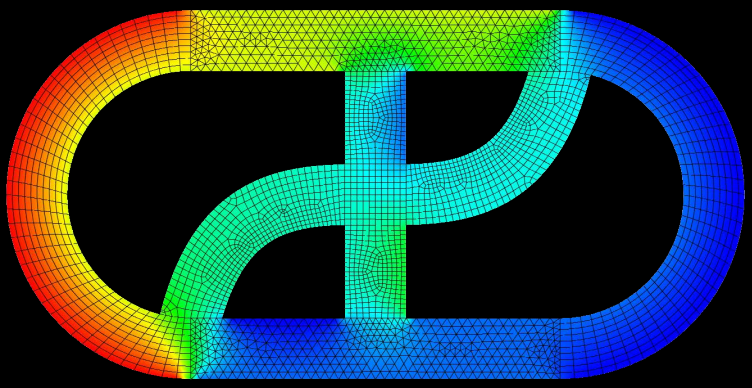


CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

©2021 ANSYS, Inc.
All Rights Reserved.
Unauthorized use, distribution
or duplication is prohibited.

Workbench Beta Features Manual

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 2021 R2
July 2021
ANSYS, Inc. and Ansys Europe, Ltd. are UL registered ISO 9001:2015 companies.

Copyright and Trademark Information

© 2021 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.

Table of Contents

Introduction 1
Enabling Beta Features in Workbench 1
Workbench Beta Features 3
CFX (Beta) Component System 3
Fluent (with CFD-Post) (Beta) Component System 4
Retaining Failed Design Points 4
Conditional Component Updates 5
Selecting the Condition Parameter 5
Specifying the Update Condition for the Input Parameter 6
Viewing Update Results 8
User String (Beta) Field 9
Fluent and EnSight 10
Acoustics in the Coupled Field Static System 10
Link Between Static Structural Solution to Speos Geometry 10
Connecting the optiSLang Data Receive Component to External Data 11
Connection from Fluent Meshing to CFX and Polyflow 12

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Introduction

Beta features are features that have not been fully tested and validated. They are working features that are still under development. Given this status, Ansys, Inc. may choose to release a feature, change the feature, or remove the feature from a given future release or as determined by the company. Furthermore, Ansys, Inc. makes no commitment to resolve defects reported against a beta feature and beta features are not subject to the Ansys Class 3 error reporting system.

Note:

- Turning the beta feature option on in Workbench also exposes beta options in supported applications, such as Mechanical. This guide does not document beta features for any other Ansys product other than Workbench.
- Ansys, Inc. does not guarantee that 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 input files before running them on other versions.

Enabling Beta Features in Workbench

To enable beta features in Workbench:

1. From the menu bar, select **Tools** → **Options** → **Appearance**.
2. Select the **Beta Options** check box.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Workbench Beta Features

The following topics describe beta features that appear in Workbench:

CFX (Beta) Component System

Fluent (with CFD-Post) (Beta) Component System

Retaining Failed Design Points

Conditional Component Updates

User String (Beta) Field

Fluent and EnSight

Acoustics in the Coupled Field Static System

Link Between Static Structural Solution to Speos Geometry

Connecting the optiSLang Data Receive Component to External Data

Connection from Fluent Meshing to CFX and Polyflow

CFX (Beta) Component System

Ansys CFX allows you to perform fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. The **CFX (Beta)** system can suppress physics errors from CFX-Pre and solve the calculations with the errors suppressed.

To use **CFX (Beta)** with Workbench:

1. To add an **CFX (Beta)** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To read a mesh and specify the Ansys CFX physics definitions in CFX-Pre:

   - Right-click the **Setup** cell and select **Edit** from the context menu.

   - To Import a previously saved case file, right-click the **Setup** cell and select **Import Case** → **Browse** from the context menu.

3. To suppress the physics errors, right-click the **Setup** cell and select **Suppress Physics Errors** from the context menu.

4. To start the solver:

   - Right-click the **Solution** cell and select **Update** from the context menu.

   - Right-click the **Solution** cell and select **Edit** from the context menu, set the solver execution controls in CFX-Solver Manager, and start the solver.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To import an existing CFX-Solver Results file, right-click the **Solution** cell and select **Import Solution** → **Browse**.

Fluent (with CFD-Post) (Beta) Component System

The **Fluent (with CFD-Post) (Beta)** component system allows you to complete fluid flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You specify the computational models, materials, boundary conditions, and solution parameters in Fluent, where the calculations are solved. You can then analyze the results of the calculations in CFD-Post.

To use **Fluent (with CFD-Post) (Beta)** with Workbench:

1. To add an **Fluent (with CFD-Post) (Beta)** component system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

2. To open Fluent in solution mode, where you can specify the computational models, materials, boundary conditions, and solution parameters, as well as perform the calculations:

   - Right-click the **Setup** cell and select **Edit** from the context menu.

   - To Import a previously saved Fluent case file or Fluent case and data files, right-click the **Setup** cell and select **Import Fluent Case** → **Browse** or **Import Fluent Case and Data** → **Browse** from the context menu.

3. To start the solver:

   - Right-click the **Solution** cell and select **Update** from the context menu.

   - Right-click the **Solution** cell and select **Edit** from the context menu, set the solver execution controls in Fluent, and start the solver.

4. To analyze the results of the calculations in CFD-Post, right-click the **Results** cell and select **Edit**.

Retaining Failed Design Points

By default, Workbench does not retain design points after they are solved or if they fail.

To keep the calculated data and/or files for design points that fail:

1. In the **Project Schematic**, double-click the **Parameter Set** bar.

2. To save the failed design point files, in the **Properties** view, select the **Keep Failed Design Points Files (Beta)** check box.

3. To save the failed design point calculated data, in the **Properties** view, select the **Retain Failed Design Points (Beta)** check box.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Conditional Component Updates

Output parameters come from a component called a *cell*. The values for an output parameter are set when that cell is updated. The **Outline** view for the **Parameter Set** bar displays all input and output parameters, grouping them by the cell to which they belong.

This section provides information on a beta feature for specifying an input parameter condition, determining whether a project or design point update is performed on a cell and any downstream cells.

During a project or design point update, Workbench checks conditional parameters for each design point, updating the design point only if logic outcomes are **True**.

Note:

An input parameter condition is not applied during a component update or a system update. For such updates, you see a warning message like this one:

```
The update condition parameter P1 for the Solution component in
Fluid Flow (CFX) evaluates to false. This component will not be
updated by project or design point updates.
```

Because updating all cells in a large project can take a long time, this beta feature allows you to avoid updating some cells based on the results of updating other cells. If you know beforehand that updating a cell is not useful for certain input parameter values, you can use this beta feature to set a condition that prevents updating the cell unnecessarily.

Selecting the Condition Parameter

You can set an update condition for any cell. Your first step is to select the input parameter to specify an update condition for.

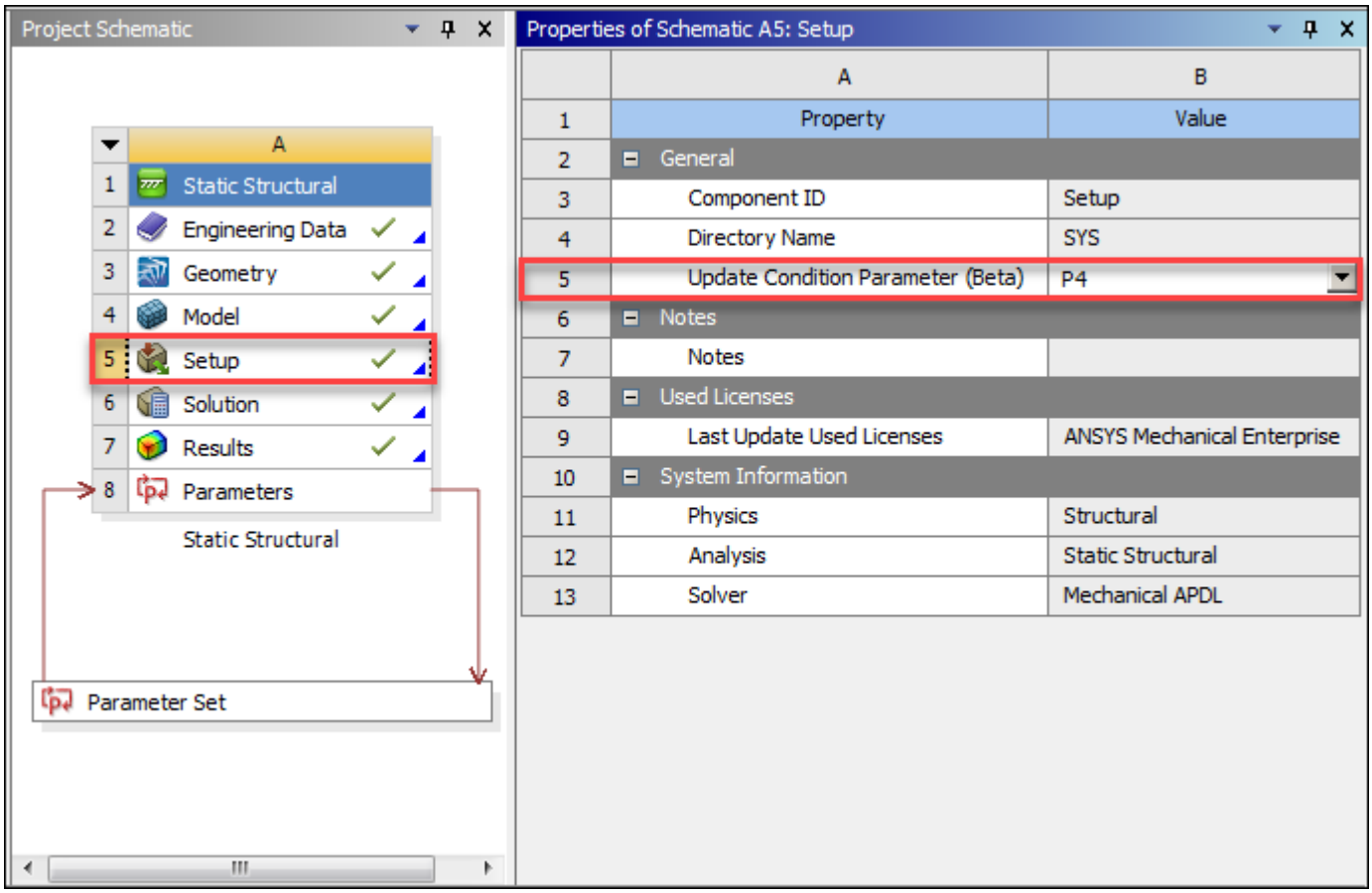
To select the condition parameter:

1. To open the **Properties** view, right-click a system cell and select **Properties**.
2. From the **Update Condition Parameter (Beta)** drop-down list, select the required input parameter identifier.

   You can select from all Boolean input parameters for the cell. By default, **None** is selected, indicating that a condition parameter has not been set. If you select a parameter that is later deleted, you have the option of changing the selection back to **None**.

In the following image, **Update Condition Parameter** for cell A5 is set to P4. The next step is to specify the update condition in the properties for input parameter P4.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specifying the Update Condition for the Input Parameter

After selecting the input parameter to set an update condition for a cell, you must specify the logic to use for update determination in the **Properties** view for this input parameter.

1. In the **Project Schematic**, double-click the **Parameter Set** bar.
2. In the **Outline** view, select the required input parameter.
3. In the **Properties** view, click the **Expression Value** cell and type the logic for determining whether to perform a project update or design point update on this cell and all downstream cells.

In the following image, P4 is selected in the **Outline** view and `P2>=5[Pa]` is entered as the expression in the **Property** view. If the output parameter value for P2 is greater than or equal to 5[Pa], the logic outcome for P4 is **True**. Otherwise, the logic outcome for P4 is **False**. For information about entering expressions, see *Expressions, Quantities, and Units* in the Workbench User's Guide.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Outline of Schematic A8: Parameters

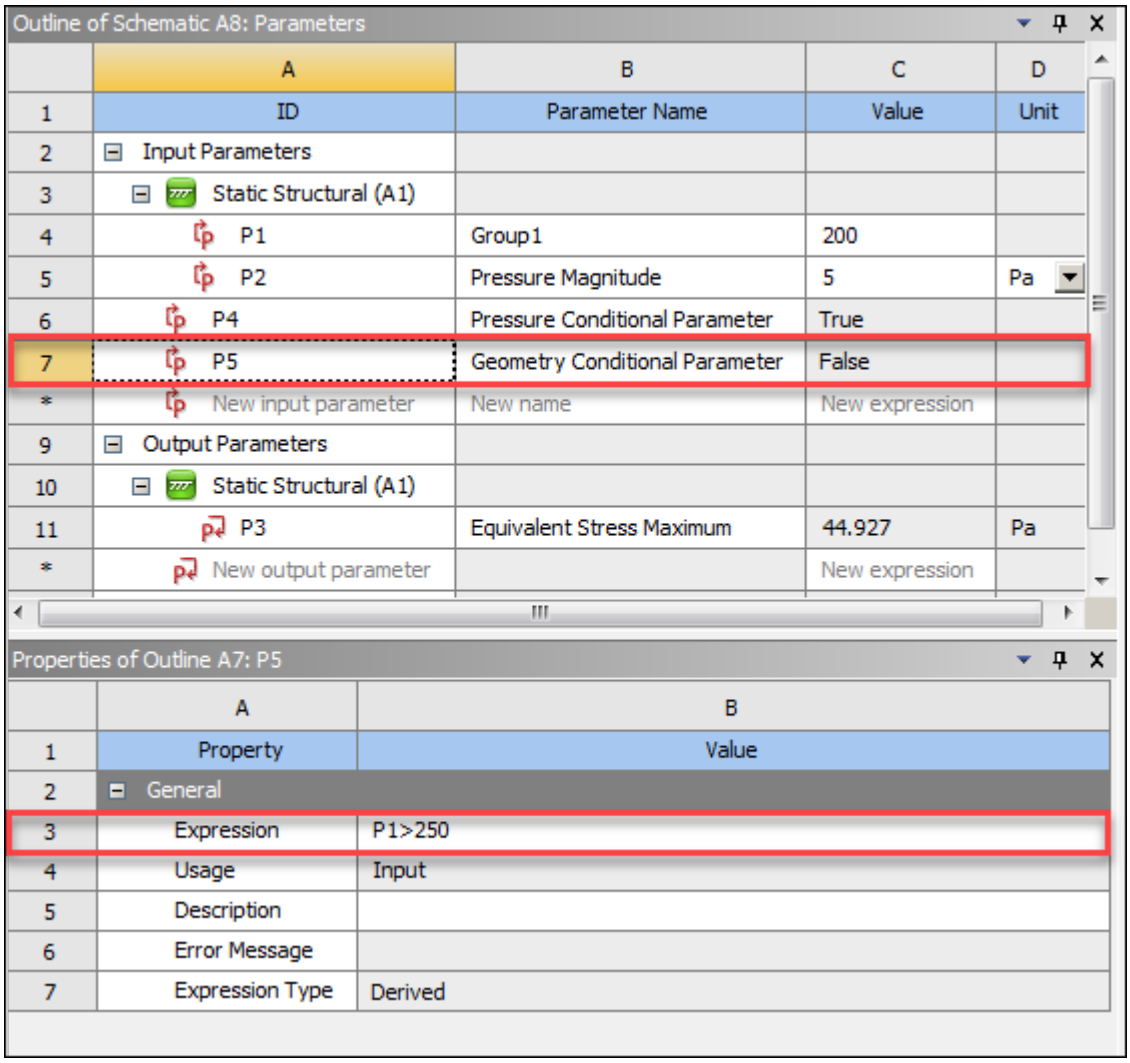
	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Static Structural (A1)			
4	P1	Group1	200	
5	P2	Pressure Magnitude	5	Pa
6	P4	Pressure Conditional Parameter	True	
7	P5	Geometry Conditional Parameter	False	
*	New input parameter	New name	New expression	
9	Output Parameters			
10	Static Structural (A1)			
11	P3	Equivalent Stress Maximum	44.927	Pa
*	New output parameter		New expression	
13	Charts			

Properties of Outline A6: P4

	A	B
1	Property	Value
2	General	
3	Expression	P2>=5[Pa]
4	Usage	Input
5	Description	
6	Error Message	
7	Expression Type	Derived

In this project, P5 is also a conditional parameter. In the properties for the **Geometry** cell, P5 is selected for **Update Condition Parameter**. The following images displays the update condition for P5. The Geometry cell and any downstream cells are updated only if the output parameter value for P1 is greater than 250.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Viewing Update Results

The design points table displays the logic outcomes for conditional parameters. When the logic outcome for any conditional parameter is **False**, the output parameter associated with the cell is not updated. All output parameter values that are not updated display an out-of-date icon.

In the following image, the design points table displays logic outcomes for P4 and P5 in columns named **P4 – Pressure Conditional Parameter** and **P5 – Geometry Conditional Parameter**. Because P4 and P5 were created after design point DP 0 was already up-to-date, DP0 still displays a value for output parameter P3. However, if you modified DP 0 and then updated all design points, DP 0 would display an out-of-date icon for P3 because it does not get updated when the logic outcome for P5 is **False**. DP 2 is the only design point for which P3 is updated because it is the only one where the logic outcomes for both condition parameters are **True**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Design Points

	A	B	C	D	E	F	G	H
1	Name	P1 - Group1	P2 - Pressure Magnitude	P4 - Pressure Conditional Parameter	P5 - Geometry Conditional Parameter	P3 - Equivalent Stress Maximum	Retain	Retaine
2	Units		Pa			Pa		
3	DP 0 (Current)	200	5	True	False	44.927	☑	✓
4	DP 1	250	6	True	False	⚡	☐	
5	DP 2	300	7	True	True	71.001	☐	
6	DP 3	100	8	True	False	⚡	☐	
7	DP 4	150	10	True	False	⚡	☐	
8	DP 5	300	1	False	True	⚡	☐	
9	DP 6	400	2	False	True	⚡	☐	
10	DP 7	500	3	False	True	⚡	☐	
11	DP 8	600	4	False	True	⚡	☐	
*							☐	

User String (Beta) Field

The **User String (Beta)** field allows you to provide custom information from Workbench to Remote Solve Manager (RSM) that performs custom actions prior to the submission of a job to the cluster.

Examples of custom information that can be provided to the cluster include the user name of the submitter, or the license necessary to execute the job.

This field is displayed in the **Properties** view when you set the **Solution Process Update Option** to **Submit to Remote Solve Manager** for an entire project, a system solution cell, or a parameter set.

The field accepts a text string in the following format:

`KEY1=value1,KEY2=value2`

The following rules apply to the text entry:

- Leading and trailing spaces around any key or value are automatically trimmed.
- Keys must consist of alphanumeric characters and underscores.
- Keys cannot contain embedded spaces.
- Values can use most characters and embedded spaces.
- Values cannot include the following special characters:

  `=,:;"\'/`()[]{}<>`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

What you type in this field is passed to RSM as environment variables. The name of each environment variable is the uppercase key, prefixed with RSM_CLIENT_, unless the key already starts with the prefix. The value of each environment variable is the value portion of each pair.

Note:

Workbench does not check the size of the user string against the environment variable space of the remote Workbench session.

Fluent and EnSight

If you are running Fluent in Workbench with Workbench BETA features enabled, Fluent exports `.cas/.h5/.dat.h5` files during the run. You can equivalently enable this feature in EnSight by selecting **Advanced EnSight features** → **Enable Beta Features** on the EnSight Task.

Acoustics in the Coupled Field Static System

The Coupled Field Static system determines the displacements, stresses, strains, and forces caused by loads that do not induce significant inertia and damping effects. This system currently supports coupling between structural and thermal physics types, during a static analysis.

It now additionally supports coupling between structural and acoustic physics types. It is off by default. To use this feature:

1. From the menu bar, select **Tools** → **Options** → **Appearance**.
2. Ensure that the **Beta Options** check box is selected.
3. From the list on the left side of the pane, select **Mechanical**.
4. Under **Beta feature selection**, select the **Allow Acoustics in Coupled Field Static** check box.
5. Click **OK**.

You can now enable **Acoustic** physics in the in the **Setup** cell properties by defining a Physics Region with Acoustics physics in Mechanical.

Link Between Static Structural Solution to Speos Geometry

You can now create a connection between the **Solution** cell of a Static Structural system and the **Geometry** cell of a Speos system. When the link is created, `pmdb` and `obj` files containing geometry, mesh, and deformed mesh information are transferred. From this information, warping is applied to the geometry.

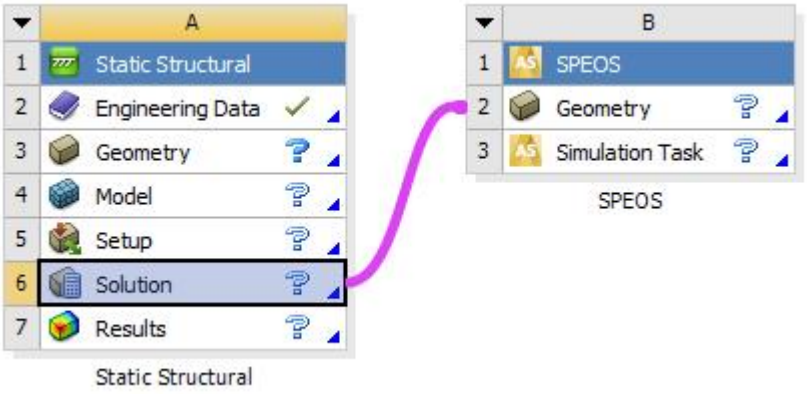
To create the link:

1. To add a Static Structural system, drag the system from the **Toolbox** to the **Project Schematic** or double-click the system in the **Toolbox**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Right-click the **Solution** cell and select **Transfer Data to New → SPEOS**.

   The Speos system is added and the connection is made.



Connecting the optiSLang Data Receive Component to External Data

You can now connect an optiSLang Data Receive component to an External Data component use this connection in design points. To do so:

1. Ensure that the Ansys Workbench optiSLang extension is installed.
2. Drag a **Data Receive** component from the **Toolbox** (listed under **optiSLang Integrations**) onto the **Project Schematic**.
3. Double-click the **Setup** cell, or right-click the **Setup** cell and select **Edit** from the context menu.
4. In the **PDMReceive - Data Receive** dialog box, click **Add**.
5. For the PDM location, in the **Type** list, select **optiSLang** and browse to the location of the file to set the path.

   Selecting this type creates an input and output parameter.
6. Click **Next**.
7. Select the optiSLang related file and click **Next**.
8. Keep or adjust the ID as required and click **Next**.
9. In the **Transfer type** list, select **External Data** and click **Next**.
10. To save the settings and close the wizard dialog box, click **Finish**.
11. Right-click the newly created cell that corresponds to the selected file and select **Transfer Data to New → External Data** from the context menu.
12. Set up the remainder of the Workbench project as needed.
13. Save and close the project.

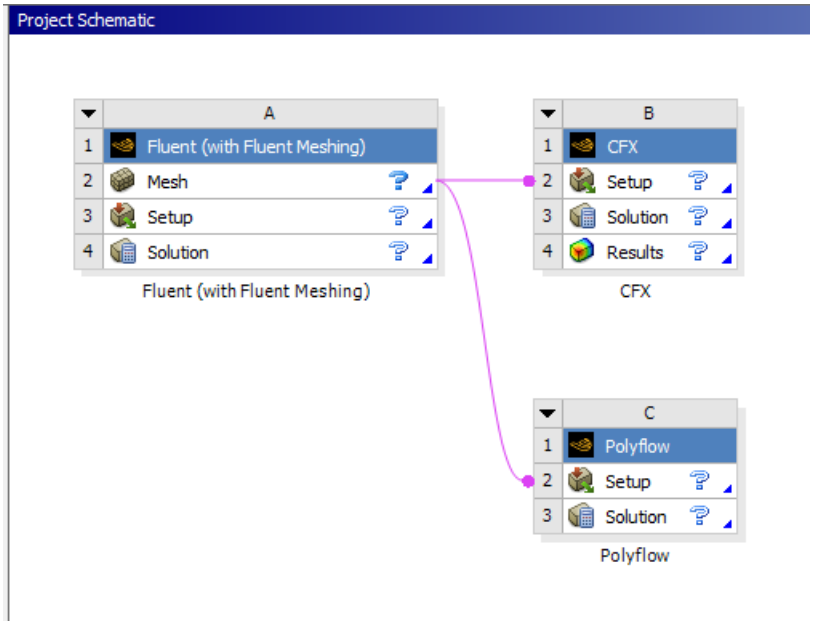
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. In optiSLang, load the Workbench project into a Workbench node. The node detects the Data Receive system and creates a file registration in the optiSLang project and the Workbench node in the Files tab.

15. Now you can place any other node that creates the necessary file in optiSLang before the Workbench node to prepare that file for design points.

Connection from Fluent Meshing to CFX and Polyflow

You can create a connection from a Fluent (with Fluent Meshing) system to a CFX and a Polyflow system.

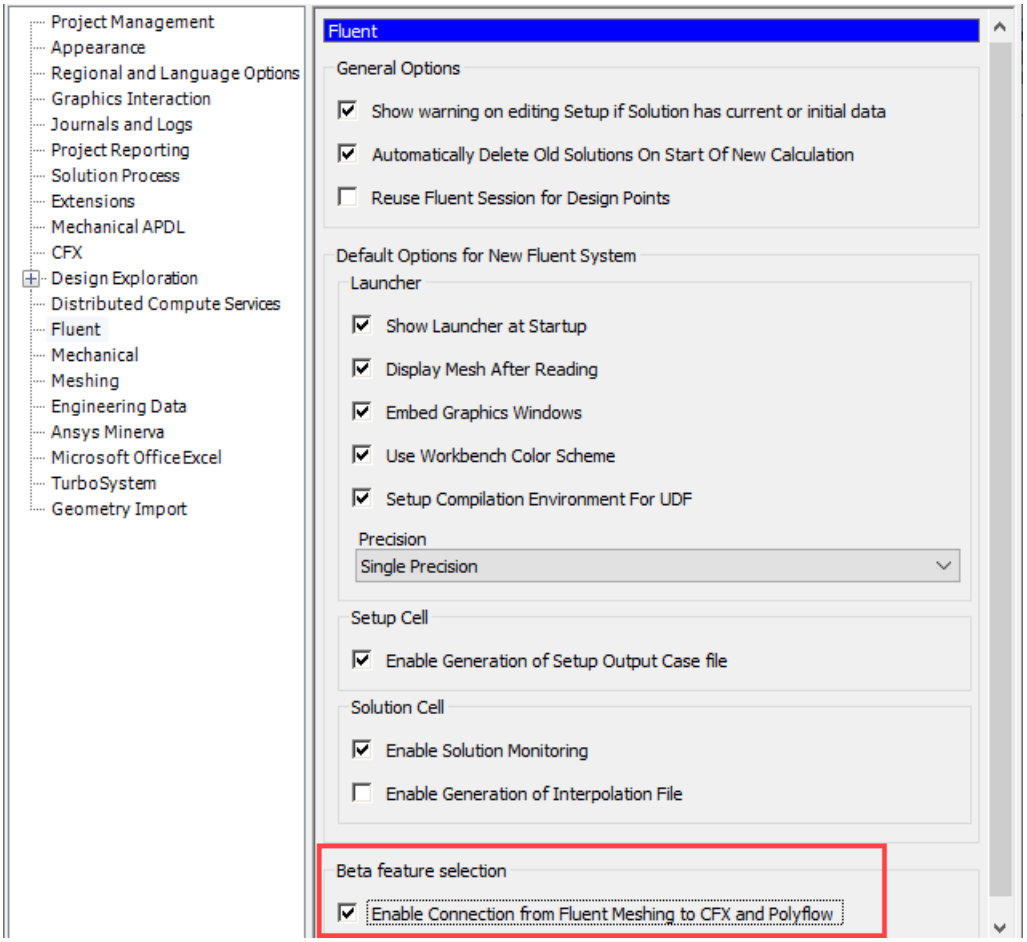


To enable this workflow:

1. From the menu bar, select **Tools** → **Options** → **Fluent**.

2. Select the **Enable Connection from Fluent Meshing to CFX and Polyflow** check box.

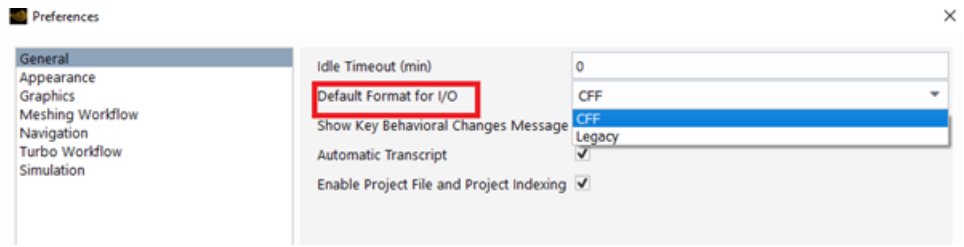
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. To save the changes and close the dialog box, click **OK**.

4. Complete one of the following options:

   - Open Fluent Meshing and manually change the **Default Format for I/O** setting to **Legacy**.



   - Run a command snippet in Fluent to change the **Default Format for I/O** setting to **Legacy**.

```
Meshing> (pref-getvar "General/DefaultIOFormat")
CFF

Meshing> (pref-setvar "General/DefaultIOFormat" "Legacy")
()

Meshing> (pref-getvar "General/DefaultIOFormat")
Legacy
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more details on Fluent, see the Fluent User's Guide.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.