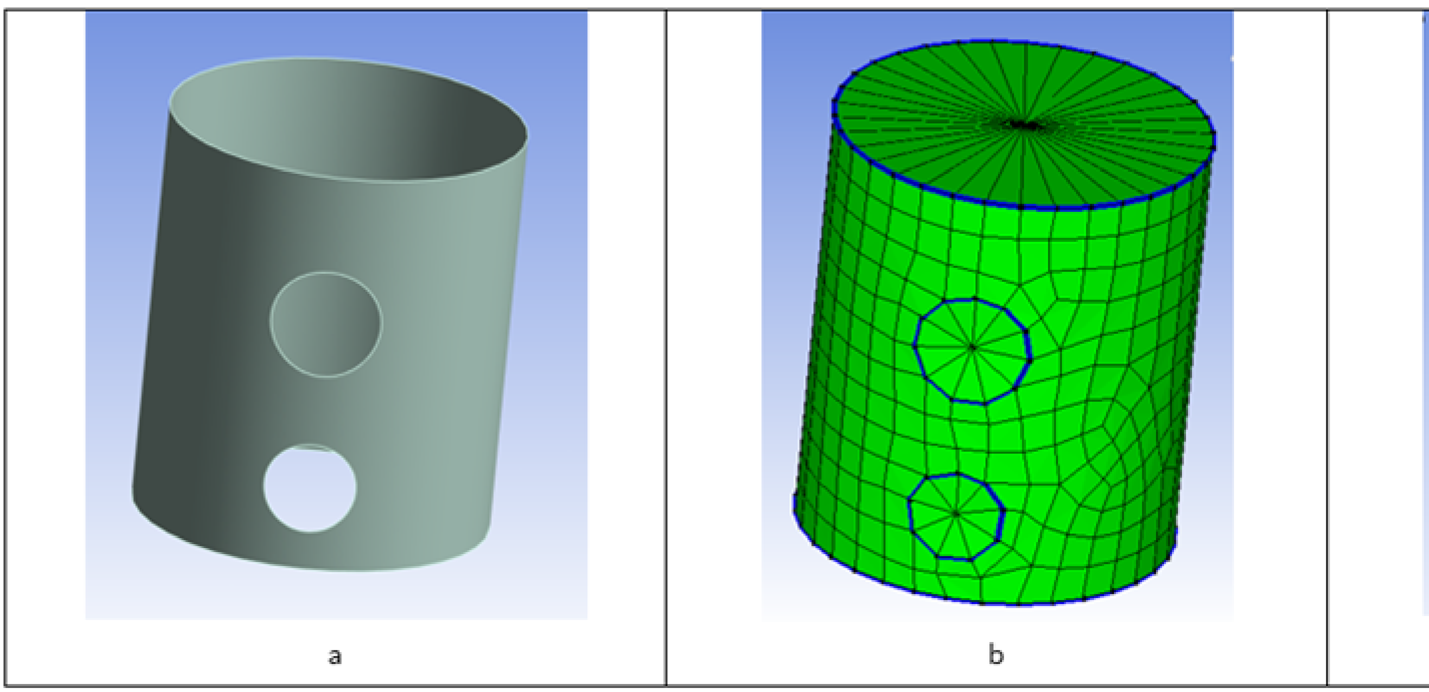
Note:

The thickness of the Eulerian surface body is not taken into account to compute the internal filling volume. However, it needs to have a non-zero thickness value assigned.

25.1.1. Closed Single Connected Euler Surface Bodies with Holes or Openings

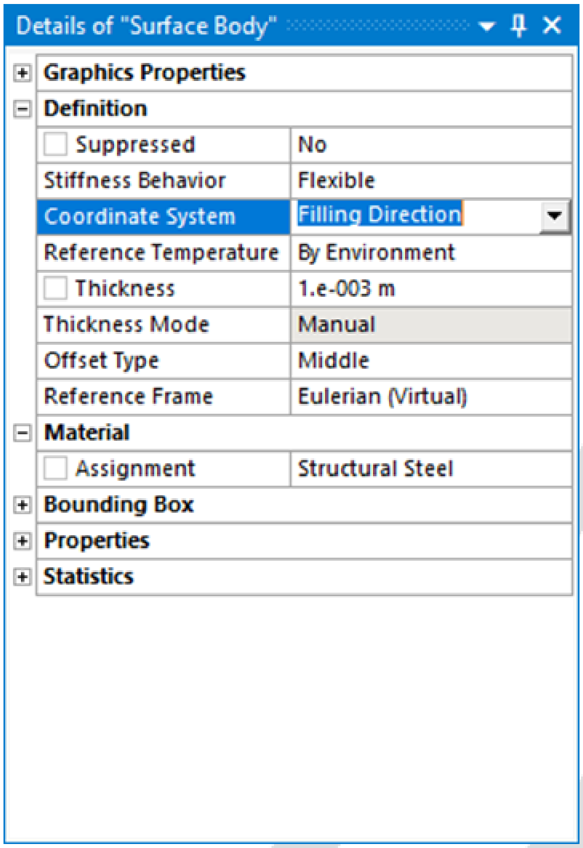
The closed single connected surface, that is used to map material and initial conditions into the MME domain, may contain holes or openings. The mapping algorithm will generate a completely closed Euler surface body by closing the holes and openings with triangular elements. The triangular elements are generated from the nodes along the edge of a hole or opening and a central node in the middle of the hole or opening, that has the average x-, y-, and z-coordinate of all nodes along the edges of the hole or opening.

Figure 25.1: Example of (a) Euler Surface with Holes and Openings, (b) Closing of Holes and Openings in Euler Surface, and (c) Resulting Material Mapping into MME Euler Domain



25.2. Single Connected Euler Surface Bodies with Filling Direction

The domain of the 3D Multi Material Euler (MME) solver can be filled with material and initial conditions using a single connected surface body in combination with a filling direction. The filling direction is defined by the direction of the z-axis of a local coordinate system, which is scoped to the Eulerian filling surface. To be used as a filling coordinate system, the origin of the local coordinate system should not coincide with the origin of the global coordinate system.

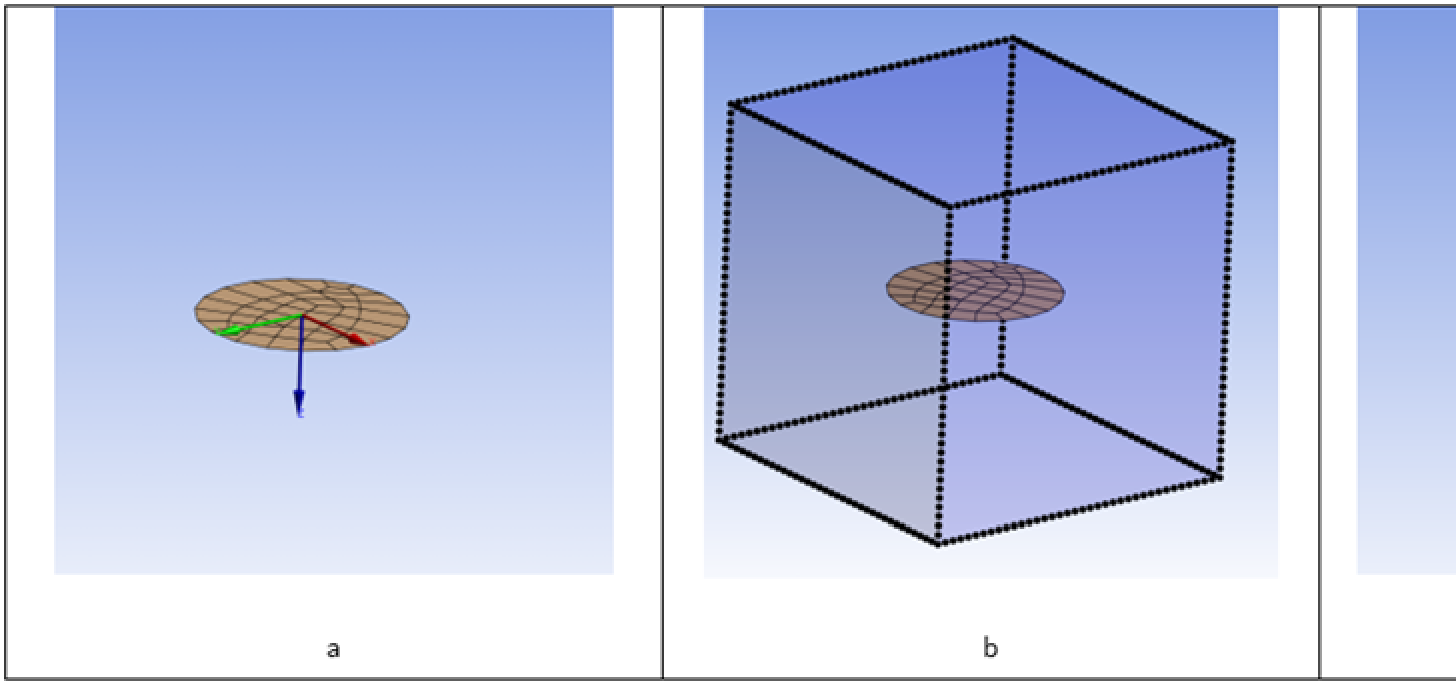


The extruded volume of the Euler surface body in the z-direction of the selected local coordinate system inside the 3D Multi Material Euler (MME) domain will be filled with the material and initial conditions of the surface body.

The extruded volume will extend to the boundary planes of the 3D Multi Material Euler (MME) domain.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.2: Example of (a) Euler Surface Body with Defined Filling Direction , (b) Location of Euler Surface Body in MME Euler Domain, and (c) Resulting Material Mapping into MME Euler Domain



25.3. Limitations

- The following are some Euler surface body limitations:
  - Eulerian surface bodies are used in 3D analyses only.
  - Eulerian surface bodies are deleted after mapping of the material and initial conditions to the Euler MME domain, and are not available for structural analysis.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 26: Command Objects for SPH Settings

These beta command objects are available as an option to control the solution settings for the SPH solver in Explicit Dynamics analyses. Although the default options are usually sufficient for many applications, these command objects may be used to overcome any numerical problems exhibited by an analysis, such as tensile instability. For further information, see Command Objects in Explicit Dynamics.

To access the command objects, turn on **Beta** in the Workbench options.

SPHSETTINGS,DENSITYCALC,CONTINUITYEQ	Calculate the density using the continuity equation (Default)
SPHSETTINGS,DENSITYCALC,KERNELSUM	Calculate the density using a kernel sum
SPHSETTINGS,SMOOTHINGLENGTH,CONSTANT	Use a constant smoothing length (Default)
SPHSETTINGS,SMOOTHINGLENGTH,VARIABLE	Use a variable smoothing length
SPHSETTINGS,SMOOTHINGFUNCTION,CUBICSPLINE	Use the cubic B-spline as the weighting function (Default)
SPHSETTINGS,SMOOTHINGFUNCTION,QUINTICSPLINE	Use a quintic spline as the weighting function
SPHSETTINGS,VISCOSITYOPTION,SPHWITHSHEARCORRECTION	Use the Monaghan type artificial viscosity (Default)
SPHSETTINGS,VISCOSITYOPTION,STANDARD	Use the Neumann and Richtmyer artificial viscosity

Density Calculation

Two alternative methods for calculating density have been implemented—the **continuity equation** is the default method. Calculating the density by kernel sum is activated using the beta command object shown in the table above. The default option is the most common found in the literature, and it works well for a wide range of applications. However, under certain circumstances, inconsistencies in the SPH equations may result in negative densities and numerical issues. In such cases, the density calculation by kernel sum may overcome such problems.

Variable Smoothing

The accuracy and robustness of the SPH processor depends on the quality (in particular, the number) of the local neighboring particles. In expansive flow of material, the distance between SPH particles increases. If this distance exceeds twice the smoothing length of the particles, the two particles will no longer interact. This loss of interaction may be unphysical and is commonly described as 'numerical fracture'.

In an attempt to reduce the problem of numerical fracture, an option to add a variable smoothing length has been included as a beta functionality, which can be activated by a command object. When

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

activated, as particles separate and their density decreases, their smoothing length increases so that interaction with neighboring particles is maintained.

Since changes in the smoothing length result from changes in density, the variable smoothing option works well for isotropic flows. However, for many continuum dynamics applications, anisotropic flow is observed. For example, in plastic flow of metals, the volume (hence density) is almost constant while the deformations are grossly anisotropic. The current implementation of variable smoothing does not work well in such situations and numerical fracture is often unavoidable.

SPH Artificial Viscosity

There are two types of artificial viscosity available for the SPH solver. The default option is the Monaghan-type artificial viscosity, which is described in Damping Controls. This is the recommended formulation to use. The Neumann and Richtmyer form of artificial viscosity is the second option. Refer to the table above to use the respective command objects.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 27: Export Material Calibration Results as New or Updated Materials in Mechanical

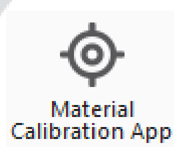
A beta feature allows you to save material calibration results as new materials when you run the Ansys Material Calibration App from within Mechanical.

Use the Ansys Material Calibration App to calibrate your test data against a material model. You can then export the results directly into your model as a new material, merge properties that were recorded under different temperatures, or replace the properties of an existing material with calibration results. This skips the step of accessing the Engineering Data workspace to add or update a material in your model.

> **Note:**
>
> As needed, see the Activate Beta Options (p. 1) section to make the feature available.

Export Material Calibration Results as New or Updated Materials

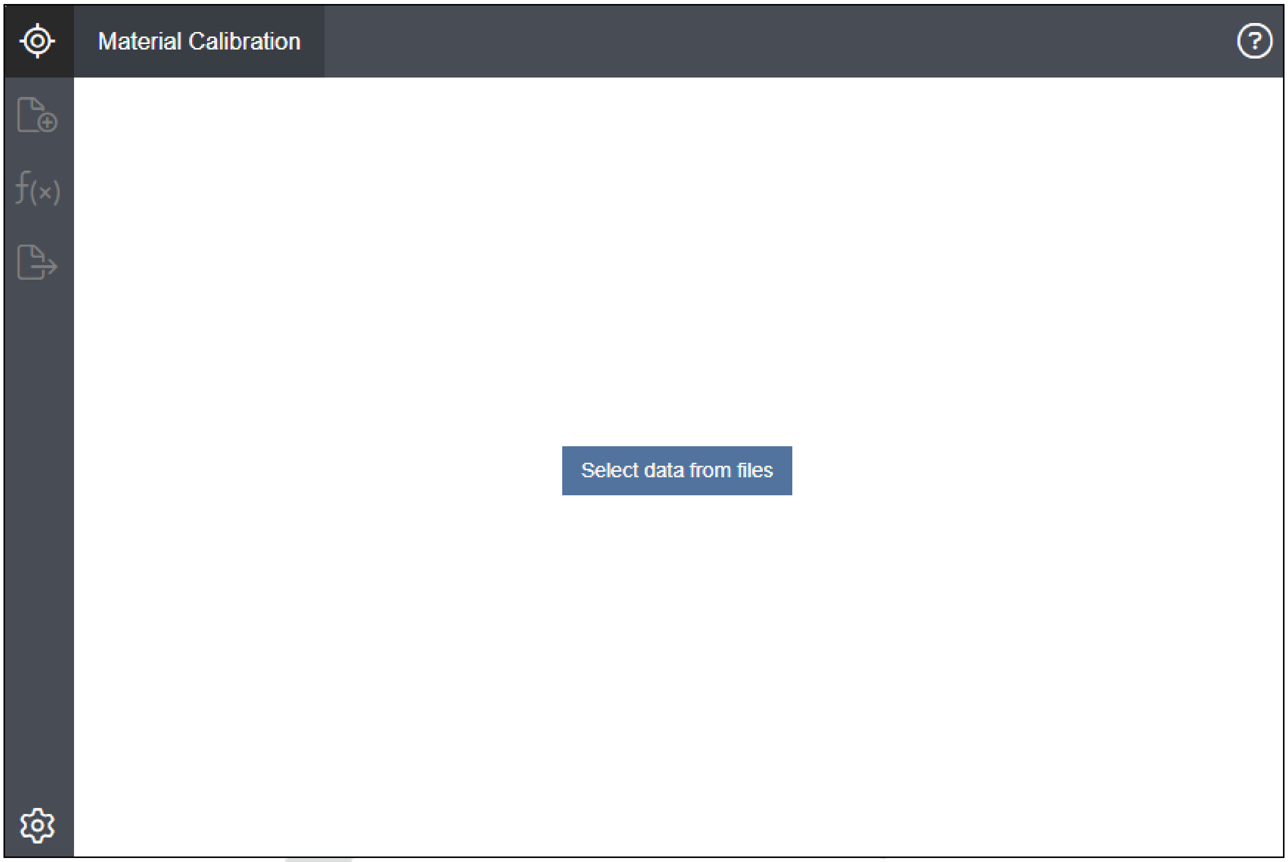
To calibrate a material from within Mechanical and export the results:

1. Click the **Material Calibration App** icon 

on the **Materials** context menu or right-click the **Materials** object and select **Material Calibration App** from the menu.

   The application launches in a separate browser window as shown below.

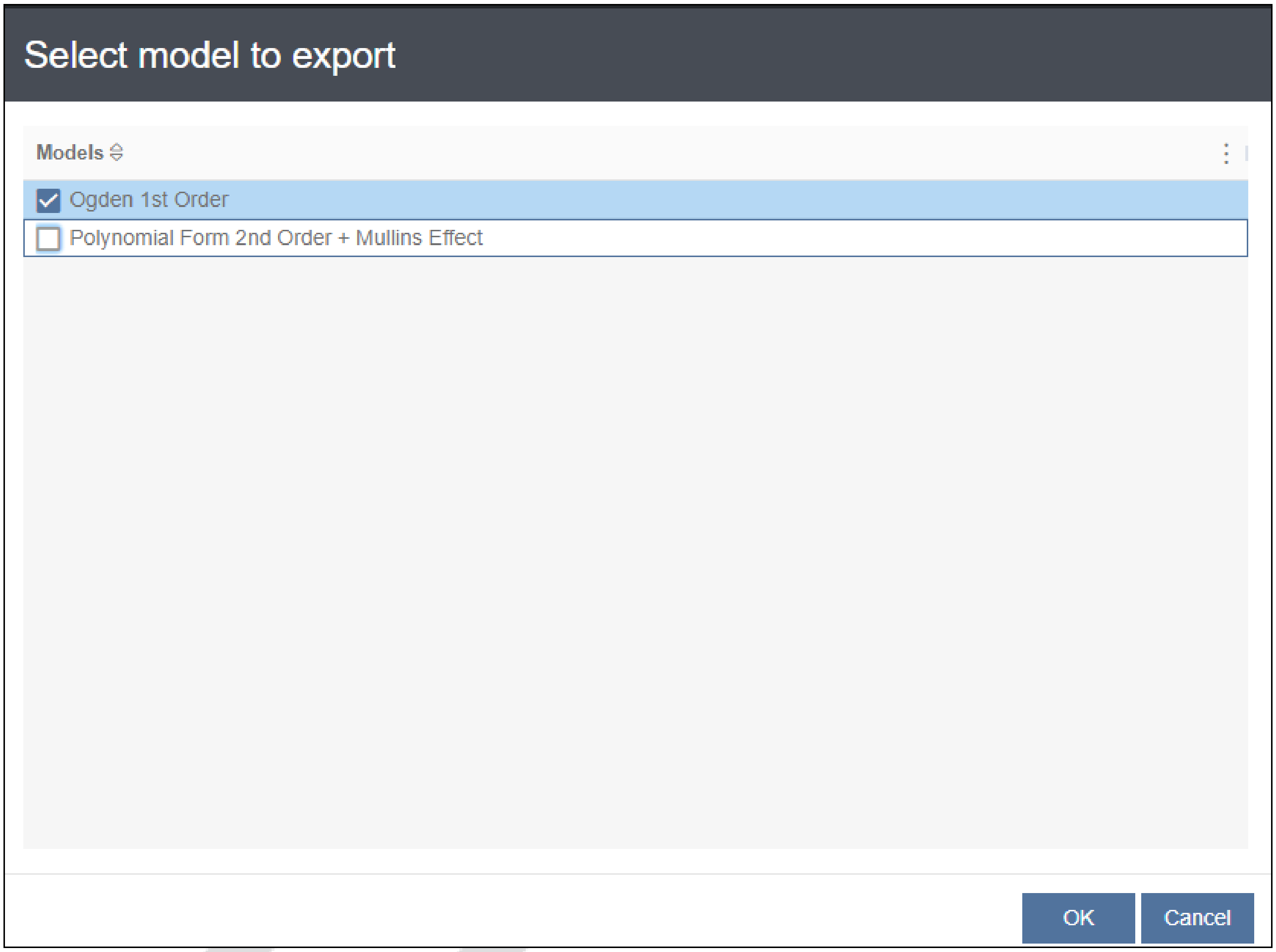
BETA



2. Follow the steps in the *ANSYS Material Calibration Guide (Standalone)* to load your test data into the Material Calibration application and calibrate the data against a material model.

3. When the calibration is finished, click the **Export** icon . The **Export settings** window displays as shown below. The names of the calibrated material models are shown under **Models**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Select the material model(s) to export and click **OK**.

> **Warning:**
>
> The Ansys Material Calibration App and Mechanical do not check whether exported calibration results contain mutually exclusive (or redundant) material models. Assigning a material with mutually exclusive material models can cause the solution to fail. This is especially a problem if you select multiple hyperelastic material models when you are exporting material calibration results.
>
> To avoid this problem, review the selected material models before you export them. Only export valid combinations of material models to the same material. See Valid Material Model Combinations for a list.

5. Choose how to export the results of your material calibration:

   - To save the calibration results as a new material, follow the steps in Save Exported Material Calibration Results as a New Material (p. 126).

   - To merge the calibration results with the properties of an existing material, follow the steps in Merge Material Calibration Results (p. 126).

- To replace the properties of an existing material with the calibration results, follow the steps in Overwrite Target Material with Material Calibration Results (p. 127).

After the export process finishes, you can assign the new or updated material to objects in your model.

Save Exported Material Calibration Results as a New Material

Use **Create New Material** to export material calibration results as a new material:

1. In the **Export settings** window, select **Create New Material** from the **Materials** drop-down menu.

2. Enter the name of the new material in the **Material Name** field.

3. Click **OK**.

The exported calibration results are saved as a new material. It is listed under the **Materials** object in the **Outline** pane of the model. You can view the properties of the new material in the Engineering Data Material View window.

Merge Material Calibration Results

Some materials exhibit different properties under different physical conditions. For example, the properties of a material can vary according to temperature. To model these materials, measure their properties under different physical conditions and use the Ansys Material Calibration App to compute material properties from the test results. Use **Merge** to combine the exported properties with those of the target material. This stores the material properties that you calibrated for the different physical conditions under test.

How Merge Affects the Properties of the Target Material

- You can merge multiple exported material models with the target material.

- If you select a material model that is not contained in the target material, the Material Calibration application appends the calibration results to the target material. (This applies regardless of temperature.)

- If you select a material model that is contained in the target material and was calibrated at the same temperature, the Material Calibration application overwrites the target material model's properties with the exported calibration results.

- If you select a material model that is contained in the target material but was calibrated at a different temperature, the Material Calibration application appends the exported calibration results to the target material.

Merging Exported Material Calibration Results

To merge material calibration results with the properties of an existing material:

1. In the **Export settings** window, select the name of the material from the **Materials** drop-down menu.

2. Select **Merge** from the **Import Options** drop-down menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The exported calibration results are merged with the properties of the selected material as described earlier in this section. You can view the merged material properties in the Engineering Data Material View window.

Overwrite Target Material with Material Calibration Results

Use **Overwrite** to completely replace the properties of an existing material with the exported material calibration results.

1. In the **Export settings** window, select the name of the target material from the **Materials** drop-down menu.

2. Select **Overwrite** from the **Import Options** drop-down menu.

All of the properties of the target material are replaced by the exported calibration results. You can view the updated properties of the material in the Engineering Data Material View window.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 28: TPA Workflow for FRF Calculator

Summary

A Transfer Path Analysis (TPA) workflow is available from the **FRF Calculator** in the **NVH Toolkit**, offering functions and properties as described below.

The following options are available under the FRF Calculator **Details**.

Start TPA Workflow

Binary property to start the TPA workflow.

Component Type

Binary property to select **Active**/**Passive**. Exposed if the **Start TPA Workflow** property is enabled.

Input Degrees of Freedom

Input the preferred DOFs as a comma-separated string of text (such as `fx,fy,fz,mx,my,mz`). This option is only available when using the **Start TPA Workflow** option with the **Component Type** set to **Active**.

Nodes Definition

If **Start TPA Workflow** is set to **Yes**, **Nodes Definition** will be set read-only to **Named Selections**. When the **Component Type** is set to **Active**, the **Input Nodes** and **Grouped Named Selection Connectivity Nodes** properties will be shown. When the **Component Type** is set to **Passive**, the **Output Nodes** and **Grouped Named Selection Connectivity Nodes** properties will be shown.

Grouped Named Selection Connectivity Nodes

Group of Nodal Named Selections that defines the Connectivity Nodes of the **FRF Worksheet**. Only one Named Selection is allowed.

If the **Start TPA Workflow** property is selected, once the **FRF Worksheet** table is filled, all the columns will be read-only except for the **Name** column. If you have selected an **Active** component, **Output Node** and **Output DOF** change to **Connectivity Node** and **Connectivity DOF** respectively. If you have selected a **Passive** component, **Input Node** and **Input DOF** change to **Connectivity Node** and **Connectivity DOF**.

Additionally, If you have selected the **Start TPA Workflow** property, you can only export displacement data from the **FRF Plotter** to an h5 file, which will be the input for the TPA Calculator.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 29: FRF Curve Fitting Analysis

You can use the NVH Add-on to perform **Curve-Fitting** analysis to extract natural frequencies and damping from UNV model FRF data.

The following options are available under FRF Calculator Details.

Curve-Fitting Options

Start Curve-Fitting Workflow

Binary property used to start the Curve-Fitting workflow. If enabled, it makes changes in the FRF Worksheet and the Stabilization Chart (p. 132) is displayed. Only available when the unv file has been loaded.

Range Minimum

The lower bound of the frequency range where Curve-Fitting is calculated.

Frequency Maximum

The upper bound of the frequency range where Curve-Fitting is calculated.

Maximum Modal Polynomial Order

The maximum order of the polynomial you wish to reach in the Curve-Fitting analysis.

If the **Start Curve-Fitting Workflow** property has been enabled, the FRF Frequency Table is hidden in the FRF Worksheet and will be shown, along with the associated buttons, in the Stabilization Chart (p. 132).

29.1. Curve-Fitting Methodology

Once the UNV data has been loaded in the FRF Calculator, you can perform **Curve-Fitting Analysis** based on a least-squares complex frequency estimation. Multiple input FRFs are summed and the obtained FRF is then used.

As a first step, the frequency range of interest is selected and input data is converted to the z-domain. To do this, a P Matrix is created and the measured FRF is scaled using this matrix, obtaining a T Matrix. The ratio of T/P is aligned with the FRF of the system which is approximated as A/B.

$$H(\omega) = \frac{A}{B} = \frac{T}{P}$$

The derivatives of the error with respect to each polynomial coefficient give the system of equations, which can be solved using a least-square method.

$P^T PA = P^T TB$

The normal equations are then combined and the contribution of the numerator coefficients (A) and denominator coefficients (B) are individually solved.

Poles can be obtained from the coefficients using the eigenvalues of the companion matrix. The eigenvalues are then converted from the z-domain to the problem domain. Stable poles are filtered based on whether the real value of the pole is negative.

Poles can be converted to natural frequencies by taking the absolute value, while the damping is the negative ratio of the real part of the poles to the natural frequency.

$\omega = abs(poles)$

$\varepsilon = -real(poles) \Big/ \omega_n$

The analysis is solved for a range of orders, and the stability of the frequencies and the damping are checked in the Stabilization Chart (p. 132). Once the order has been established, the synthetic FRF can be obtained as:

$$H_{jk}(\omega) = \sum_{r=1}^{N} \left(\frac{r_{jk}}{(\omega_r^2 - \omega^2) + 2i\varepsilon_r \omega_r \omega} \right) + \frac{LR}{\omega^2} + UR$$

where:

ω_r = the r natural frequency

ε_r = the r damping value

ω = the angular frequency

r_{jk} = the residue, which is the product of the mode shape and the mode participation factor. This first term is the contribution from the modes in the selected frequency range.

LR = (Lower Residual) a coefficient to account for correction of frequencies lower than those in our analysis

UR = (Upper Residual) a coefficient to account for correction of frequencies higher than those in our analysis

29.2. Stabilization Chart

The **Stabilization Chart** displays the information from the Curve-Fitting analysis. After generating the analysis, this will show the sum of the test FRFs selected from the FRF Worksheet, as well as the information regarding the stability of the poles. Once the modal order has been set and the **Extract Curve-Fitting parameters data** option selected, this information will be provided in the second **Frequency Table**, and the synthesized FRF will be shown in the plot. Selecting the modes will update this synthesized FRF dynamically, based on the modes selected.

The **Stabilization Chart** is divided into three parts:

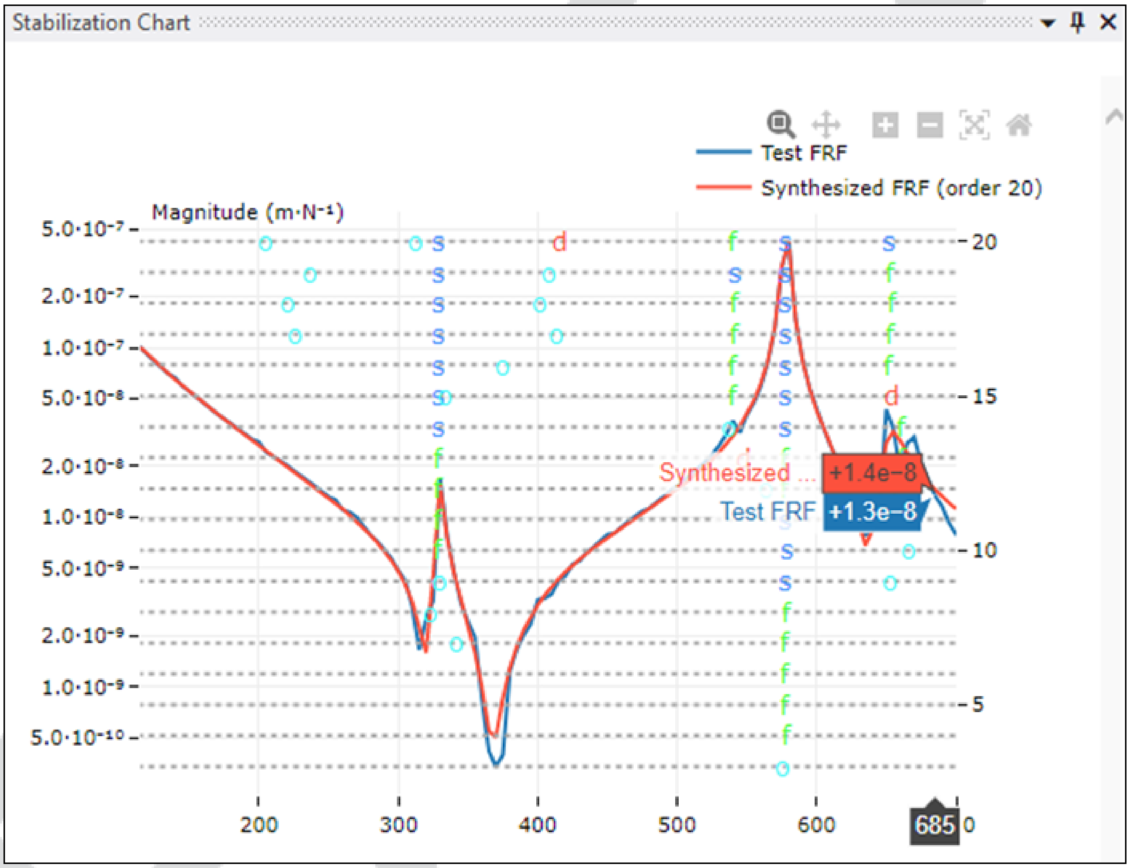
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stabilization Plot

A dynamic plot that allows hovering, zooming and panning operations. This plot is dynamically updated using the **Stabilization Chart Options** and the second **Frequency Table**, which contains Curve-Fitting stable modes. The maximum and minimum frequency values of the x-axis are restricted to those defined in the Curve-Fitting section of the FRF Calculator Details.

The entire data range can be plotted by selecting the **Autoscale** option. Clicking the **Home** icon (reset axes) replots the data in the user-defined frequency range.

If the **Extract Curve-Fitting parameters data** option has not been selected, the plot will only show the test FRF and the stability of the poles. Otherwise, it will also show the FRF estimation of the **Modal Order** selected in the **Stabilization Chart Options**, with the modes selected in the second **Frequency Table**.



Stabilization Chart Options

The modal orders are increased from a low to a high value and the stability of generated modes can be studied visually. Depending on the threshold set, the modes identified at each order are tagged with one of the following:

- **o** : Newly detected mode
- **f** : Only frequency stable
- **d** : Only damping stable

- **s** : Both frequency and damping stable

The following properties are available:

Frequency Tolerance (%)

Allows you to change the frequency tolerance to consider a pole to be stable based on its frequency, compared to the previous iteration. This updates the plot dynamically.

Damping Tolerance (%)

Allows you to change the damping tolerance to consider a pole to be stable based on its damping value, compared to the previous iteration. This updates the plot dynamically.

Modal Order

Allows you to change the modal order for extracting the damping values and for estimating the FRF for the analysis.

Stabilization Chart Options	
Frequency Tolerance (%)	1
Damping Tolerance (%)	10
Modal Order	20

Frequency Tables

The first table has the same behavior as the **FRF Frequency Table** in the **FRF Worksheet**. The second table contains the mode information for the selected **Modal Order** in the **Stabilization Chart Options**. It only shows stable modes, both in frequency and in damping. This information is updated when you click the **Extract Curve-Fitting parameters data** button.

Select or clear modes to update the **Synthesized FRF** in the **Stabilization Chart** plot and to understand the contribution of each mode.

Use in FRF	Damped Freq. (Hz)	Damp. Ratio (%)
☑	64.7567	2.0000
☑	65.2566	2.0000
☑	66.3798	2.0000
☑	68.6985	2.0000
☑	89.7723	2.0000
☑	91.3932	2.0000
☑	121.3255	2.0000
☑	121.3274	2.0000
☑	132.1981	2.0000
☑	132.4670	2.0000
☑	134.1542	2.0000
☑	134.1545	2.0000
☑	134.5915	2.0000

Use in Cur...	Damped Freq. (Hz)	Damp. Ratio (%)
☑	329.4710	0.6575
☑	578.1518	0.2923
☑	651.5498	1.2688

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 30: Welding Toolbox Extension

The Welding Toolbox is a native Mechanical extension that is designed to help manage welding process simulation. This tool eases setup in Mechanical and helps to optimize the weld sequence and process parameters in order to:

- Assess temperature distribution.
- Predict distortion and residual stresses.

Note:

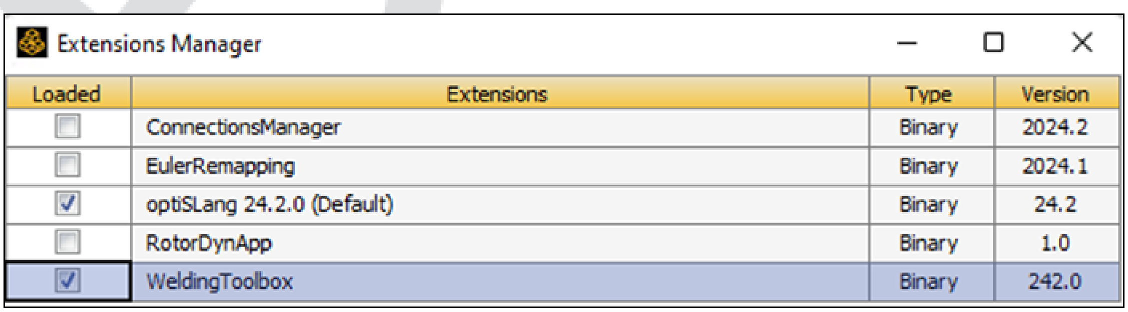
As needed, see the Activate Beta Options (p. 1) section to make the feature available.

The following sections describe how to use the Welding Toolbox:

30.1. Enabling the Welding Toolbox Extension

To enable the **Welding Toolbox** extension:

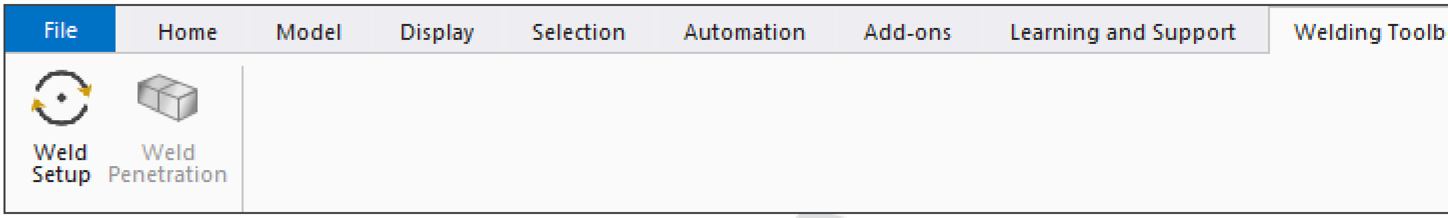
1. Follow the directions in Activate Beta Options (p. 1) to make the feature available.
2. In the Workbench toolbar, under the **Extensions** options, select **Manage Extensions...**.
3. Select the **Welding Toolbox** check box to enable the extension.


Extensions Manager

Loaded	Extensions	Type	Version
☐	ConnectionsManager	Binary	2024.2
☐	EulerRemapping	Binary	2024.1
☑	optiSLang 24.2.0 (Default)	Binary	24.2
☐	RotorDynApp	Binary	1.0
☑	WeldingToolbox	Binary	242.0

4. Once loaded, the **Welding Toolbox** extension will appear in the Mechanical ribbon.

BETA



30.2. Using the Welding Toolbox Extension

The **Welding Toolbox** supports three workflows to simulate weld object behavior:

- **Transient Thermal** analysis.
- **Static Structural** analysis.
- **Transient Thermal** linked to **Static Structural** analysis.

Transient Thermal Analysis

In this case, only temperature distribution is calculated in the analysis. The temperature results will depend on the welding inputs (Initial **Temperature**/**Power**, **Velocity**) and the proper thermal boundary conditions assigned in the model.

Static Structural Analysis

This type of analysis works under the assumption that the welding bodies have volumetric shrinkage between an initial theoretical temperature and room temperature. This shrinkage can be calibrated to account for the deformation of the connected bodies. A temperature distribution is not calculated in this approach.

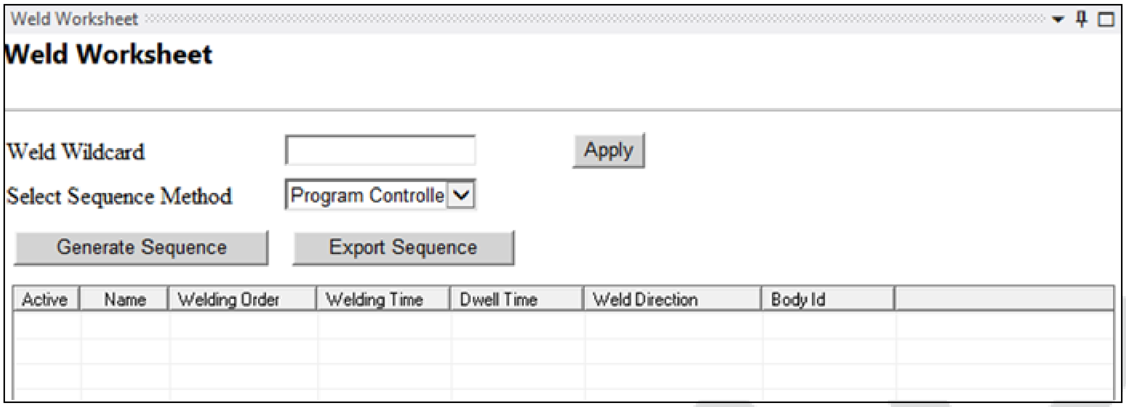
Transient Thermal Linked to Static Structural Analysis

This combines the previous two cases. First, the correct temperature distribution over time is calculated based on welding inputs and thermal boundary conditions. Then, these temperatures are imported into the structural analysis, to be used together with the mechanical setup (material properties, structural boundary conditions) to account for the weld deformation and stress state during the welding process.

The Mechanical UI process for these three types of analysis is described in the sections below. A new **Weld Setup** object has been developed, available in Mechanical once the extension is loaded.

A **Weld Setup** object can be inserted under a **Transient Thermal** or **Static Structural** analysis. When this weld object is inserted, the **Weld Worksheet** opens in the Mechanical editor.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



30.3. Weld Worksheet Operations

The **Weld Worksheet** contains the following parameters and controls:

Weld Wildcard

This field is used to import the weld bodies present in your assembly by name. The wildcard string is compared against the names of the bodies in the assembly and matches are imported to the worksheet. The wildcard string can be a substring of the names of the weld bodies. For example, using **`WELD`** as wildcard string will match the bodies named **`WELD_1`** and **`WELD_2`**. Clicking the **Apply** button populates the worksheet with the matching bodies.

Select Sequence Method

Three options are available to select the sequence in which the operations on the chosen weld bodies are to be performed:

- **Program Controlled**: the sequence is the order in which the bodies appear in the tree under the **Geometry** object.
- **By Location**: the sequence is determined relative to a coordinate system. When selecting this option, two additional fields are exposed to allow you to select a coordinate system and to select the type of arrangement with respect to the chosen coordinate system. The options for the latter are **Absolute**, in which case the sequence is by absolute distance of the weld bodies from the given coordinate system, and **X Dir**, **Y Dir**, or **Z Dir** , in which case weld bodies are arranged in order of distance in the chosen coordinate system direction, from lowest to highest.
- **CSV**: the sequence and all other values for the worksheet can be directly imported from a `.csv` file. The weld names in the `.csv` file are matched against the welds in the tree and matches are populated in the worksheet.

Generate Sequence

Once the sequence method is chosen, clicking the **Generate Sequence** button populates the **Welding Order** fields for the respective welds.

Export Sequence

Clicking this button exports the worksheet in `.csv` format.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Each worksheet row has two context menu options:

Flip Weld

This can be used to flip the direction of welding for the chosen bodies.

Zoom To selection

This can be used to zoom to the weld bodies that were chosen in the worksheet.

30.4. Weld Setup Properties

A **Weld Setup** object has the following configurable parameters:

Selection Method

Two options are available for selecting weld bodies. **Manual Input** allows you to select the weld bodies interactively or by named selections, while **Weld Worksheet** allows you to select the weld bodies using the wildcard option in the worksheet.

Weld Heat Input

Three options are available to supply the load to the weld object:

- **Temperature**: this option allows you to assign a direct temperature to the group of weld elements active in a specific load step.
- **Power - Direct**: direct heat per unit volume is applied to the weld object. This option exposes the **Direct Heat Generation** parameter.
- **Power - Indirect**: net energy is calculated and applied to the weld object. This option exposes the following parameters

  Machine Parameter

  Multiplication parameter specific to the welding tool to calibrate machine heat deposition rate.

  Voltage

  Electrical voltage of the welding tool.

  Current

  Electrical current drawn by the welding tool.

  Power is calculated as follows: *Machine Parameter * Voltage * Current / Number of groups*.

Thermal Strain Scaling Factor

Factor used in calculating the temperature at which the thermal strain begins acting on the model. The melting temperature is multiplied by this factor to set the body reference temperature. This is available only when the **Weld Setup** object is in a **Static Structural** environment.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
BETA

Ambient Temperature

This is used to specify the environment temperature of the weld process.

Weld Velocity

The velocity of the welding tool. This determines the **Welding Time** in the worksheet and is used to automatically set the correct load step timing table.

Weld Mode

Two options are available:

- **Material Deposition**: Element kill and alive technology is used to create the welding elements in order to simulate weld feeding material deposition.
- **Direct Energy Deposition**: Only thermal input is applied, but there is no activation or deactivation of elements, to simulate welding process without filler material.

Elements Groups per Weld

This parameter determines the clustering or grouping of elements within a weld. The value is used to determine the number of fractions into which the weld volume would be split during the analysis. The number of load steps in the analysis settings is equal to the total number of elements groups plus additional cooling time for each welding line.

Note:

Weld Direction calculation is done using the longest edge of the weld body. Welds should be modeled as long continuous bodies where possible. The accuracy of **Weld Time** is dependent on effective defeaturing in the model.

30.5. Understanding Behavior in Terms of Solver Inputs

This section describes the effect of some of the above parameters on the commands that are input to the solver.

Weld Heat Input

The following behavior only applies to **Transient Thermal** analysis:

- **Temperature**: For this option, `D,ALL,TEMP,%Temperature value%` is supplied to the weld selection.
- **Power - Direct**: For this option, `BFE,ALL,HGEN,,%DirectHeatGenerationValue%` is supplied to the weld selection.
- **Power - Indirect**: For this option, the total power is calculated as the product of **Machine Constant**, **Voltage** and **Current**. This is then divided by the volume of the weld selection and this value is applied as `BFE,ALL,HGEN,,%CalculatedValue%`.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ambient Temperature

The specified value is used in a **Static Structural** analysis to define the ambient temperature. `BF,ALL,TEMP,%AmbientTemperature%` is used in the solver.

Weld Mode

Two mode options are available:

- **Material Deposition**: adds `ekill` and `ealive` commands to the solver and is executed according to the weld selections.
- **Direct Energy Deposition**: applies load without `ekill` and `ealive` commands to the solver and is executed according to the weld selections.

Thermal Strain Scaling Factor

This factor can be used to account for unknown effects in your simulation that may affect the final welded bodies' distortion or stress state, for example, thermal boundary conditions (which directly affect temperature distribution) or temperature dependent material properties. It can also be used to account for the simplifications made when using a structural standalone analysis.

The thermal deformation of a structural body is calculated as follows:

$$\varepsilon_{th} = \alpha \cdot \left(T - T_{ref} \right)$$

where ε_{th} is the thermal deformation and α is the scaling factor.

By modifying the **Reference Temperature** T_{ref} of a specific body, you can select at which temperature the bodies are in a non-deformed state in the simulation.

For example, if T_{ref} is defined as the initial temperature when the bodies are activated, the initial thermal strain of those bodies would be equal to zero. This will therefore achieve the maximum volumetric shrinkage when the bodies are cooled down to room temperature.

$$T_{ref} = T_{(i=0)}$$

$$\varepsilon_{th(i=0)} = \alpha \cdot \left(T_{(i=0)} - T_{ref} \right) = 0$$

$$\varepsilon_{th(i=end)} = \alpha \cdot \left(T_{(i=end)} - T_{(i=0)} \right) = -value$$

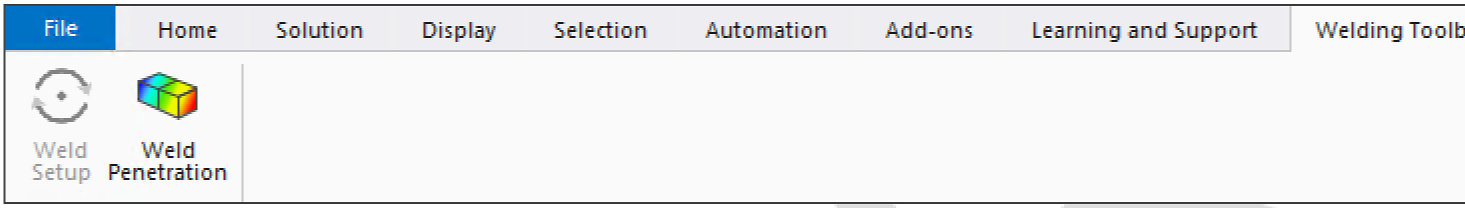
30.6. Weld Penetration Result Object

The **Weld Penetration** result object is used to visualize the effect of welding process on the temperature profile around the weld regions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
BETA

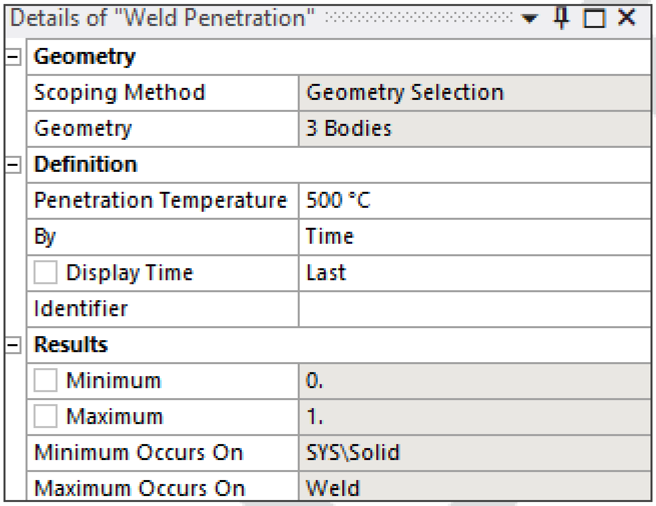
30.6.1. Adding a Weld Penetration Result Object

To insert a **Weld Penetration** result object, click the **Weld Penetration** icon under the Welding Toolbox Extension. This option is available only for transient thermal analyses.



30.6.2. Setting up a Weld Penetration Result Object

Specify the following information for the **Weld Penetration** result object.



Category	Property Name and Description
Geometry	**Scoping Method** Select the bodies where you want to visualize the depth of the welding process. Valid scoping methods are: **Geometry Selection** The object is applied to a body or bodies that you choose using graphical selection tools. Select the desired geometry, then click **Apply** in the **Geometry** field below the Scoping Method. This field then displays the type of geometry (for example: Body) and the number of geometric entities (for example: 1 Body) to which the probe object is scoped. **Named Selection** Scopes the object to a geometry-based named selection.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Property Name and Description
	Material ID Scopes the object to a specific material.
Definition	**Penetration Temperature** The filtering temperature for assessing the depth for visuaulizing the effect of the welding process. **By** Select the input for your results display. • **Time** • **Result Set** • **Maximum Over Time** • **Time of Maximum** • **Minumum Over Time** • **Time of Minimum** **Display Time** If you are displaying results by time, enter the display time for your results. **Penetration Temperature by Time** indicates how much of the model has reached the specified **Penetration Temperature** value by the given **Display Time**. **Last** is the most recent display time for which there is a solution. **Result Set** If you are displaying results by result set, enter the desired result set number. **Penetration Temperature by Result Set** indicates how much of the model has reached the specified **Penetration Temperature** value by the given **Result Set**. **Last** is the most recent result set number for which there is a solution.
Results	This category displays the following read-only properties for the Weld Penetration object: **Minimum** The minimum plotted value of the penetration temperature, represented by 0.

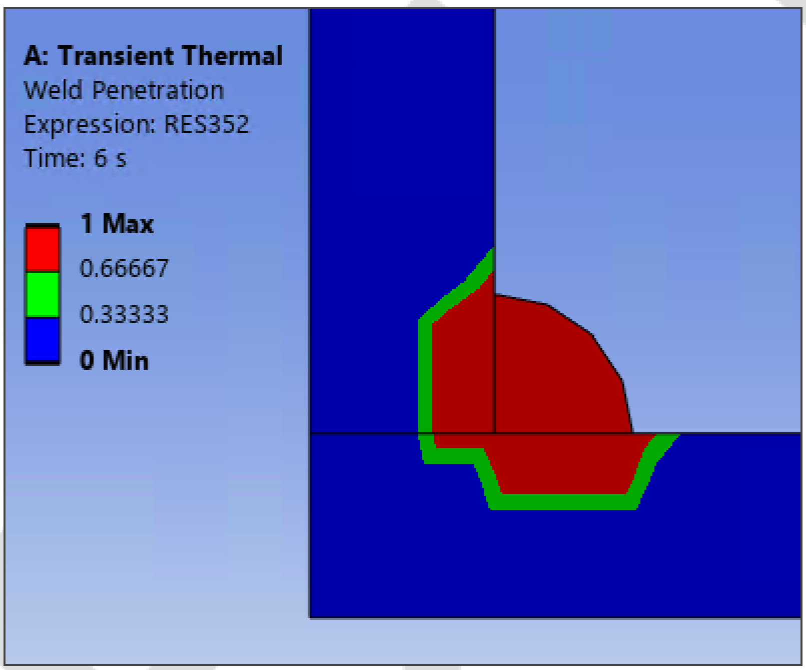
BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Property Name and Description
	Maximum The maximum plotted value of the penetration temperature, represented by 1. **Minimum Occurs On** The location of the minimum temperature value. **Maximum Occurs On** The location of the maximum temperature value.

30.6.3. Viewing the Weld Penetration Plot

The weld penetration plot shows the depth to which the effect of the welding process is seen, as filtered by the **Penetration Temperature**. The plot's contours are defined by the **Status** field. A value of 1 indicates the elements that reached a higher temperature than the **Renetration Temperature** (shown in red on the plot).



BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 31: Data Translator Application

The **Data Translator** is a beta application that converts results files between different formats. It contains two translation applications:

- A **Results Translator** that converts Mechanical application results into H5 files. An H5 (or HDF5) file is saved in the Hierarchical Data Format (HDF), which is designed to store and organize large amounts of information. Use the **Results Translator** to filter and export the results of your simulation into applications that use the H5 file format.

  The **Results Translator** is available through the independent **Data Translator** application and as a native extension in the Mechanical application.

- A **Solver Deck Translator** that converts input files from Abaqus into a file format that is readable by the Mechanical application. Use it to import Abaqus models into the Ansys Mechanical application.

  This application is only available through the **Data Translator**, which is opened independently of the Mechanical application.

The Data Translator is only supported for the Windows platform.

Note:

As needed, see the Activate Beta Options (p. 1) section to make the feature available.

For more information about the Data Translator, see the following sections:

31.1. Accessing the Data Translator Application

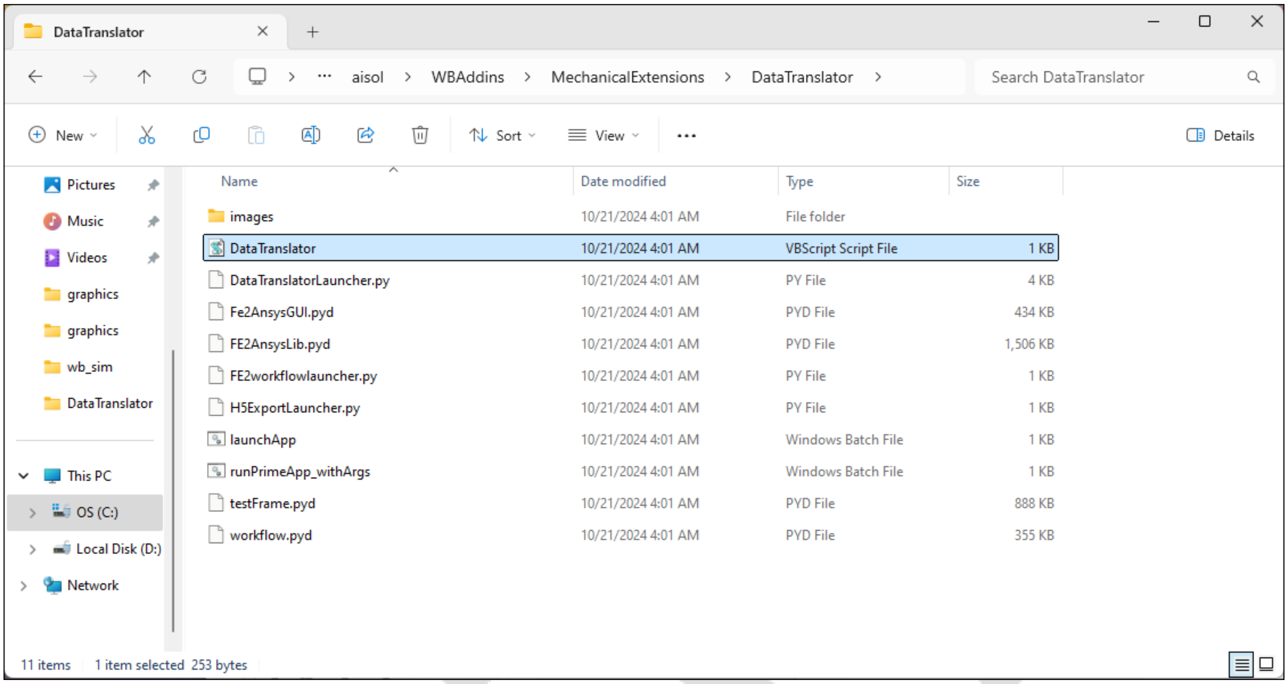
The Data Translator application is opened independently of the Mechanical application.

To access the Data Translator app, do the following:

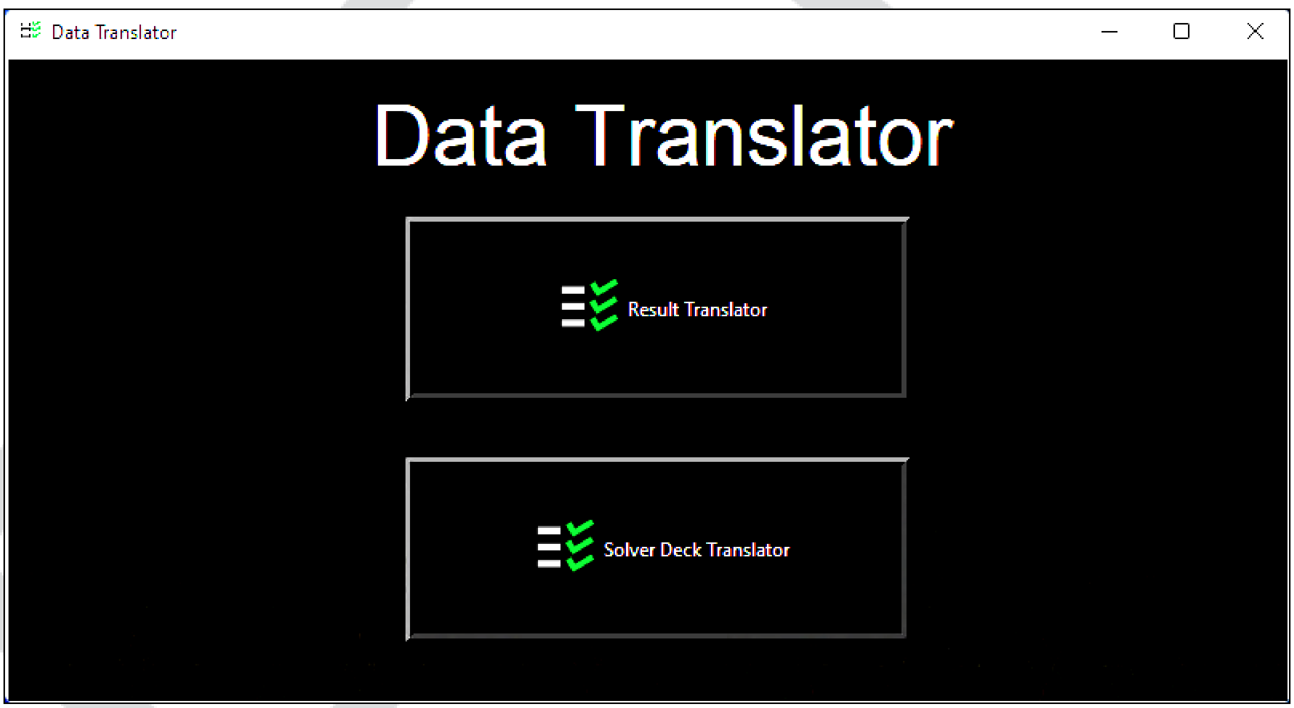
1. Navigate to the following directory:

   `\ANSYS Inc\v251\aisol\WBAddins\MechanicalExtensions\DataTranslator`

   Alternatively, use the Windows search bar to find the file `DataTranslator.vbs`.



2. Double click the file `DataTranslator.vbs`. The following dialog box is displayed.



3. Do one of the following to launch the **Data Translator** applications:

- Click the **Results Translator** button to run that application. Alternatively, you can access it through the Mechanical application. For more information, see Using the Results Translator (p. 147).

- Click the **Solver Deck Translator** button to run that application. For more information, see Using the Solver Deck Translator (p. 157).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2. Using the Results Translator

Use the **Results Translator** to save the results of your simulation to an H5 Hierarchical Data Format (HDF) file. The H5 (or HDF5) file format stores and organizes large amounts of information. This allows you to export the results of your simulation to applications that use the H5 file format.

As part of this translation process, you can do the following:

- Select the compression algorithm for the exported file.
- Specify which parts of your analysis to export by scoping the export to specific results and using named selections.
- Filter results to transform and reduce the amount of data stored in the exported file.

The **Results Translator** is available as both a standlone application and an extension in the Mechanical application.

See the following sections for more information about this application:

31.2.1. Workflow for Using the Results Translator

To translate solution files with the **Results Translator**, follow this general workflow:

1. Generate a solution for your model and review the results.
2. Access (p. 147) the **Results Translator**.
3. Specify the options (p. 150) for exporting results, including the compression algorithm, scoping, and filtering options.
4. Export (p. 156) the selected results to an H5 file.

31.2.2. Accessing the Results Translator

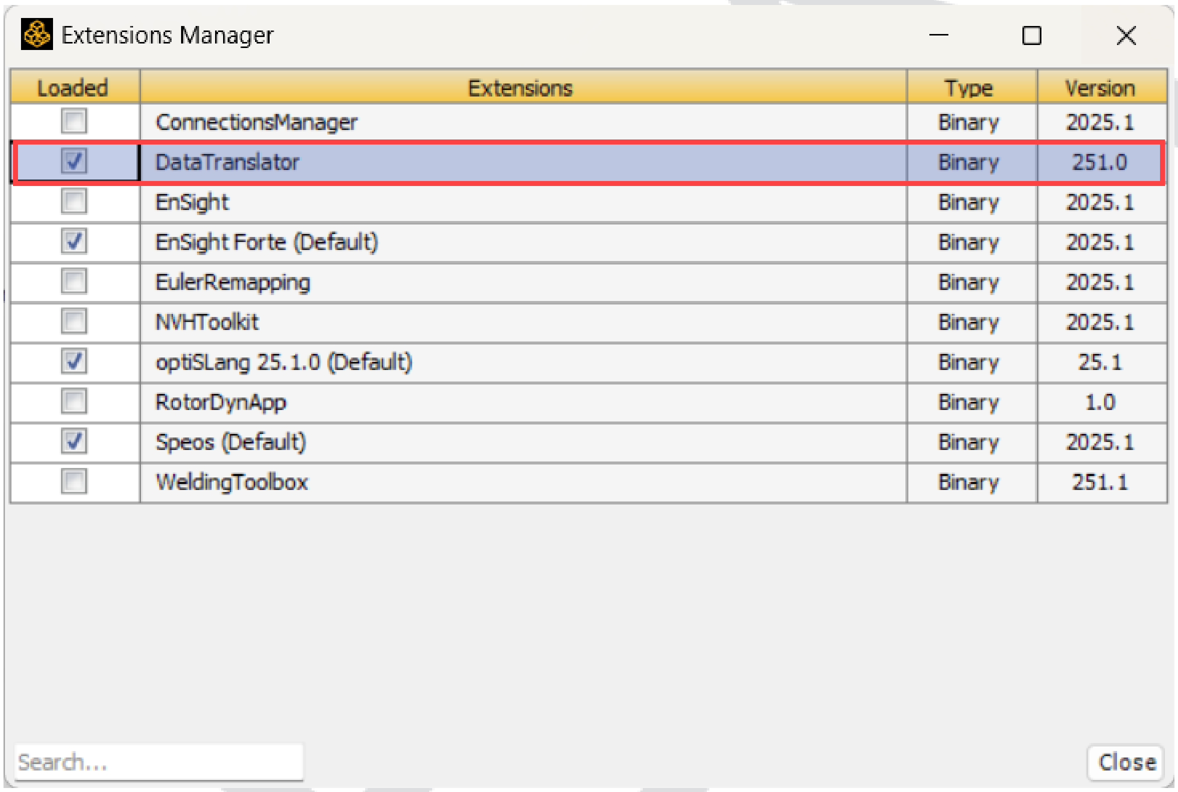
You can access the **Results Translator** in two ways:

- By running the independent **Data Translator** application as described in Accessing the Data Translator Application (p. 145).
- By loading it as an extension to the Mechanical application as described in Loading the Results Translator Extension in Workbench (p. 147).

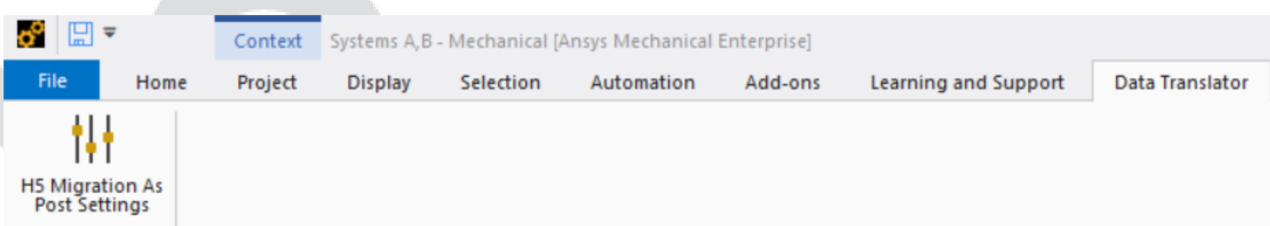
Loading the Results Translator Extension in Workbench

Follow these steps to load the **Results Translator** extension in Workbench:

1. Run the Workbench application.
2. Follow the directions in Activate Beta Options (p. 1) to make this feature available.
3. Open your project.
4. Click **Extensions** > **Manage Extensions**in the Workbench Project page. The Extensions pane is displayed as shown below.



5. Click the check box next to **Data Translator** (highlighted in red above).
6. Click the **Close** button to save your changes.
7. Open your Workbench project in the Mechanical application. The **Data Translator** extension appears in the ribbon as shown below.

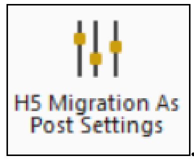


Run the Results Translator in the Mechanical Application

To access the Results Translator in the Mechanical application, do the following:

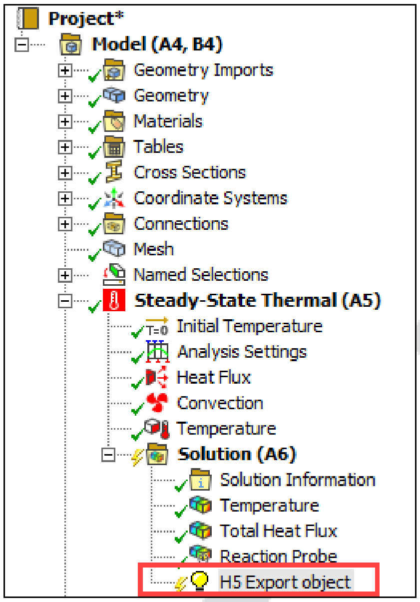
1. Add an **H5 Export Object** to the **Solution** branch of your project. Do one of the following:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the **H5 Migration As Post Settings** icon 
.

- Right-click the **Solution** object to open the context menu, then choose **Insert** > **H5 Export Object**.

The Results Translator adds an **H5 Export object** under the **Solution** branch of the project tree.



2. Right-click the **H5 Export object** and select **Generate** from the menu. The **Results Translator** dialog is then displayed.

H5 Export Object Status

When you create an **H5 Export object**, the application inserts it into the **Solution** tree.

The following icons and fields show the status of the **H5 Export object**.

- A check mark (✔) indicates that all required information has been entered for the **H5 Export object** object and an H5 file was generated.
- A lightning bolt indicates that a valid H5 file has not been generated. One or more fields could be empty or contain invalid information.
- The **H5 Export Status** field in the **Details** pane gives more information about the status of the **H5 Export object**.

  Export awaited

  An H5 file has not been generated.

  Exported H5

  An H5 file was successfully generated.

BETA

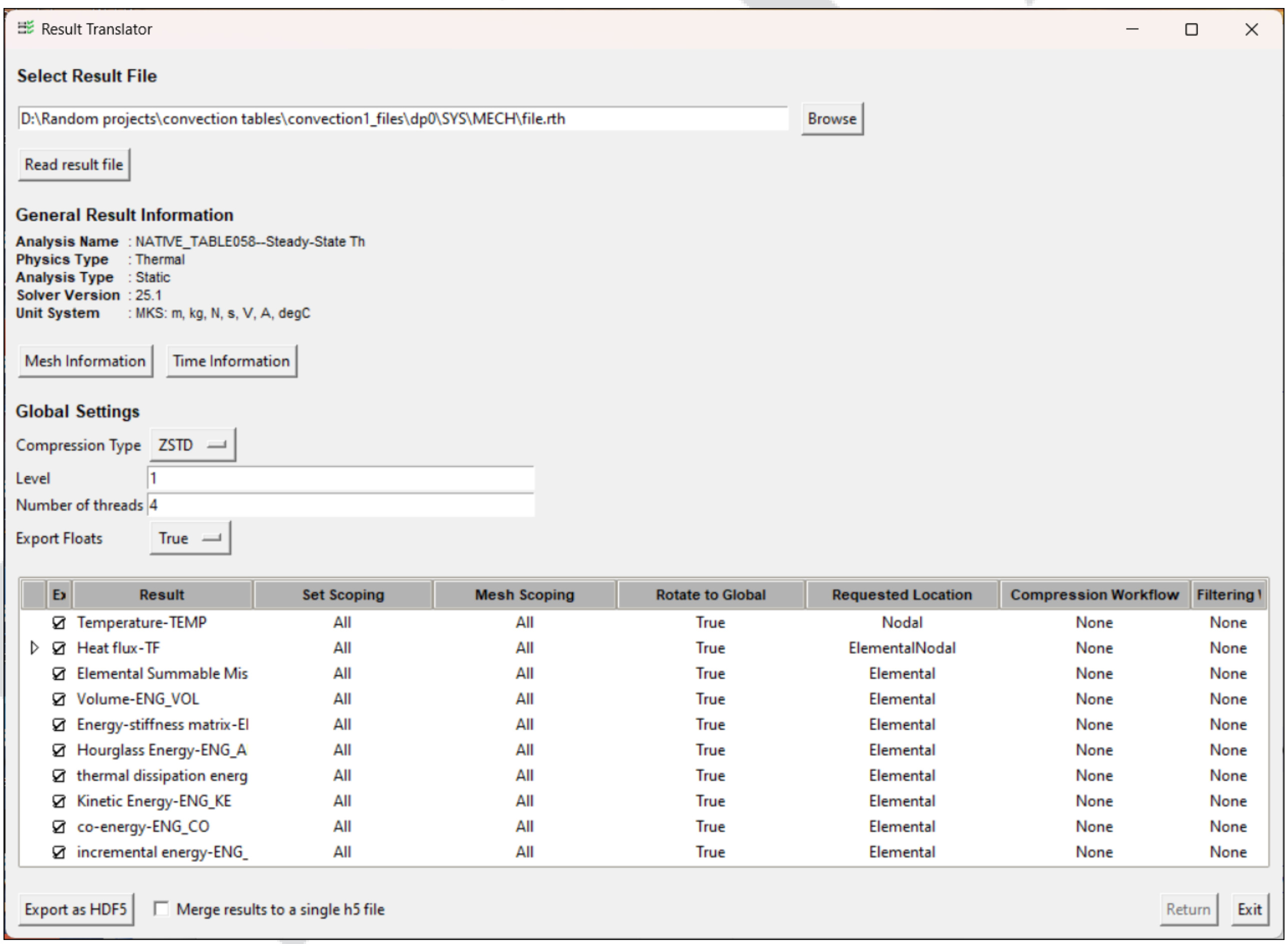
See Exporting Solution Files with the Results Translator (p. 150).

31.2.3. Exporting Solution Files with the Results Translator

When you run the Results Translator, the following dialog appears. If you accessed this extension from within the Mechanical application, the fields are pre-filled with information from the current project.

Follow the steps below to select and export results as an H5 file.

1. Select the results file (p. 150) to be exported. The **Result Translator** dialog shows information (p. 151) about the selected file.
2. Select the compression type. (p. 152).
3. Select the specific solution results (p. 153) to be exported.
4. Optionally, edit the export settings (p. 154).
5. Export the results (p. 156) to an H5 file.



Select the Result File

To select the result file to convert:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enter the file name and directory path in the **Select Result File** field. Alternatively, click the **Browse** button to navigate to the desired file. The supported file formats are `*.rst`, `*.d3plot`, `*.rth`, and `*.h5`.

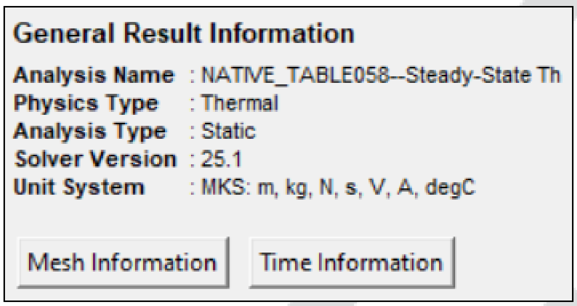
2. Click **Read Results File** to load the selected file.

Note:

If the name of the result file ends with `_0` (or another number), the Results Translator assumes that it contains results from a distributed solution. The extension then searches for other files with the same name and attempts to load them. If the Results Translator successfully loads the distributed result files, it processes the files in parallel.

View Information About the Results

The following fields display information about the selected results file.



General Result Information

Analysis Name

The name of your Ansys project.

Physics Type

The solver that generated the solution.

Analysis Type

The type of simulation that was performed.

Solver Version

The release version of the solver.

Unit System

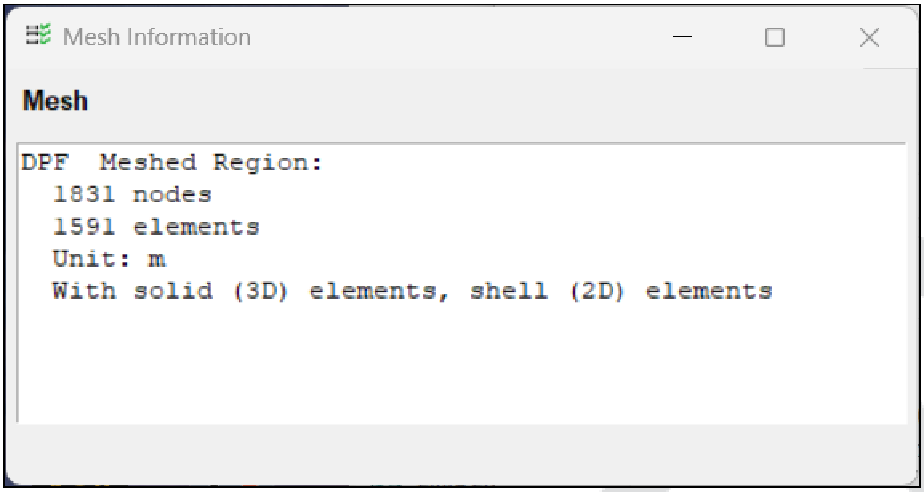
The system of measurement for the project.

Mesh Information

Click **Mesh Information** to view information about the model's finite element mesh, including the number and types of nodes and elements.

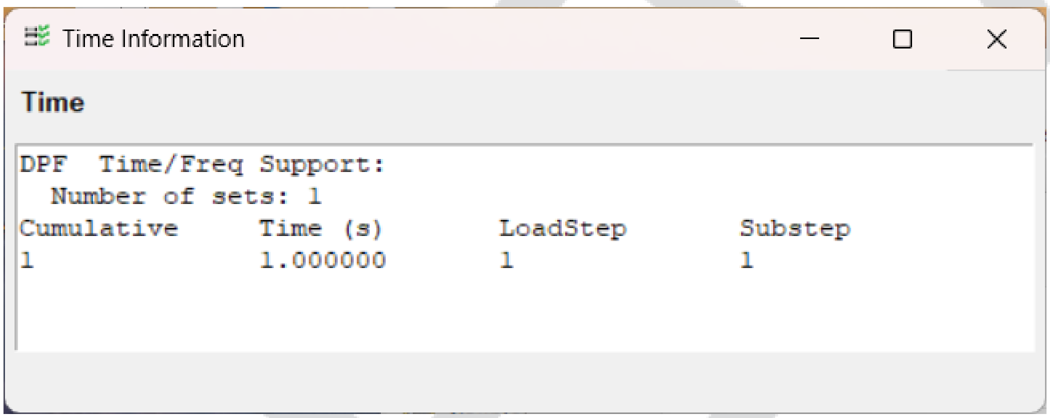
BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



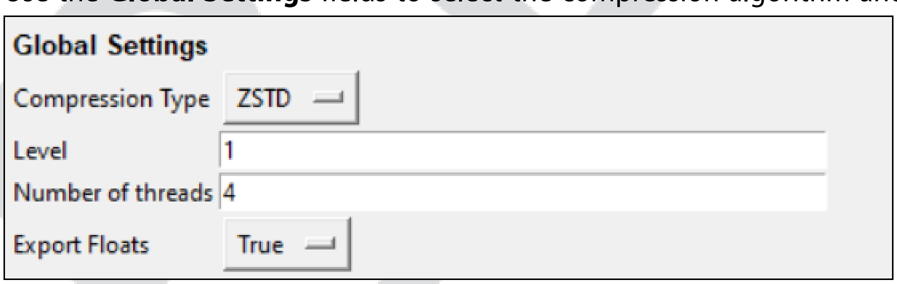
Time Information

Click **Time Information** to view the time and frequency steps in the result file.



Select Compression Type

Use the **Global Settings** fields to select the compression algorithm and specify its settings.



Compression Type

Choose the compression algorithm that is used to export the result file. Numeric values with integer, float, and double data types are compressed.

- **ZSTD** (default) uses the Zstandard algorithm for fast lossless data compression
- **GZIP** uses the DEFLATE algorithm for data compression

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **None** performs no data compression

Level

(*ZTSD and GZIP only.*) Determines the level of compression. Low values typically result in lower compression rates with high compression and decompression speed. Increasing the compression level increases the compression rate at the expense of performance.

The available compression levels for both ZSTD and GZIP are between 1 and 20. The default compression level is 1. Values higher than 6 for ZSTD and 3 for GZIP are rarely useful. The improvement to the compression rate is low compared to the additional time it takes to compress the data.

Number of threads

(*ZTSD only.*) You can optionally increase the number of threads used during the compression step. The default is 4 threads.

Export Floats

Specifies whether all available data is exported as floating point numbers. The default (**True**) is that degree of freedom (DOF) results are exported as double precision numbers while elemental results are exported as floating point numbers.

Select Results for Export

The **Result** table displays the available results from the selected result file. The rows list the types of results stored in the file and the columns list the default export settings for individual results.

1. To select a row of results for export, click the check box under **Export** in the Result table.
2. Optionally, you can modify the export settings for a result by double-clicking anywhere in that row of the table. See Editing Results Export Settings (p. 154) for more information.

	Export	Result	Set Scoping	Mesh Scoping	Rotate to Global	Requested Location	Compression Workflow	Filtering Workflow
	☑	Temperature-TEMP	All	All	True	Nodal	None	None
▷	☑	Heat flux-TF	All	All	True	ElementalNodal	None	None
	☑	Elemental Summable Mis	All	All	True	Elemental	None	None
	☑	Volume-ENG_VOL	All	All	True	Elemental	None	None
	☑	Energy-stiffness matrix-EN	All	All	True	Elemental	None	None
	☑	Hourglass Energy-ENG_AH	All	All	True	Elemental	None	None
	☑	thermal dissipation energy	All	All	True	Elemental	None	None
	☑	Kinetic Energy-ENG_KE	All	All	True	Elemental	None	None
	☑	co-energy-ENG_CO	All	All	True	Elemental	None	None
	☑	incremental energy-ENG_I	All	All	True	Elemental	None	None

Export

Selects individual results for export.

Result

The outputs from the simulation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set Scoping

Controls selected sets for a particular result, enabling users to only export a subset of the times/frequencies stored in the original result file.

Mesh Scoping

Selects the parts of the model where a particular result is exported. In the Mechanical application, this is done by using named selection.

Rotate to Global

Whether the results are rotate to the global coordinate system.

Requested Location

Specifies how the exported results are stored.

Compression Workflow

Specifies the compression workflow algorithm. See Compression Workflows (p. 156).

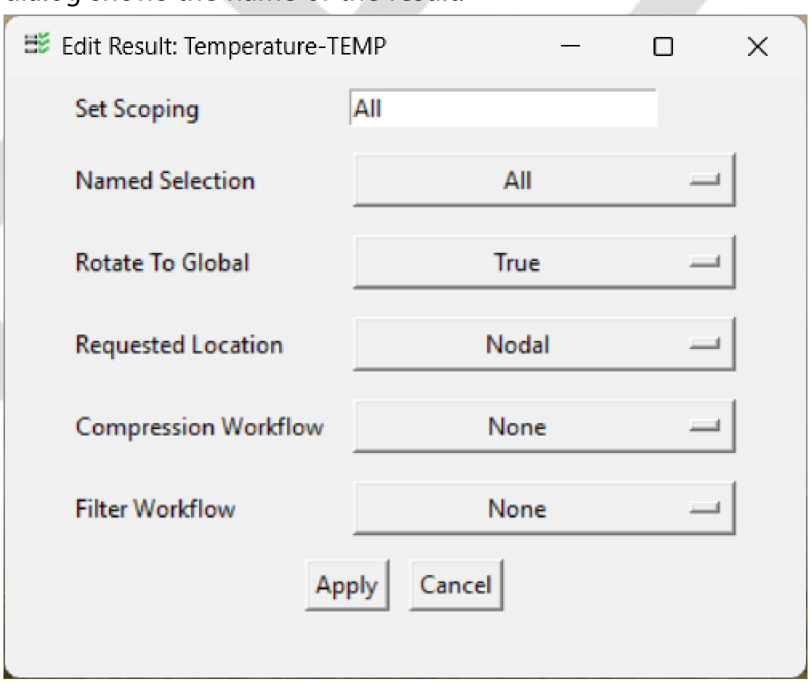
Filtering Workflow

Specifies the filtering workflow algorithm. See Filtering Workflows (p. 157).

Editing Results Export Settings

You can optionally edit the values of the export settings for individual rows in the **Results** table:

1. Double-click the row of the table that contains the desired solution results. The **Edit Results** dialog shows the name of the result.



2. Modify the desired export settings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Apply** to save your changes.

You can edit the following settings:

Set Scoping

Select the solution time steps and frequencies to export. The default is **All**.

Named Selection

Select the desired named selection from the drop-down menu. The default is **All**.

Rotate to Global

If set to **True** (*default*), this option rotates the results to the global coordinate system.

Note:

If **Rotate to Global** is set to **False**, the coordinate system rotation cannot be changed in the exported file. This setting cannot be modified upon reading the exported file using Data Processing Framework (DPF) operators.

Requested Location

The location of the solution. Select one of the following:

Nodal (*default*): the mean

Elemental: the elemental mean

ElementalNodal: the unaveraged

Compression Workflow

Specifies the workflow for compressing the results file.

None (*default*)

POD Compression

Enhanced POD Compression

For more information, see Compression Workflows (p. 156).

Filtering Workflow

Specifies the workflow for compressing the results file.

None (*default*)

Convert to Skin

Corner Nodes

For more information, see Filtering Workflows (p. 157).

Exporting Results to an HDF5 File

When you finish specifying the settings for your export, do the following to convert the selected results to HDF and save them in an H5 file:

1. To save the exported results in one H5 file, select **Merge Results to a Single h5 File**.
2. Click **Export Results as HDF5**.

Depending on the size of your model and your available computing resources, this may take a few minutes.

The Results Translator saves the H5 file under the same name and directory path as the selected results file. For example, if you selected `file.rth`, the exported file is saved as `file.h5` in that directory.

The file `h5Log_file.log` stores status messages for the conversion process.

31.2.4. Compression and Filtering Workflows

Compression Workflows

Data passed through the compression workflow can be compressed with one of the following lossy algorithms.

POD Compression

Proper Orthogonal Decomposition (POD) compression captures the most relevant aspects of the data by discarding insignificant information,

Enhanced POD Compression

This enhanced version of POD compression uses additional vectors to reconstruct more accurate results by capturing the discrepancies between the original data and its "smooth" representation.

Both workflows allow you to choose the desired threshold for compression. The threshold value is based on the desired precision of the decompressed data. This value should be between 0 and 1, where 0 gives the maximum possible precision of the decompressed data with the asked earlier threshold and 1 gives the results from POD compression. The default value for the compression threshold is 1e-4.

Important:

Ansys recommends that you use these two workflows only on the results of a static, transient, or harmonic analysis. For mode superposition analysis, use the On Demand Expansion procedure and the existing modal basis instead of POD or Enhanced POD compression. For more information about this solution option, see the discussion of modal analyses in Analysis Types and Analysis Settings and Solution.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filtering Workflows

Filtering workflows are used to transform and reduce the number of data stored in the result file.

Convert to Skin

Reduces the amount of data stored in the result file by only storing the results on the outer skin of the model. This workflow transforms the mesh so that only skin nodes are kept. It then creates a new layer of surface elements to map the surface of the model. The original mesh and volume results are discarded unless some other results use a different option.

Corner Nodes

Used only in models with quadratic elements (such as Hex20, Tet10, or Quad8). In these models, associated results are typically stored in mid-side nodes because these nodes are used in the solution. In some cases, you might want to discard these results because corner nodes have enough information to represent an accurate enough solution.

When you select this workflow, the solution will be extrapolated to the mid-side nodes when reading the result file. This creates a different solution for these nodes as a simple interpolation will be performed.

Normalized High Pass Filter

Filters small values from the result file. This is useful in models where the interest zone is small or there is a concentration of stresses in a particular zone. Enter a threshold for the maximum value of the result for the step that will be used for filtering.

31.3. Using the Solver Deck Translator

Use the Solver Deck Translator to convert Abaqus input files to Ansys-readable `*.cdb` and related files. You can then import the converted file into the Ansys Mechanical application.

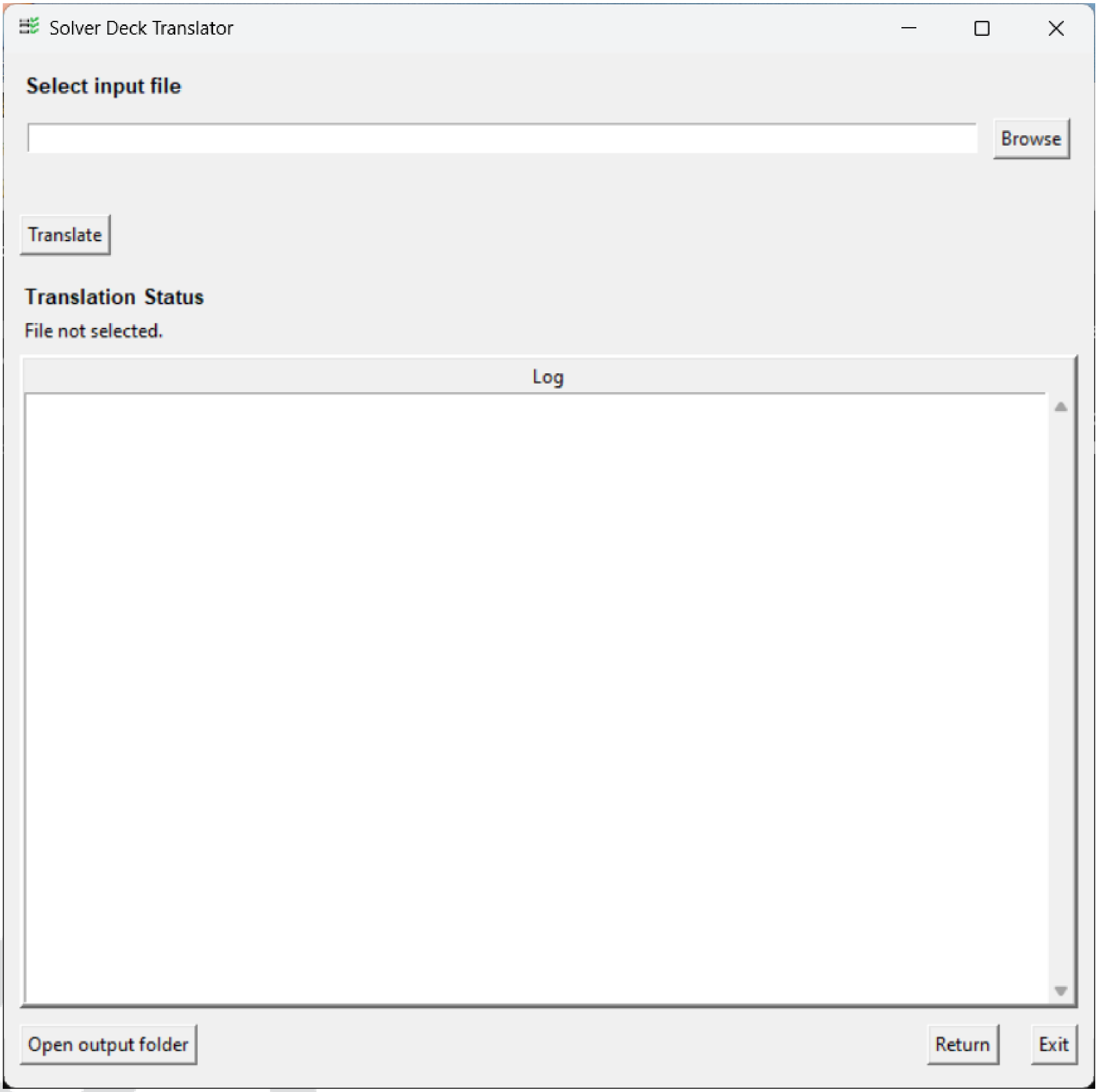
This application is only available through the **Data Translator**, which is opened independently of the Mechanical application.

See the following sections for more information about this tool:

31.3.1. Translating Abaqus Files

To use the **Solver Deck Translator** to translate Abaqus files:

1. Run the **Data Translator** as described in Accessing the Data Translator Application (p. 145).
2. Click **Solver Deck Translator**. The following dialog box is displayed:



3. Choose the Abaqus file you wish to convert. Enter the file name and directory path of the Abaqus result file in the **Select Input File** field. Alternatively, click the **Browse** button to navigate to the desired file and load it. The supported file format is Abaqus input format (`*.inp`).

4. Click **Translate** to launch the translation process.

   The **Log** area displays information about the status of the translation. See the discussion of the summary file under Solver Deck Translation Results (p. 159).

5. Click **Open output folder** to open the folder in which the translated files are stored. By default, this is the same directory as the input file.

 Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **Return** to close the **Solver Deck Translator** and returns to the **Data Translator**. Click **Exit** to close all applications.

31.3.2. Solver Deck Translation Results

The Solver Deck Translator generates the following files, which are stored in the same directory as the Abaqus input file. *filename* is the name of the input file.

***filename*.cdb**

A Coded Database (CDB) file that contains the translated model.

***filename*_server.txt**

A text file that logs errors and warnings from the Ansys Prime Server.

***filename*_summary.txt**

A log file that summarizes the translation process and lists any errors that may have occurred. This file is also displayed in the **Log** area of the **Solver Deck Translator** dialog.

***filename*_Run_MAPDL.dat**

The Mechanical APDL input file generated for the model.

***filename*_PyPrime.txt**

Logs Python-related errors during the translation process.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 32: Strain Scaling Factor Tables

A new dependent variable – **Strain Scaling Factor (Beta)** – was added to the Tables function. The strain scaling factor (SSF) is a direct multiplier to predicted strain values in laser powder bed fusion (LPBF) additive simulations using the LPBF Process Add-on. It is used as a calibration factor to account for differences in machines and materials.

The table function allows you to specify strain scaling factor values that vary by location (that is, by X and Y-coordinates). For more information on how to use strain scaling factor tables, see Location Specific Strain Scaling Factor (Beta) in the *Additive Manufacturing Beta Features*.

Requirements and Limitations for Strain Scaling Factor Tables

Be aware of the following when you are creating tables that include strain scaling factor values.

- The dependent variable **Strain Scaling Factor** is only used by the LPBF Process Add-on. The Mechanical APDL solver ignores this variable for other simulation types.
- Select **X Coordinate** and **Y Coordinate** as the independent variables for strain scaling factor tables. The LPBF Process Add-on ignores other independent variables.
- Strain scaling factor tables must follow the Table Requirements and Limitations.

BETA

BETA

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.