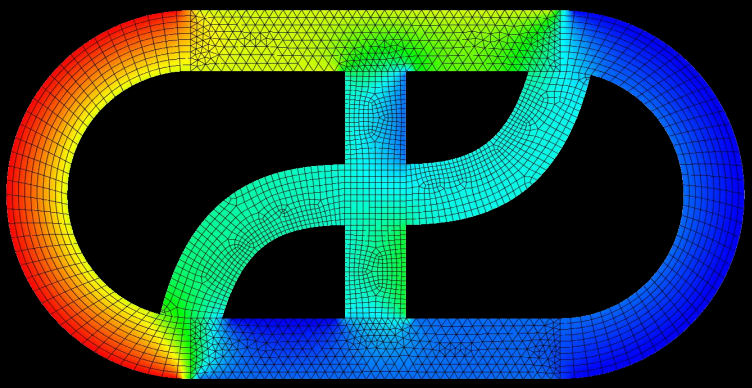


CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys 2021 R2
Engineering What's Ahead.

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

DesignXplorer 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

Beta Features 1
DesignXplorer Beta Features 3
F-Test Filtering 3
Export a Response Surface as a VHDL-AMS Language File 4
Define as Starting Point 5
Cumulative Distribution Plot Type 5
Export ROM Snapshots to Twin Builder 7
Manual Production of ROM Files from Standalone Fluent 7
Setting Up the Workbench Project 8
Setting Up ROM Building in Fluent 8
Creating the ROM in the ROM Builder 10
Creating Mesh and Snapshot Files in Fluent Standalone 11
Importing the Mesh and Snapshot Files into the DOE and Generating the ROM 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.

Beta Features

Beta features have not been fully tested and validated. Ansys, Inc. may, at its discretion, fully release, change, or withdraw beta features in future revisions. It makes no commitment to doing so on any particular schedule. Ansys, Inc. makes no commitment to resolve defects reported against beta features. However, your feedback will help us improve the quality of the product.

Beta features are not subject to the Ansys Class 3 error reporting system. 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.

To enable beta features in Workbench, select **Tools** → **Options** → **Appearance** and then 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.

DesignXplorer Beta Features

The following topics describe beta features that appear in various panes in DesignXplorer systems:

F-Test Filtering

Export a Response Surface as a VHDL-AMS Language File

Define as Starting Point

Cumulative Distribution Plot Type

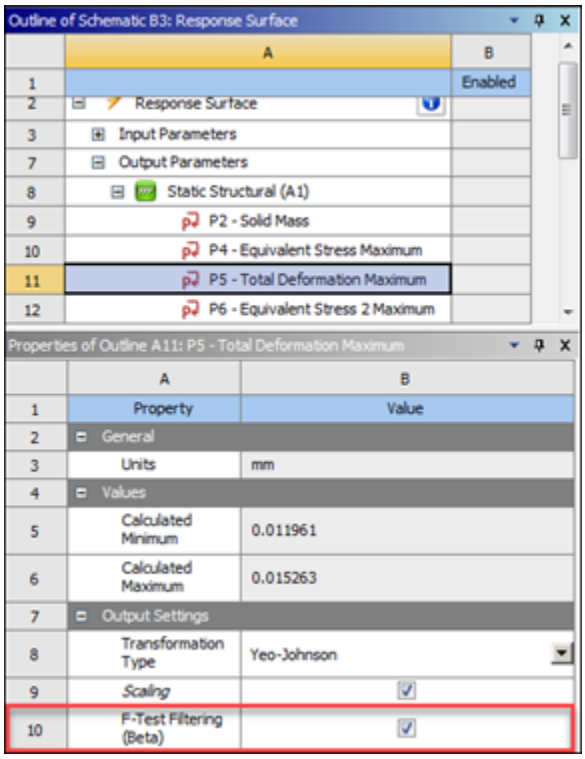
Export ROM Snapshots to Twin Builder

Manual Production of ROM Files from Standalone Fluent

F-Test Filtering

When you are configuring a **Response Surface** cell, you must set the type of response surface to generate. If you set **Response Surface Type** to **Standard Response Surface - Full 2nd-Order Polynomials**, **F-Test Filter (Beta)** displays in the properties for output parameters. This option, which appears under **Output Settings**, determines whether to filter significant terms of the polynomial regression. When the check box is selected (default), the **Significance Level** value specified in the response surface properties is used as the threshold for this filtering. If you clear this check box, significant terms are not filtered. All terms are kept.

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



Note:

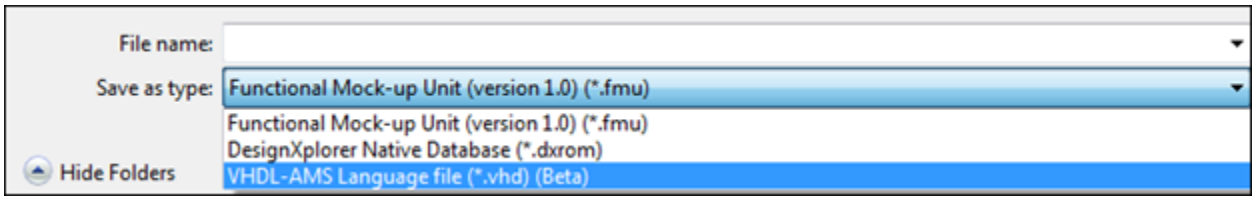
Setting **Significance Level** in the response surface properties to **1** is equivalent to clearing the **F-Test Filter (Beta)** check box for all output parameters.

Export a Response Surface as a VHDL-AMS Language File

When a **Response Surface** cell is open, the response surface that you generate can be exported to a VHDL-AMLS Language file if the following conditions are met:

- There are no discrete input parameters.
- The response surface type is either Full 2nd-Order Polynomials or Kriging.

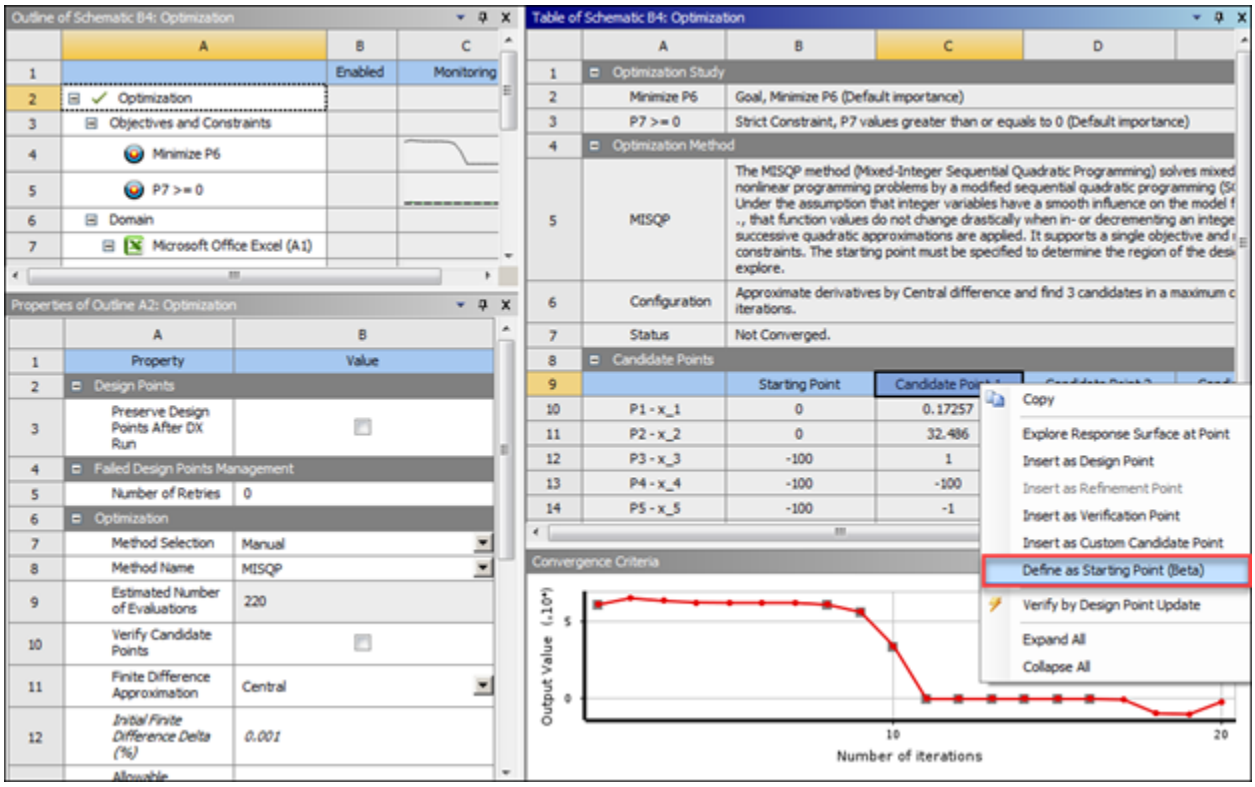
When these conditions are met, when you click **Export Response Surface** in the toolbar, **VHDL-AMS Language file (*.vhd) (Beta)** is listed as a file type to which you can save.



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

Define as Starting Point

When an **Optimization** cell is open and **Optimization** is selected in the **Outline** pane, the **Table** pane displays candidate points. If you right-click a candidate point, **Define as Starting Point (Beta)** is available in the context menu if the optimization method is NLPQL, MISQP, or an external optimization method defined in an ACT extension that requires a starting point. When this option is selected, the values for this candidate's input parameters are used to set the starting point.



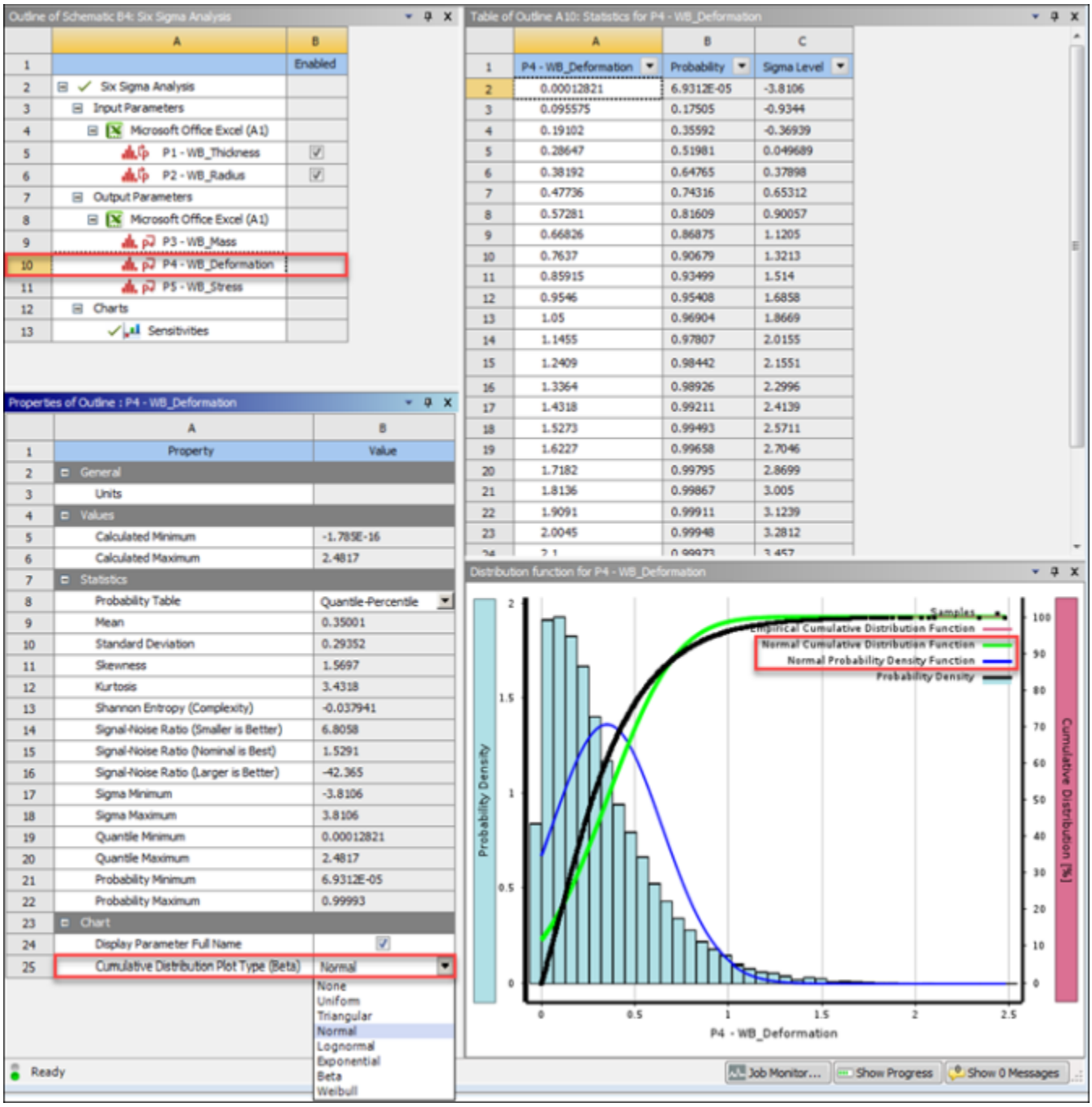
Cumulative Distribution Plot Type

The sample points generated by a Six Sigma Analysis allow the building of statistical results such as the Probability Density Function and the Empirical Cumulative Distribution Function for each parameter. This means that you can see the propagation of uncertainties on input parameter to output parameters.

If you know and define the distribution types of input parameters in the **Design of Experiments** cell, the distribution types of output parameters are unknown. When you select an output parameter in the **Outline** pane for the **Six Sigma Analysis** cell, the **Properties** pane displays chart properties. For **Cumulative Distribution Plot Type (Beta)**, you select a distribution type. All attributes of this distribution, such as mean, standard deviation, and so on, are then automatically selected to fit as best as possible the Empirical Cumulative Distribution Function.

You can see which distribution type is closest to the empirical distribution by comparing two Cumulative Distribution Functions and by comparing the Probability Density Function with the Probability Density.

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



By default, **Cumulative Distribution Plot Type (Beta)** is set to **Normal**. This means that in the chart for the output parameter, the solid blue and green curves are the cumulative distribution function and probability density function based on the normal distribution.

If you set **Cumulative Distribution Plot Type (Beta)** to a different distribution type, the labels for the solid blue and green curves are updated according. If you set **Cumulative Distribution Plot Type (Beta)** to **None**, no solid blue or green curves are shown.

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

Export ROM Snapshots to Twin Builder

Users of Twin Builder can export the ROM snapshot files (ROMSNP) that are generated by the DOE in a DesignXplorer **3D ROM** system to build a static ROM in Twin Builder.

To export ROM snapshot files to Twin Builder:

1. Right-click in an empty area of the Workbench **Project Schematic** and select **Export ROM Snapshots to Twin Builder (Beta)**.
2. In the **Browse For Folder** window that opens, select or create the folder in which to save the snapshot files to the format supported by Twin Builder.

In the **Progress** view, you can see the snapshot files being exported.

Note:

- All snapshot files are exported, including those from any 3D ROM system and from any of these tables: Design of Experiments, Refinement Points, and Verification Points.
- In the exported snapshot files, the simplified mesh files contain only 3D-point coordinates. You can visualize these coordinates in the Twin Builder Static ROM Builder, where each coordinate is represented with a sphere.

Once the export finishes, you can start Twin Builder and select **Twin Builder → Launch Static ROM Builder**. In the **Select a project directory** window that opens, you select the folder to which the snapshot files were exported.

For more information about ROM snapshot files, search the DesignXplorer help for the "ROM Snapshot Files" topic.

For more information about the Twin Builder Static ROM Builder and viewer, search the Twin Builder help for the “Using Static ROM Builder” topic, paying attention to these child topics:

- “Preparing the Data,” which describes the format to which ROM snapshot files were exported
- “Viewer (Evaluate Tab),” which explains how to evaluate the output for given values of input parameters

Manual Production of ROM Files from Standalone Fluent

This section provides a ROM workflow example (beta) for advanced Fluent users only. If you are not an advanced Fluent user, you should use the standard workflow for automated ROM production, which is described in "Using ROMs" in the *DesignXplorer User's Guide*.

This advanced example describes how to manually define or generate in standalone Fluent all of the appropriate files needed to finally build a ROM in Workbench:

Setting Up the Workbench Project

Setting Up ROM Building in Fluent

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

Creating the ROM in the ROM Builder

Creating Mesh and Snapshot Files in Fluent Standalone

Importing the Mesh and Snapshot Files into the DOE and Generating the ROM

The case, which is for a steady fluids simulation of a heat exchanger, is run independently for each design point (design of experiments). The ROM-specific files are created manually in standalone Fluent for later import into a **3D ROM** system in a Workbench project.

Note:

- DesignXplorer advanced options must be enabled. Select **Tools** → **Options** → **Design Exploration** and then select the **Show Advanced Options** check box and click **OK**. DesignXplorer now displays advanced options in italic type in various panes. Advanced ROM options provide for opening the ROM Builder log file and setting a user-generated ROM mesh file.
- The CAS file for this advanced example is included in the same ZIP file as the sample files for the standard example. To download the ZIP file, in "Using ROMs" in the *DesignXplorer User's Guide*, see the "Prerequisites" topic.

Setting Up the Workbench Project

This section explains how to insert and configure in a Workbench project the Fluent system from which to build the ROM.

1. In the **Toolbox** under **Component Systems**, double-click **Fluent** to add a system of this type to the **Project Schematic**.
2. Right-click the **Setup** cell, select **Import Fluent Case** → **Browse**, navigate to the directory where you extracted the sample files, and open the CAS file (`HeatExchanger.cas`).

   The **Parameter Set** bar is added to the **Project Schematic**, indicating that input and output parameters are defined in the CAS file.
3. Double-click the **Parameter Set** bar to open it.
4. Verify that input and output parameters are defined appropriately, which they are.
5. Close the **Parameter Set** bar.
6. Save the project, specifying a location and file name.

Setting Up ROM Building in Fluent

This section explains how to enable and load the ROM feature in Fluent and then select flow variables and cell zones.

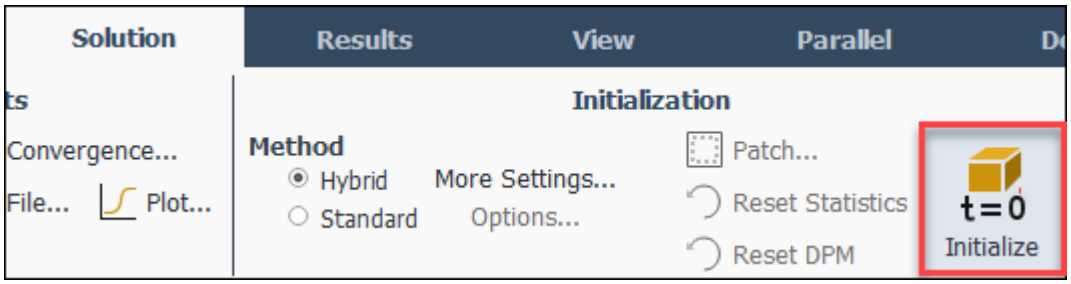
1. In the **Project Schematic**, double-click the **Solution** cell to open the case in Fluent.

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

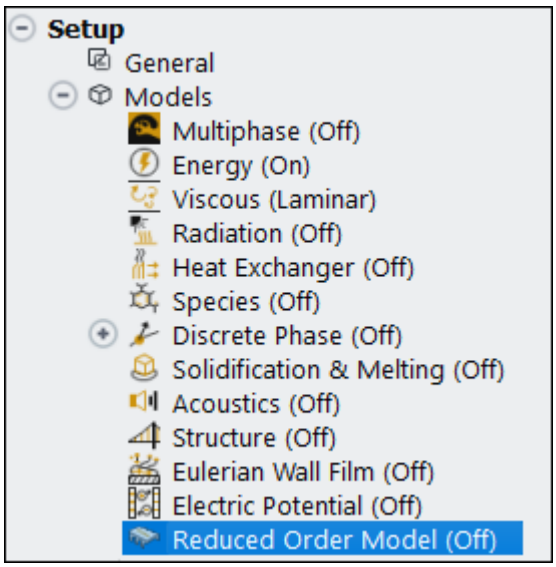
2. To make **Reduced Order Model (Off)** visible in the tree under **Setup** → **Models**, enable and load the ROM addon module by executing this command from the Fluent console:

   ```
   define models addon 11
   ```

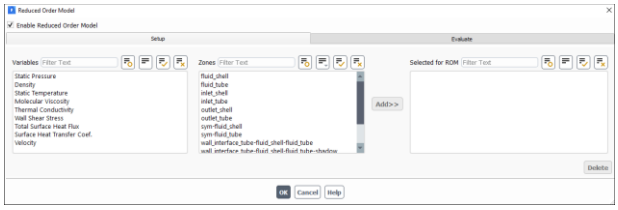
3. On the ribbon's **Solution** tab, click the button in the **Initialization** group for initializing the entire flow field.



4. In the tree under **Setup** → **Models**, double-click **Reduced Order Model (Off)**.



5. In the **Reduced Order Model** window that opens, select the **Enable Reduced Order Model** check box to expand the window so that you can set up the ROM.



6. For this example, select:

   - **Static Temperature**: all zones
   - **Velocity**: all zones except wall zones

7. Once you have finalized your selections, click **OK**.

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

8. Select **File** → **Export** → **Case**, name the file `HeatExchanger_RomSetup.cas`, and click **OK**.
9. Select **File** → **Close Fluent** to close Fluent and push your changes to the Workbench project.

Creating the ROM in the ROM Builder

This section explains how to add, configure, and update the **3D ROM** system in Workbench.

1. In the **Toolbox** under **Design Exploration**, double-click **3D ROM** to add a system of this type to the **Project Schematic**.

   A **3D ROM** system is inserted under the **Parameter Set** bar.
2. Double-click the **Design of Experiments (3D ROM)** cell to open it.

   For this example, you will keep only one input parameter for the ROM.
3. In the **Outline** pane, select **P1**.
4. In the **Properties** pane, set the lower and upper bounds for **P1** to **0.001** and **0.01** m/s.
5. In the **Outline** pane, clear the check boxes for **P2**, **P3**, and **P4** to disable them.

   Their constant values, which are reported in the **Properties** pane by default, can be kept.
6. In the **Outline** pane, select **Design of Experiments (3D ROM)**.
7. In the **Properties** pane, set **Number of Samples** to **4**.
8. In the toolbar, click **Preview** to generate the four design points without updating them.

Table of Outline A2: Design Points of Design of Experiments

	A	B	C	
1	Name	P1 - inlet_velocity_shell (m s^-1)	P5 - outlet_temperature_shell (K)	P6 - outle
2	1	0.004375		
3	2	0.008875		
4	3	0.002125		
5	4	0.006625		

9. Take note of the generated inlet velocity value for **P1**: **0.004375**.

> **Tip:**
>
> You can export the data in the table to a CSV file.

10. Save the project and close Workbench.

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

Creating Mesh and Snapshot Files in Fluent Standalone

This section describes how to use the previewed design points in the DOE to update the Fluent solution for each design point, create a mesh in the ROM format, and create the snapshot files that are then used to build the ROM.

Tip:

You can execute these steps on a cluster without using Workbench.

---

1. Start standalone Fluent.
2. Select **File** → **Read** → **Case** and select `HeatExchanger_RomSetup.cas`.
3. If an information dialog box opens, click **OK**.
4. On the ribbon's **User-Defined** tab, click **Execute on Demand** to open the **Execute on Demand** window.
5. Select the command **`write_rombuilder_setup_file::rom`** and click **Execute**.

   This generates the setup files that are used when creating the mesh and snapshot files. These setup files are written in the Fluent working directory and can be safely removed at the end of this procedure.
6. Close the **Execute on Demand** window.
7. In the **Console**, execute this command:

   ```
   (%rom-write-mesh-file "mesh.rommsh")
   ```

   This creates the file `mesh.rommsh` in the Fluent working directory. This mesh file is used to display the ROM.
8. In the tree, under **Parameters & Customization > Parameters > Input Parameters**, enter the value for each parameter for the first design point that you previewed in the DOE:
   a. Set **inlet_velocity_shell** to **0.004375** m/s.
   b. Leave the default values for the other parameters.
9. On the ribbon's **Solution** tab, do the following:
   a. Click the button in the **Initialization** group for initializing the entire flow field.
   b. When asked if you want to discard the data and proceed, click **OK**.
   c. Click **Calculate** in the **Run Calculation** group.
   d. When the calculate finishes, click **OK** to close the information window.
10. In the **Console**, execute this command:

    ```
    (%rom-write-snapshot-file "snp_1.romsnp")
    ```

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

This creates the file `snp_1.romsnp` in the Fluent working directory.

11. For each of the three additional design points generated in the **Design of Experiments** cell, repeat steps 8 through 10, entering the corresponding value for **inlet_velocity_shell** and the appropriate name for the snapshot file:

    - **0.008875** m/s and `snp_2.romsnp`
    - **0.002125** m/s and `snp_3.romsnp`
    - **0.006625** m/s and `snp_4.romsnp`

12. Close Fluent and download the ROMSNP and the ROMMSH files to the computer with Workbench installed.

Tip:

You can automate this step using Fluent scripting.

Importing the Mesh and Snapshot Files into the DOE and Generating the ROM

This section describes how to use the mesh file and four snapshot files to produce the ROM.

1. Locate the file `Rom.MeshProcessing.exe` in the Workbench installation folder under `Addins\ROM\bin\Win64` for Windows or `Addins\ROM\bin\Linux64` for Linux.

Note:

The next step, which can be computationally expensive, can be performed on a separate computer.

---

2. Create the file `mesh_processed.rommsh` from the command line:

    - For Windows, run the following command, where *`Path`* is the location where you want to create the file:

```
Rom.MeshProcessing.exe C:\Path\mesh.rommsh C:\Path\mesh_processed.rommsh
```

    - For Linux, the path in the following command assumes that Ansys is installed using the symbolic link **`/ansys_inc`**. If necessary, substitute your installation path for the path given:

```
/ansys_inc/v212/Framework/bin/Linux64/runwb2 -cmd /ansys_inc/v212/Addins/ROM/bin/Linux64/Rom.MeshProcessin
```

3. After the file `mesh_processed.rommsh` is created, start Workbench and open the project `HeatExchanger_RomProduction.wbpj`.

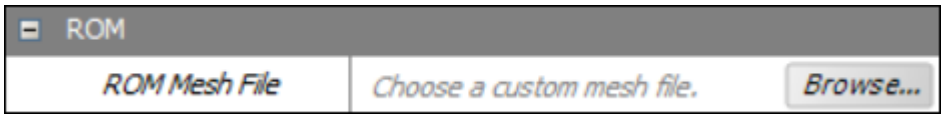
4. In the **Project Schematic**, double-click the **Design of Experiments (3D ROM)** cell to open it.

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

5. In the **Outline** pane, select **Design of Experiments (3D ROM)**.

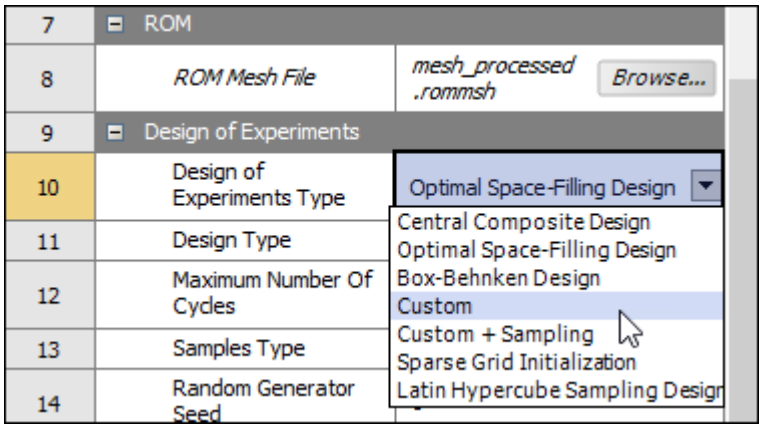
   When DesignXplorer advanced options are shown, **ROM Mesh File** is visible in the **Properties** pane.

6. For **ROM Mesh File**, click **Browse** and select the file `mesh_processed.rommsh` that you created in step 2.

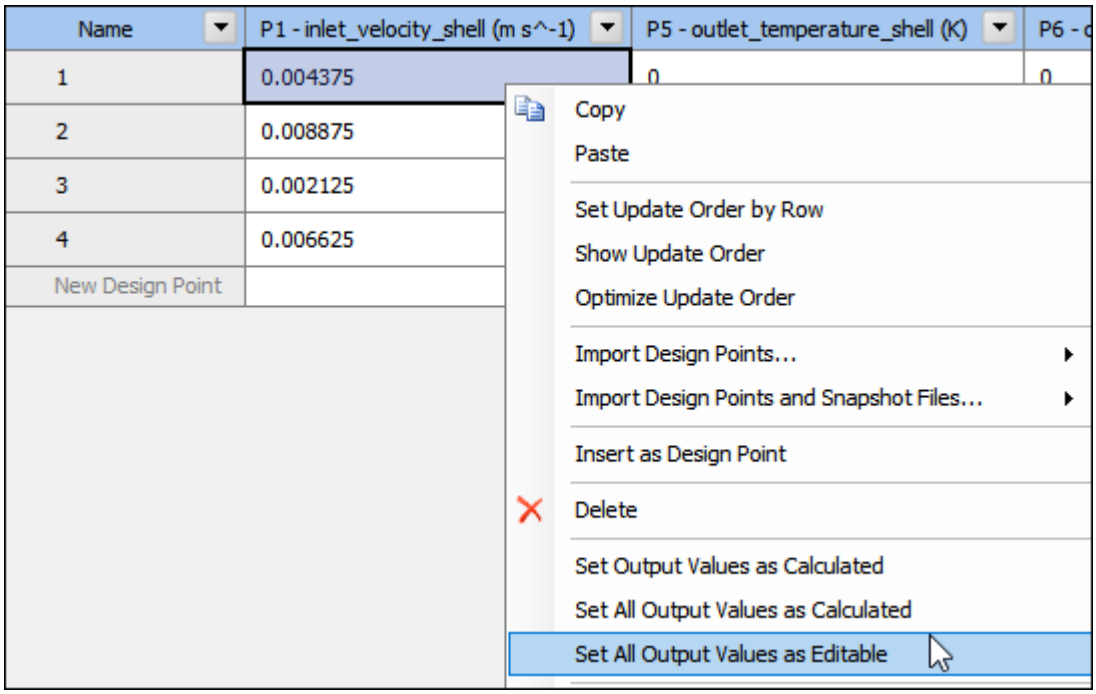


7. When prompted, select **Copy and Proceed**.

8. Set **Design of Experiments Type** to **Custom**.



9. Right-click in the **Table** pane and select **Set All Output Values as Editable**.



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

10. Manually set the snapshot file for each design point:

    a. In the first line, enter a dummy value such as **0** for **P5** and **P6**. This value is not used for ROM building.

    b. In the first line, right-click the **Fluent ROM Snapshot** cell, select **Set Snapshot File**, browse to `snp_1.romsnp`, and when prompted click **Copy and Proceed**.

    c. Repeat step b for the next three design points, selecting the corresponding snapshot file (`snp_2.romsnp` through `snp_4.romsnp`).

    Warning:

    When manually setting a snapshot file, if you mistakenly select a file that does not match the point, the resulting ROM will have invalid values. To avoid errors, you should automate this step by importing a ROM snapshot archive file (SNPZ). Like a 7z or ZIP file, a SNPZ file is a compressed file. It contains all snapshot files and one CSV file.

11. Save the project and update the **Design of Experiments (3D ROM)** cell.

    This operation is fast because all of the outputs are already calculated.

12. In the **ROM Builder** cell, set **Number of Modes** to **4**, which is the number of DOE points, and click **Update**.

13. When the update completes, in the toolbar, click **Open in ROM Viewer**.

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