How to create a large assembly from smaller sub-assemblies?

Assembly FEM workflows

Assembly FEMs support two basic workflows:

1. Associative workflow:
Associates an assembly FEM with an existing assembly of parts, mapping new or existing component FEMs to each component part.

2. Non-associative workflow:
Creates an empty assembly FEM first, then adds component FEMs to the assembly FEM, and finally defines the position and orientation of component FEMs.

Choosing between associative and non-associative Assembly FEM workflows depends on the specific requirements of your project, such as the need for design updates, the complexity of the model, and the desired level of integration between CAD and CAE.

Associative Assembly FEM Workflow

In an associative Assembly FEM workflow, the assembly FEM model is directly linked to the CAD model. Any changes made to the CAD model are automatically reflected in the assembly FEM model. This workflow ensures that the simulation model is always up-to-date with the latest design changes.

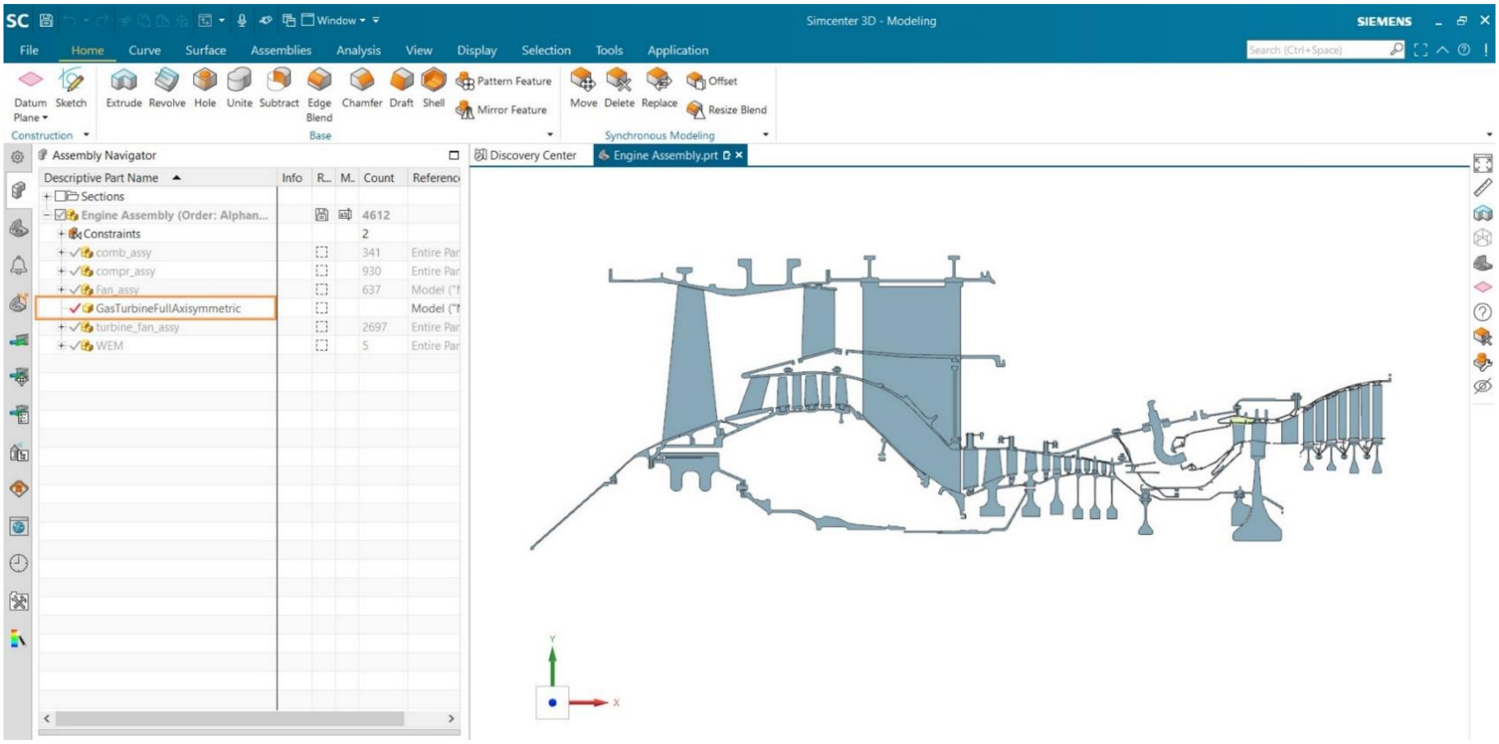
Use cases:

Recommended for: Projects where frequent design changes are expected, and maintaining an up-to-date simulation model is critical. This is particularly useful in the early stages of design when iterations are common.

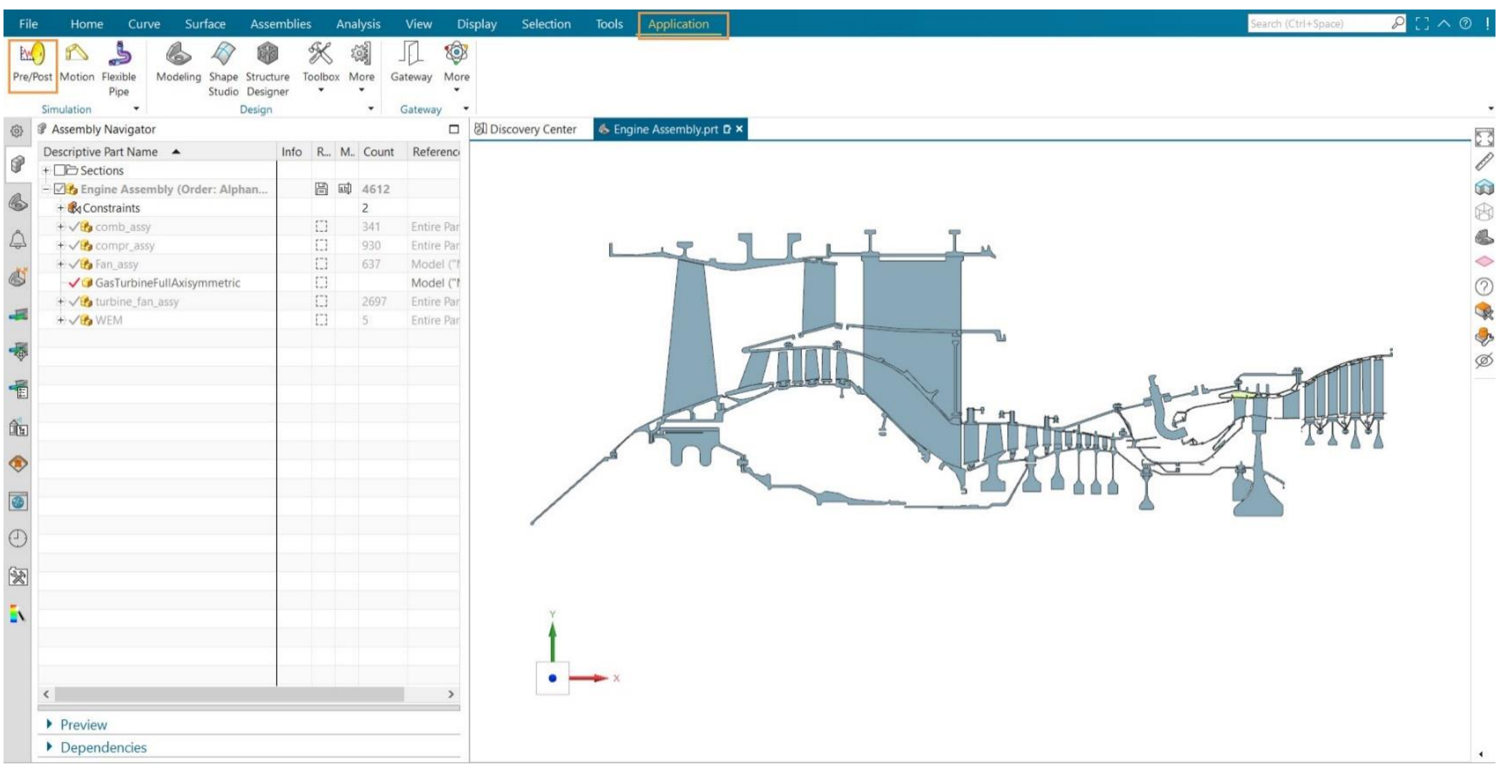
- **Turbomachinery**:
  Example: When developing a new turbine blade design, where iterative changes are made to optimize performance, the associative workflow ensures that the FEM model is always aligned with the latest design.
- **Space Systems Thermal**:
  Example: When designing a satellite's thermal control system, where iterative changes are made to ensure optimal heat dissipation.
- **Electronics Thermal Cooling**:
  Example: When developing a new cooling system for a high-performance server, where iterative changes are made to enhance cooling efficiency.

To create an associative assembly FEM,

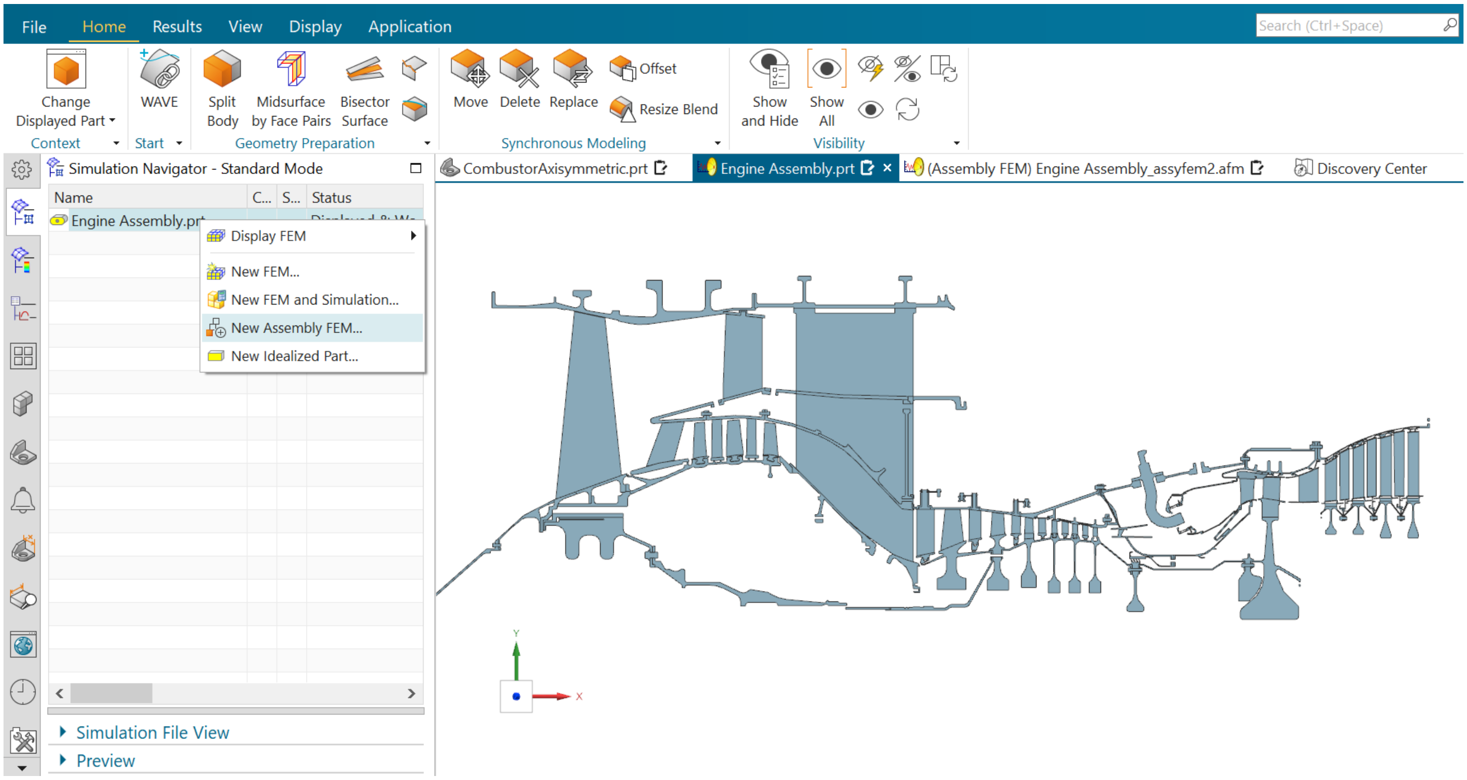
1. In the **Modeling** application, select the assembly part that you want to use for the analysis.
   In this example, the gas turbine axisymmetric model is used.



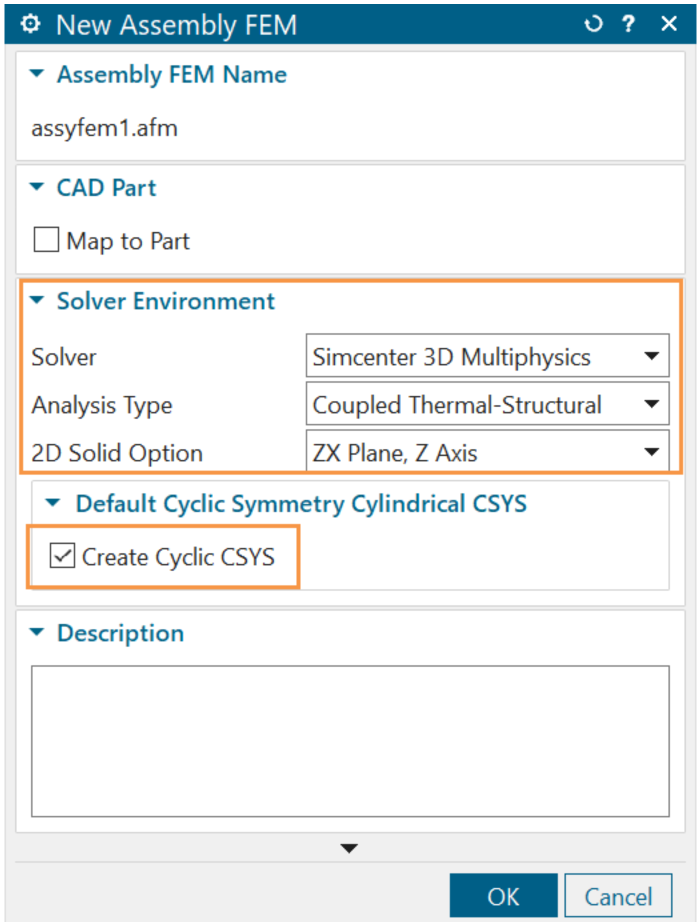
2. In the **Application** tab, select **Pre/Post**.



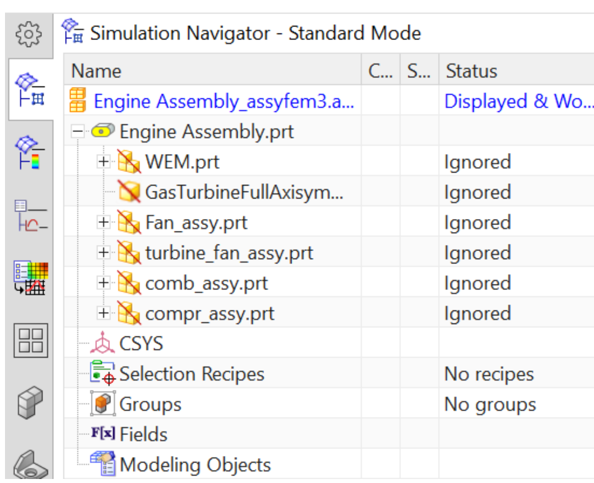
3. Right-click the assembly part node in the **Simulation Navigator**, and choose **New Assembly FEM**.



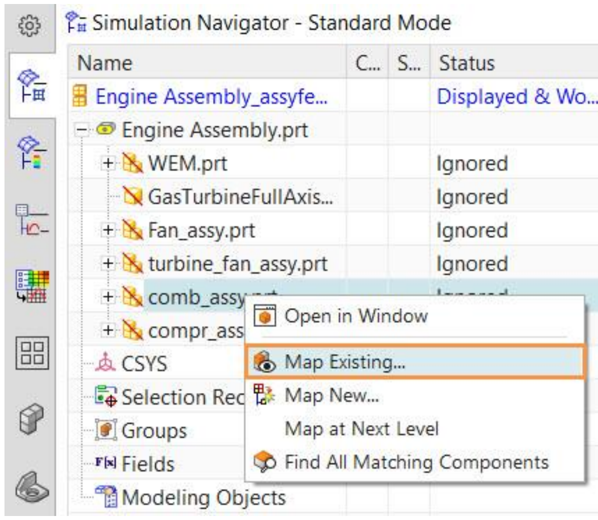
4. In the **New Assembly FEM** dialog box, select the **Simcenter 3D Multiphysics** solver environment, **Coupled Thermal-Structural** analysis.
5. Specify the **2D Solid Option** as the model part is axisymmetric in the ZX plane. Therefore, select **ZX Plane, Z Axis**.
6. Select the **Create Cyclic CSYS** check box to create a global cyclic analysis coordinate system and set it as the default coordinate system for boundary conditions. If **2D Solid Option** is set to **ZX Plane, Z Axis**—The Z-axis of the cylindrical coordinate system aligns with the global Z-axis, and the rotation for the cylindrical coordinate system aligns with the global X-axis.



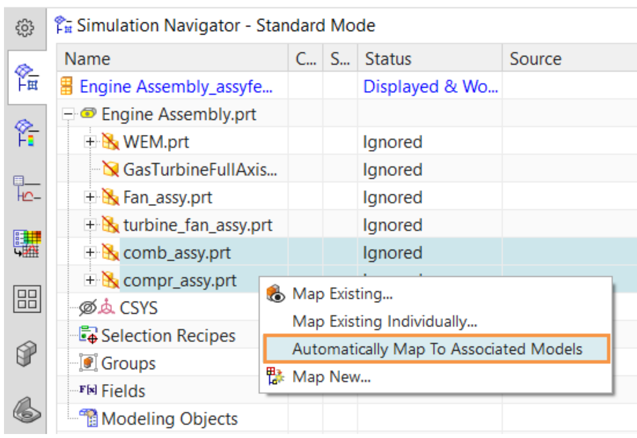
4. In the **Simulation Navigator**, you can view all the component parts comprising the assembly. Because the CAD components are not currently mapped to component FEMs, their status is set to **Ignored**, which means they will not be considered in the analysis. Because there are no FEM components in the new assembly FEM, the graphics window is empty. When you create an associative assembly FEM, the loaded assembly part is shown as a child of the assembly FEM node, and the component parts are shown as children of the assembly part node.



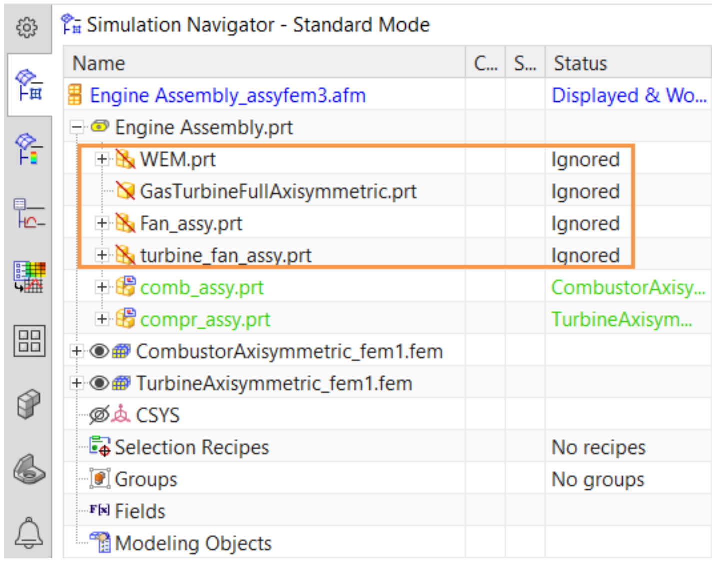
5. Import the separate FEM files to each of their respective places by right-clicking and selecting **Map Existing**.



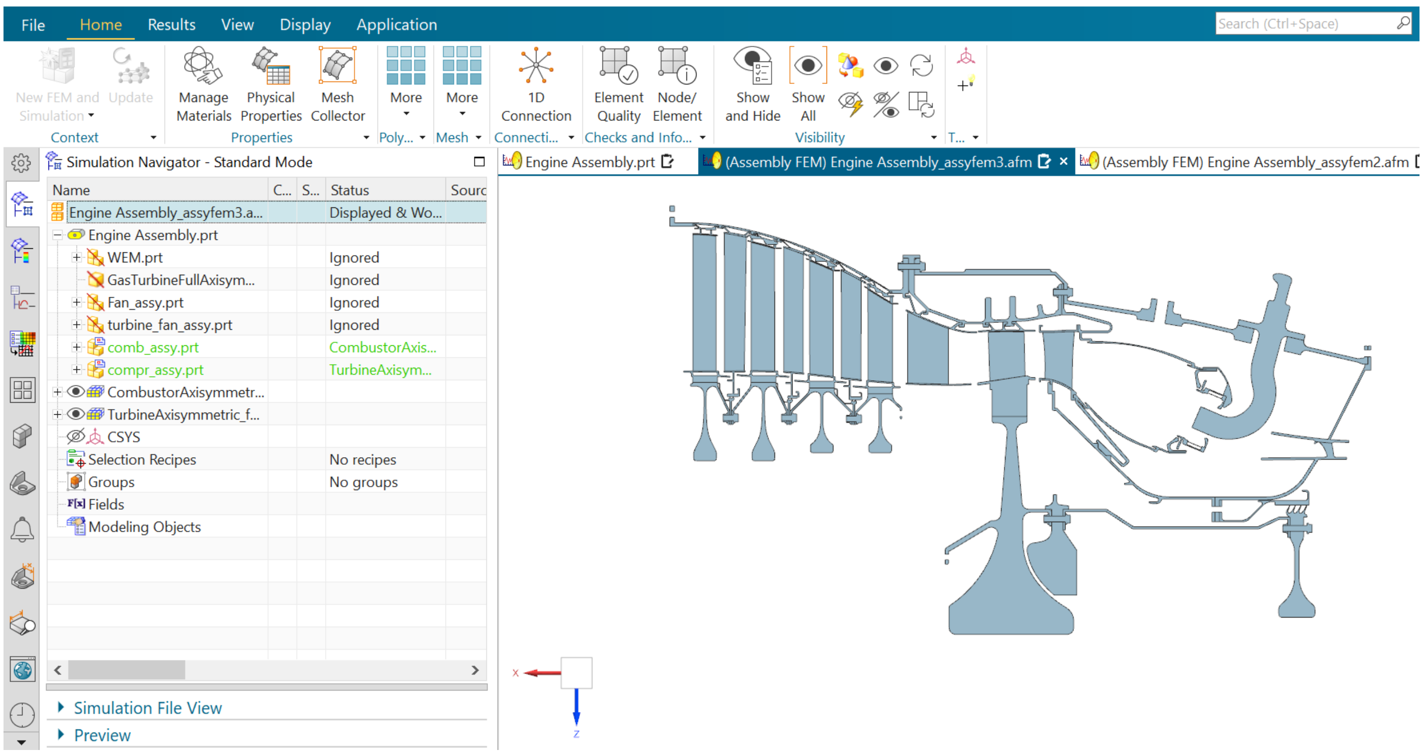
Or by selecting desired components, right-clicking and choosing **Automatically Map To Associated Models**.



The parts that were not mapped to the FEM will remain marked as **Ignored** and will not be included in the analysis.

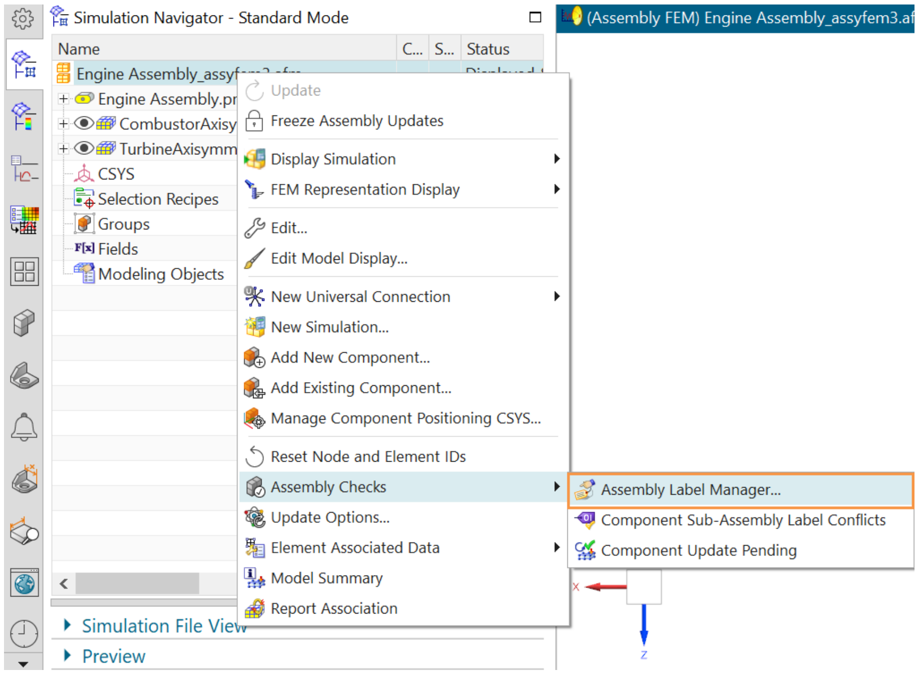


One large assembly FEM has now been created, but it is not yet ready to be used in an analysis.

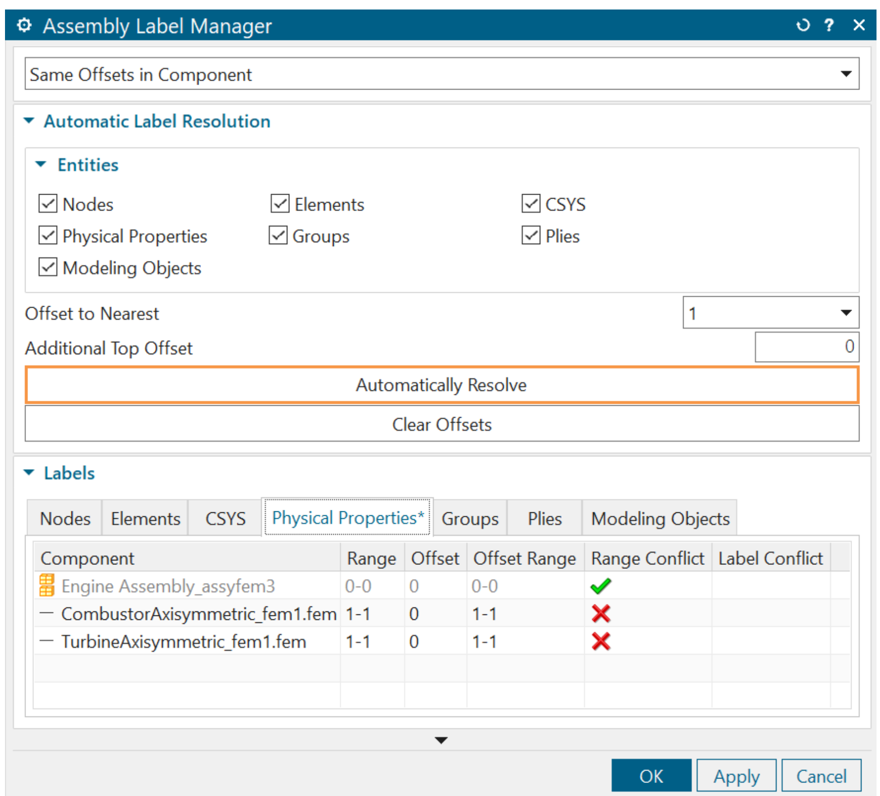


Note: If the CAD assembly on which the assembly is based is modified, the assembly FEM is out-of-date. You must update the AFEM by clicking **Update** in the **Home** tab→**Context** group.

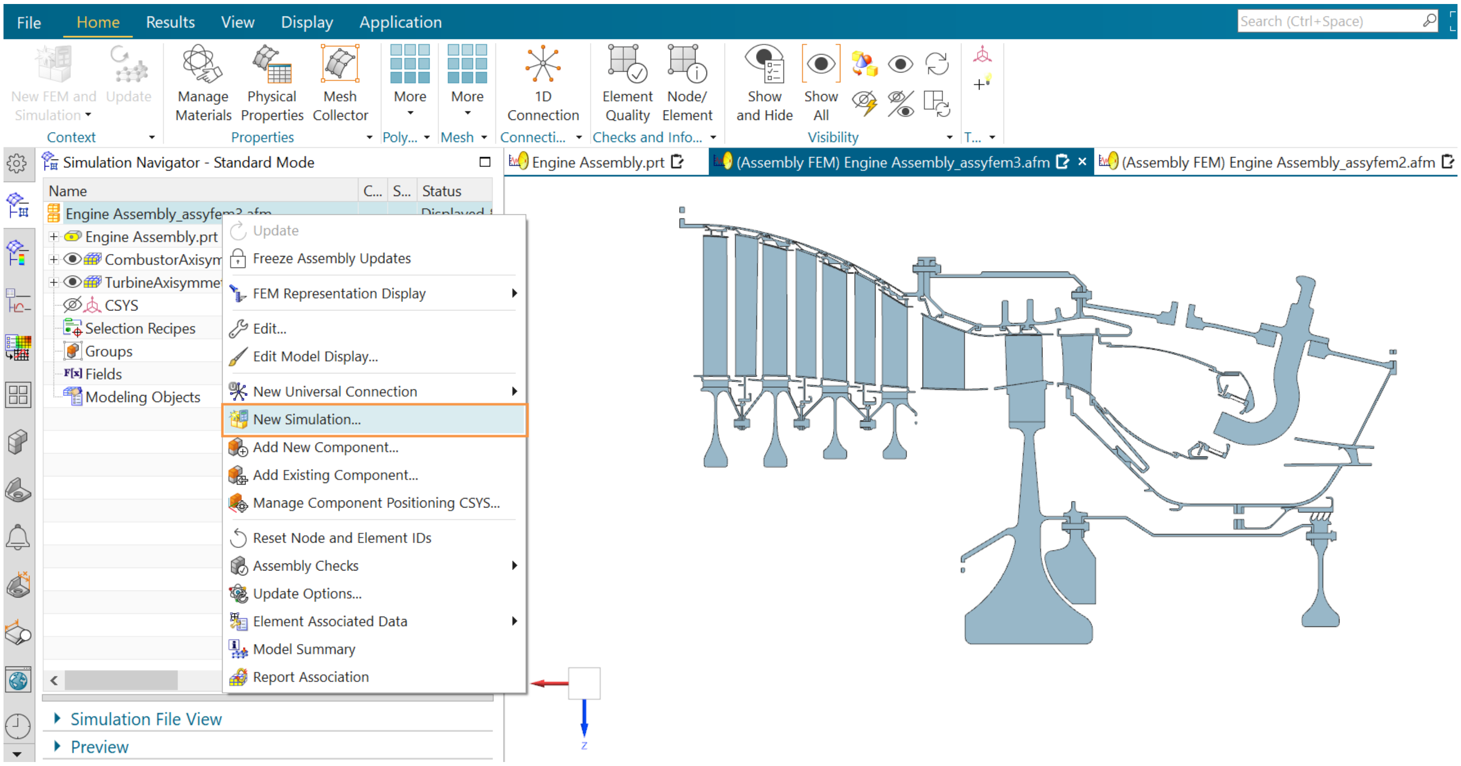
6. Resolve label conflicts by right-clicking the afm node in the **Simulation Navigator** and selecting **Assembly Checks→Assembly Label Manager**.



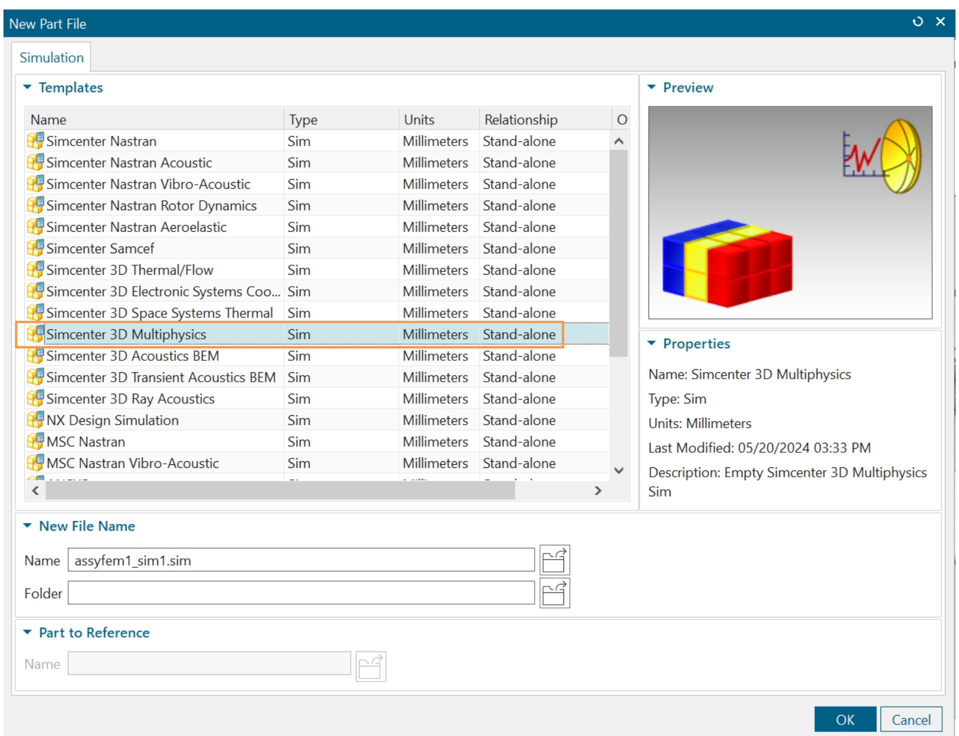
7. In the **Assembly Label Manager** dialog box, you can click **Automatically Resolve**, or you can manually modify the offset.



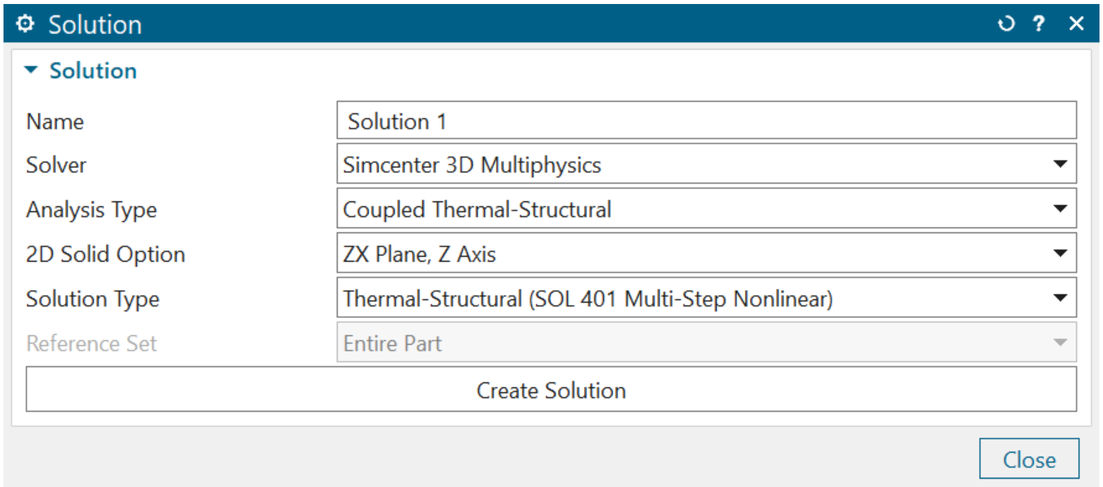
8. Create a simulation from the assembly fem by right-clicking the afm node in the **Simulation Navigator** and selecting **New Simulation**.



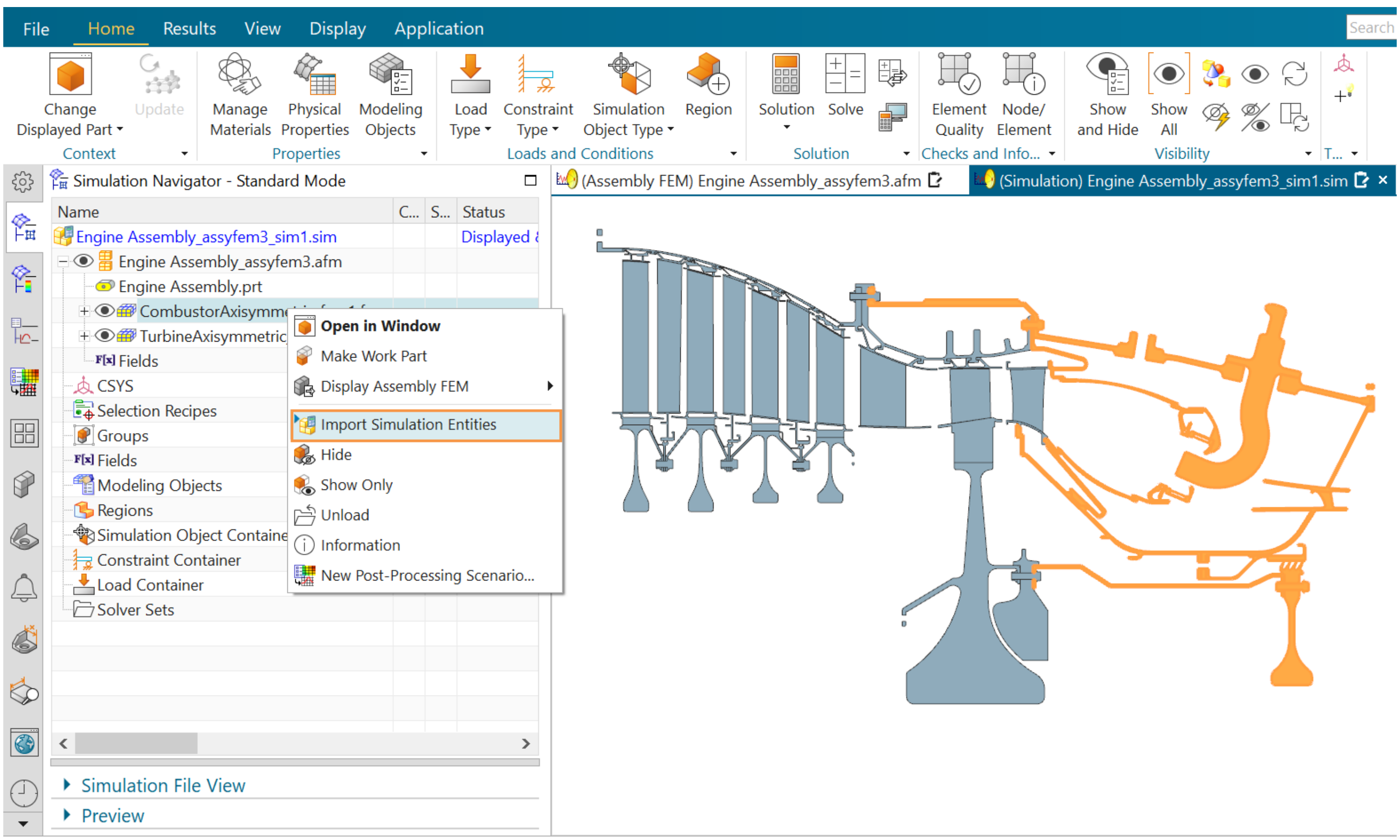
9. Select the **Simcenter 3D Multiphysics** solver environment.



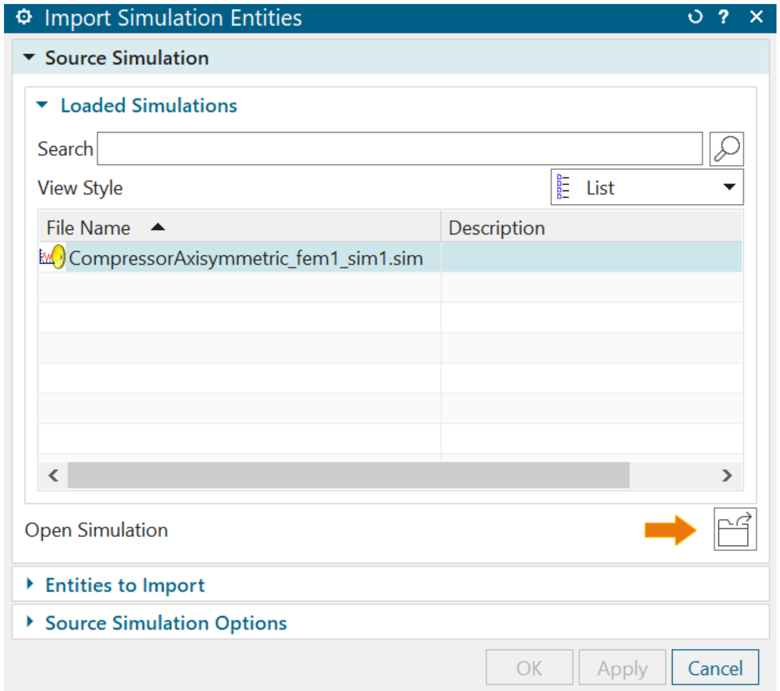
10. In the **Solution** dialog box, select the **Simcenter 3D Multiphysics** environment, **Coupled Thermal-Structural** analysis, and **ZX Plane, Z Axis** 2D solid option.



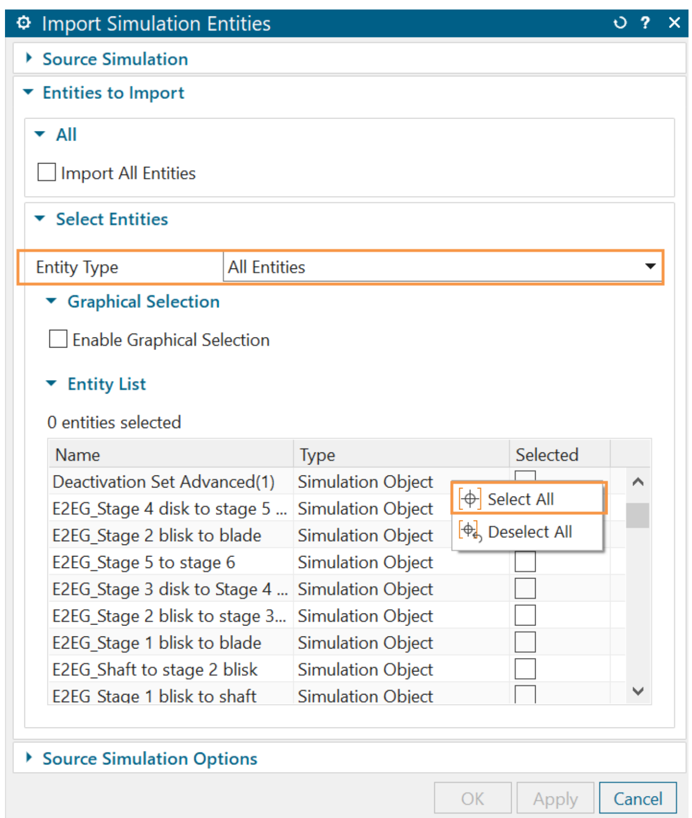
11. To add the previously created loads, constraints, and simulation objects of each fem, right-click the fem node in the **Simulation Navigator** and select **Import Simulation Entities**.



12. **Browse** to **Open Simulation** and select the corresponding simulation file.



13. From the **Entity Type** list, select an entity type to import into the target simulation file. In this example, **All Entities** are selected and in the **Entity List**, right-click and choose **Select All**.



14. Add simulation objects to connect the sub-assemblies, such as **Edge-to-Edge or Surface-to-Surface Gluing**, **Thermal Stream Junction**, and **Thermal Voids**.
15. Verify that all necessary simulation entities are present and the model is ready to be solved.

Non-Associative Assembly FEM Workflow

In a non-associative Assembly FEM workflow, the assembly FEM model is created independently of the CAD model. Changes in the CAD model do not automatically update the assembly FEM model. This workflow requires manual updates to the assembly FEM model whenever the CAD model changes.

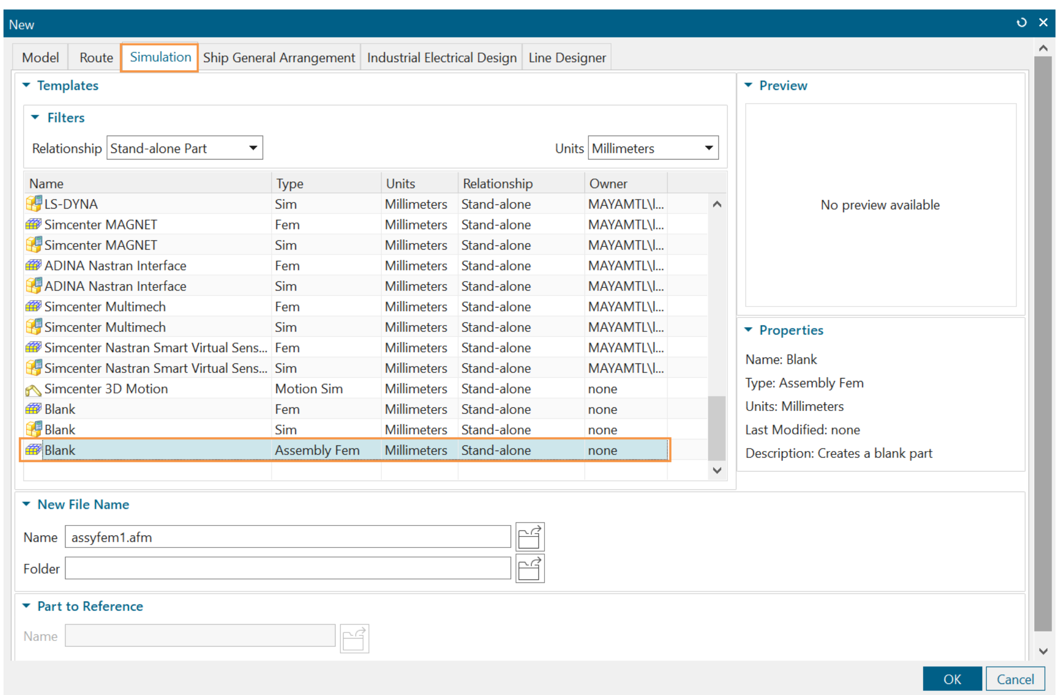
Use cases:

Recommended for: Projects where the design is relatively stable, and changes are infrequent. This workflow is suitable for later stages of design or for legacy models where the design is unlikely to change.

- **Turbomachinery:**
  Example: When performing a detailed analysis of a well-established compressor design, where the geometry is unlikely to change, the non-associative workflow can be more straightforward and efficient.
- **Space Systems Thermal:**
  Example: When analyzing the thermal performance of a well-established satellite component.
- **Electronics Systems Cooling**:
  Example: When performing a detailed analysis of a mature cooling system for a consumer electronics product.

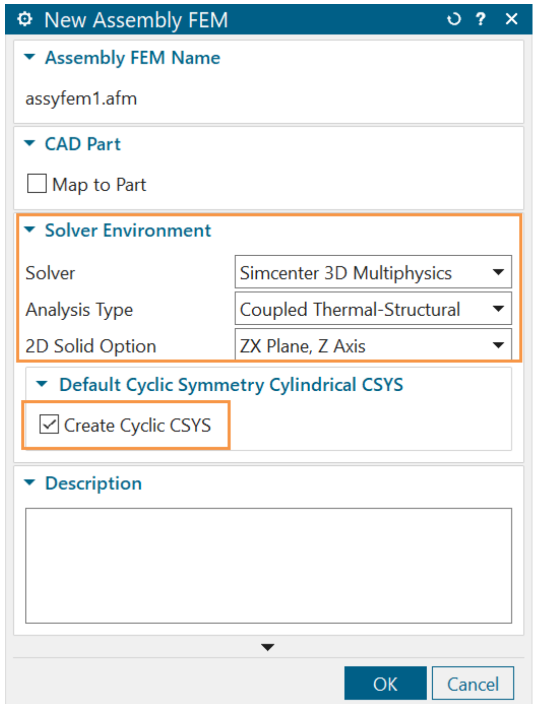
To create an non-associative assembly FEM:

1. Choose **File→New**.
2. Click the **Simulation** tab and select an Assembly Fem template.

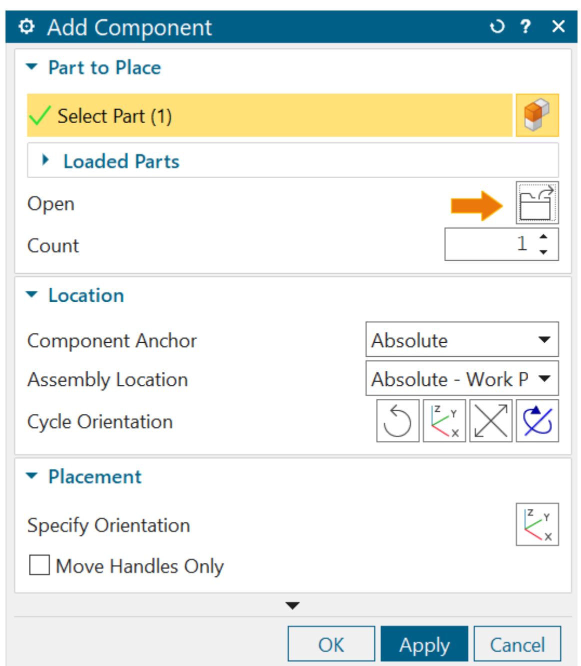


3. In the **New Assembly FEM** dialog box, select the **Simcenter 3D Multiphysics** solver environment and **Coupled Thermal-Structural** analysis.

4. In the **2D Solid Option** list, choose the plane on which you can create axisymmetric, plane strain, and plane stress elements. For example, **ZX Plane, Z Axis**.



5. Select the fem files that you want to add to assembly fem by right-clicking the afm node in the **Simulation Navigator** and selecting **Add Existing Component**.



6. Define component position and orientation.
   If any changes are made in the CAD assembly, make sure that the AFEM is updated to reflect those changes.
7. Perform the steps from 6 to 16 as in the associative workflow.

   By understanding the differences between these workflows and following the detailed procedures provided, engineers can effectively choose the workflow that best fits their project's requirements.