

A background graphic consisting of a stream of blue particles of varying sizes, moving from the top center towards the right side of the frame. The particles are more densely packed in the center of the stream and become more sparse towards the edges.

Model checks in Space Systems Thermal

Why do model checks?

Model checks are a set of verification processes used to ensure the accuracy, quality, and reliability of a simulation model.

You do model checks to:

- Detect errors and inconsistencies in the simulation setup.
- Reveal incorrect simplifications.
- Check that the boundary conditions are correctly defined and represent the real-world conditions accurately.
- Identify issues such as mesh distortion, element quality, or excessive skewness.
- Verify the material and optical properties and their consistency.
- Check geometry errors, such as overlapping surfaces or gaps.
- Identify areas where improvements can be made.
- Diagnose the thermal model results to ensure they are reasonable and align with expectations.

Model checklist

- ✓ Check units
- ✓ Check material and optical properties
- ✓ Verify elements quality and mesh
- ✓ Verify mesh normals
- ✓ Check for duplicate nodes
- ✓ Do model mass check
- ✓ Verify the element thickness
- ✓ Display all thermal couplings
- ✓ Check all warning messages for thermal couplings and consistent units
- ✓ Check radiation enclosures
- ✓ Check MLI set up
- ✓ Check heaters and heat loads, verify consistent units
- ✓ Resolve ID conflicts
- ✓ View thermal results on model: look for unexpected sharp gradients or unreasonable values, including on insulation.
- ✓ View absorbed solar/planetary flux on model surfaces
- ✓ Check orbits for valid parameters
- ✓ Check log file for errors and warnings
- ✓ Run model setup check
- ✓ Use the traceback patch

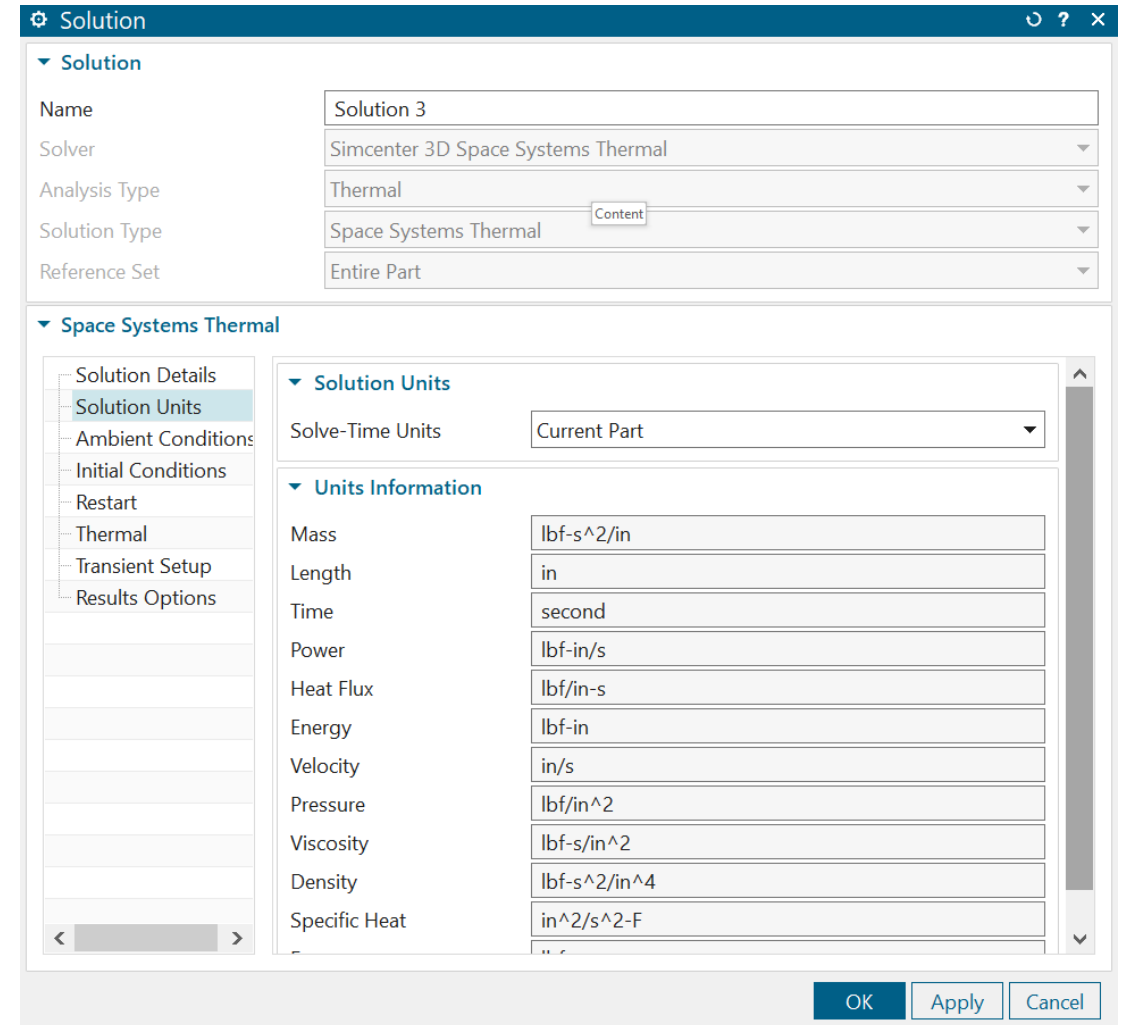
Most common issues

- Units
- Material and optical property assignments
- Poor mesh quality, inadequate resolution near the blades, or insufficient refinement in boundary layers
- Mesh distortion, skewness, and element quality
- Improper boundary conditions
- Thermal coupling setup
- Radiation setup

Verifying units

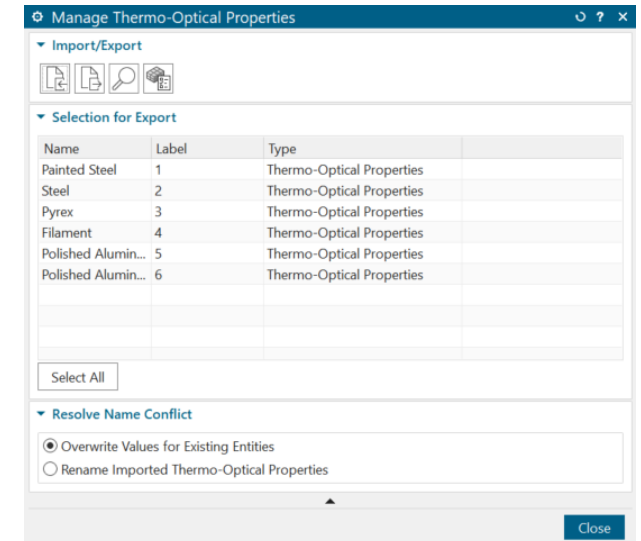
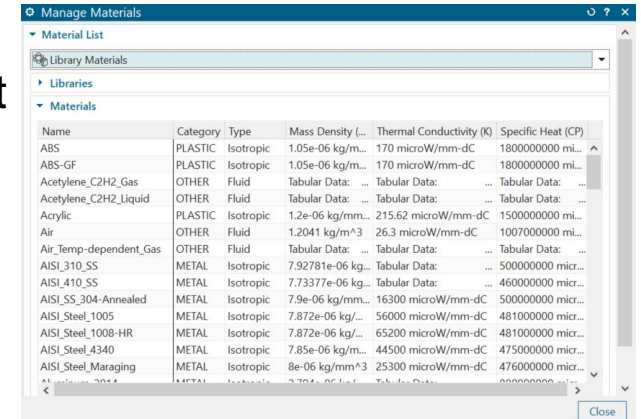
Solution units are controlled in the **Solution Units** group.

- Check user logic (expressions) for problems that could arise from changing units.
- Make units consistent.
- Do one of the following, since not all units are case insensitive.
 - Use the auto-complete suggestion for the unit.
 - Use the case for the unit as shown in the **Units Manager** dialog box.



Reviewing materials

- Check the **Material Library** source and make sure that thermal properties such as thermal conductivity, specific heat and density are defined if required (i.e. not MLI).
- Add columns in the **Manage Materials** dialog box for quicker visualization of important thermal properties such as ρ , k , and C_p by right-clicking any column heading and selecting **Columns**→**Configure**.
- Check overrides for density and conductivity on surfaces and solids
- Use the **Material Information** command to inspect the material properties of the selected elements.
- Create and maintain a central repository of thermo-optical properties to reduce data entry errors, using the **Manage Thermo-Optical Properties** command. You can export and import these properties using an xml file.



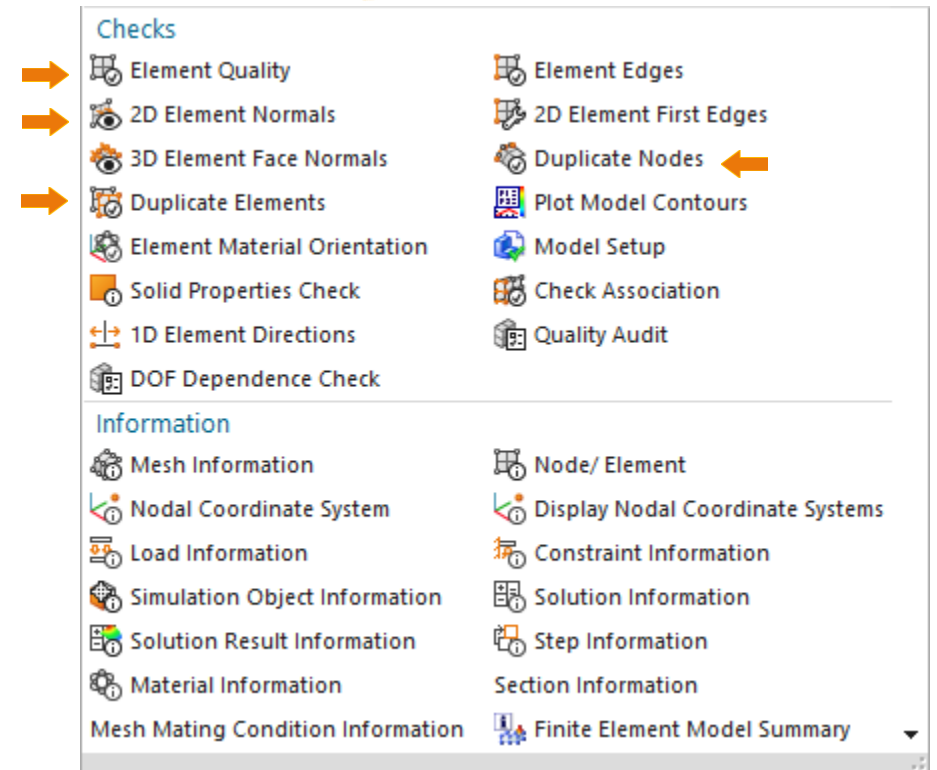
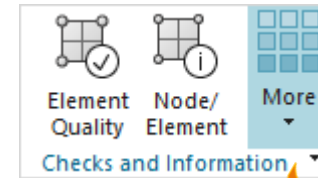
Verifying mesh density

- Check visually for appropriate mesh density.
 - Are there enough elements to capture temperature gradients? A common mistake is creating meshes that are too fine.
- Perform a mesh sensitivity study to assess mesh adequacy. However, this may not always be practical due to resource constraints, such as computing resources or time limitations.
- Follow the guidelines:
 - Start your analysis with a coarse mesh to evaluate a first approximate set of results.
 - Create finer meshes in areas where temperature variations are largest and in areas of specific interest.
 - Minimize any distortions by improving or recreating your mesh.
 - Use **Mesh Controls** options to control the mesh density in specific areas. It helps improve quality issues.
 - Avoid having multiple highly distorted and stretched elements in one area of your model.

Using finite element model checks

Use the finite element model check commands to:

- Check how well the model's CAE geometry conforms to the underlying CAD geometry.
- Ensure the quality and consistency of your mesh.
- Validate that the model is complete and ready to solve.



Checking and orienting the element normals

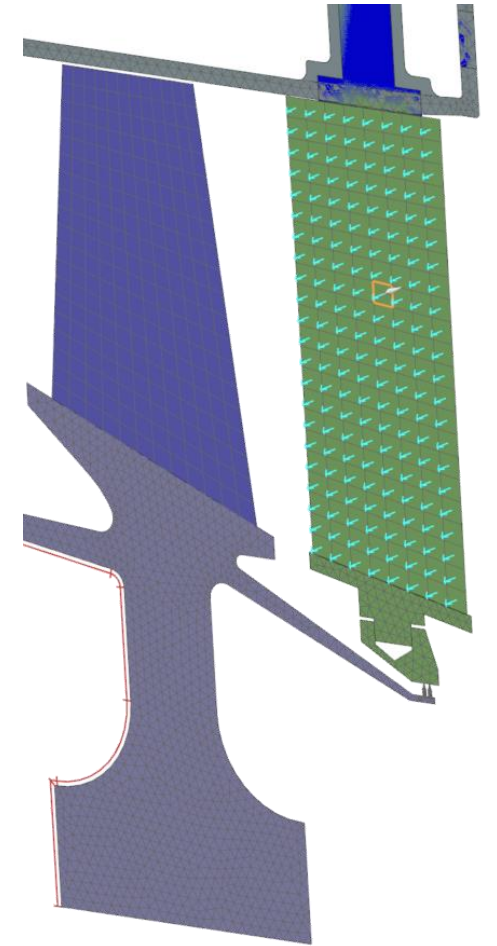
Use the **Element Normals** command to:

- Display and reverse the normals of the elements.
- Create a group of inconsistent elements.
- Automatically align the normals of a selected set of elements.

All 2D elements have a normal that establishes their top and bottom. Consistent element normals help ensure the overall quality of your FE model.

For example, consistent normals are important for:

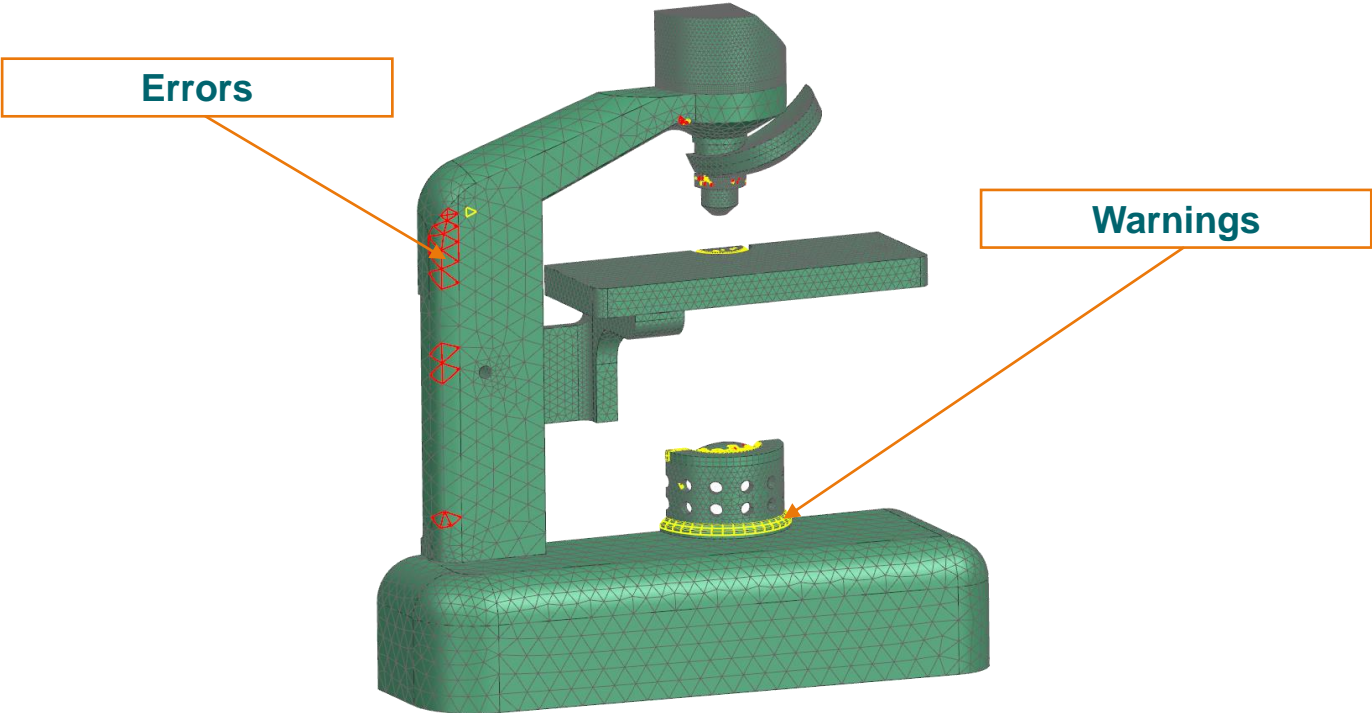
- Properly defining contact between surfaces.
- Properly defining top and bottom in the radiation request.



Evaluating element quality

Use the **Element Quality** command to perform element quality check with following values for the thermal solver.

	Interior angle, α	Warp angle, β (quads only)
Thermal elements	$1^\circ < \alpha < 179^\circ$	$\beta < 15^\circ$



Element Quality [?] [X]

- ▼ Elements to Check
 - Selected
 - * Select Object (0) [Select] [...]
 - ▶ Element Labels
- ▶ Limits Check
- ▶ General Geometry Checks
- ▼ Solver Specific Geometry Checks
 - Use Element Type Specific Values [Info]
 - Select All [Deselect All]
 - Error Limit
 - Jacobian Ratio > 30
 - Jacobian Zero <= 0
 - Aspect Ratio > 20
 - Skew Angle > 30 °
 - Maximum Interior Angle > 179 °
 - Minimum Interior Angle < 1 °
 - Taper < 0.5
 - Warp > 15 °
- ▶ Display Settings
- ▶ Output Settings

Check Elements [Correct Failed Elements] [Close]

Identifying coincident nodes

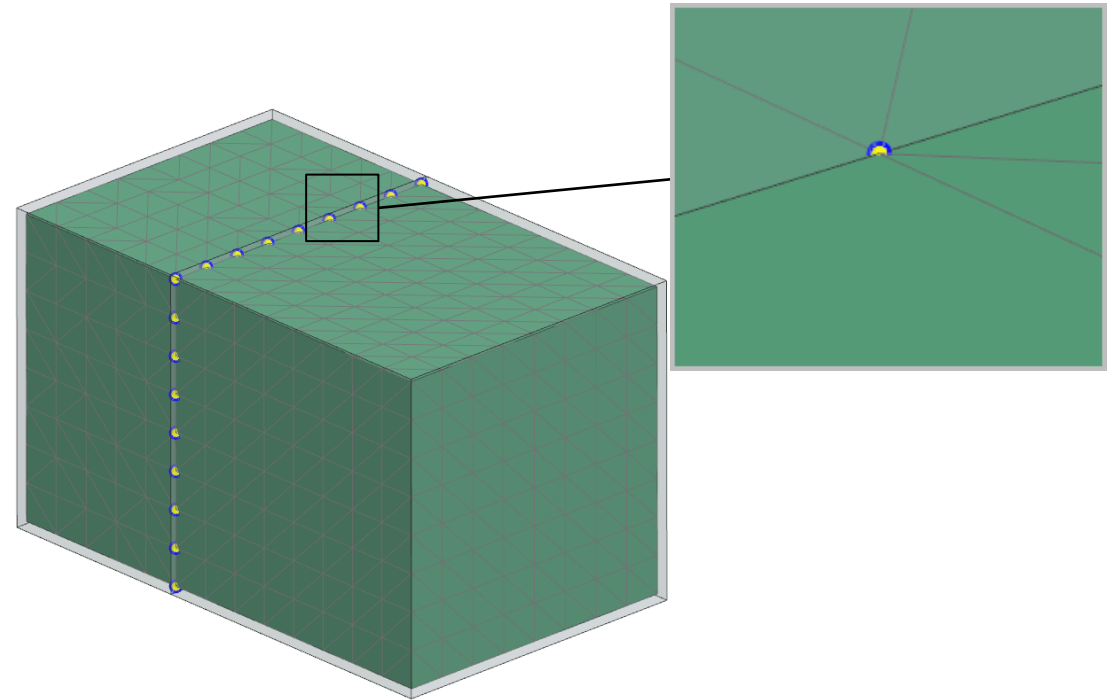
Check for coincident nodes which are duplicate nodes lying on top of each other.

If you try to solve a model that contains coincident nodes, singularities or other rigid body motion errors can occur during the solution.

Modeling conduction requires you to create meshes with shared nodes to preserve continuity.

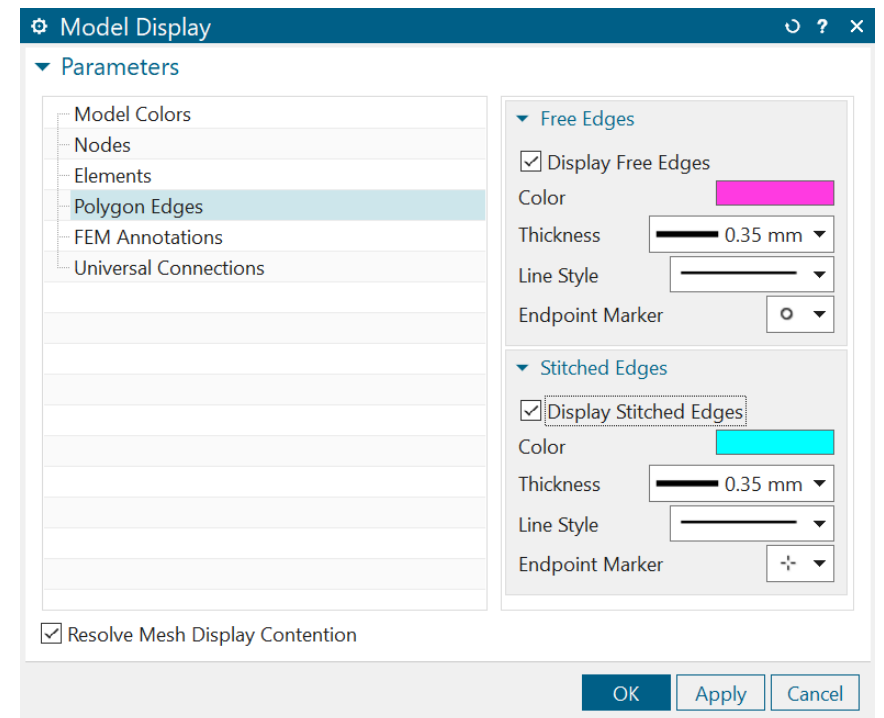
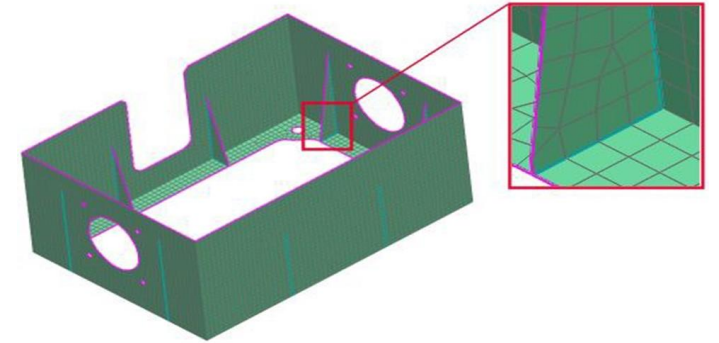
To avoid, check, or resolve duplicate node issues, use the **Mesh Mating Conditions** or **Duplicate Nodes** commands.

Visual representation of coincident nodes



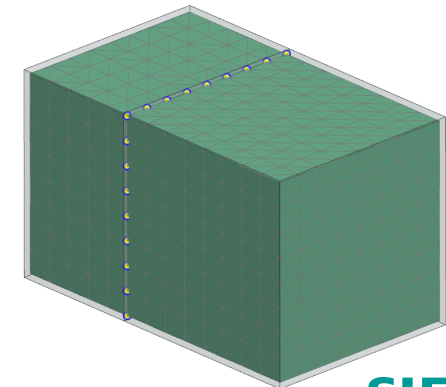
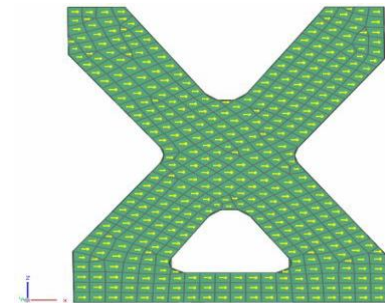
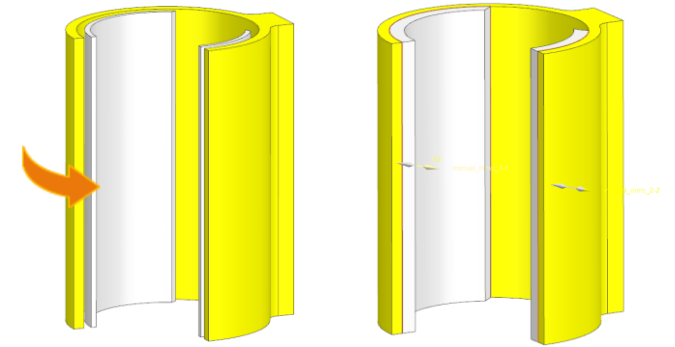
Conducting geometry checks

- Check for free (unconnected) element edges within a 2D mesh using the **Element Edges** command. A *free* element edge is an edge that is referenced by only one element.
- If you have problem edges, use the **Stitch Edge** command to stitch problem edges either automatically or manually.
- If there is a large number of problem edges or if the part fails to mesh, you may need to repair the underlying master part geometry in **Modeling**.
- If there is a small number of localized problem edges, use manual node and element operations to directly repair the problem areas.
- Experiment with increasing the tolerances used by the meshing algorithms. Note: Excessively large tolerances may cause unpredictable results in other areas of the model.
- In the **Model Display** dialog box, select the **Display Free Edges** option to highlight all free edges in your model to identify edges that need to be stitched prior to meshing.



Conducting geometry checks

- Use the **Mesh Mating** command to:
 - Modify polygon body geometry so that surfaces share a common definition.
 - Enforce common surface meshes where polygon bodies mate.
- Display the material orientation of 2D or 3D elements in your model using the **Element Material Orientation** command.
- Check for duplicate bodies or faces.

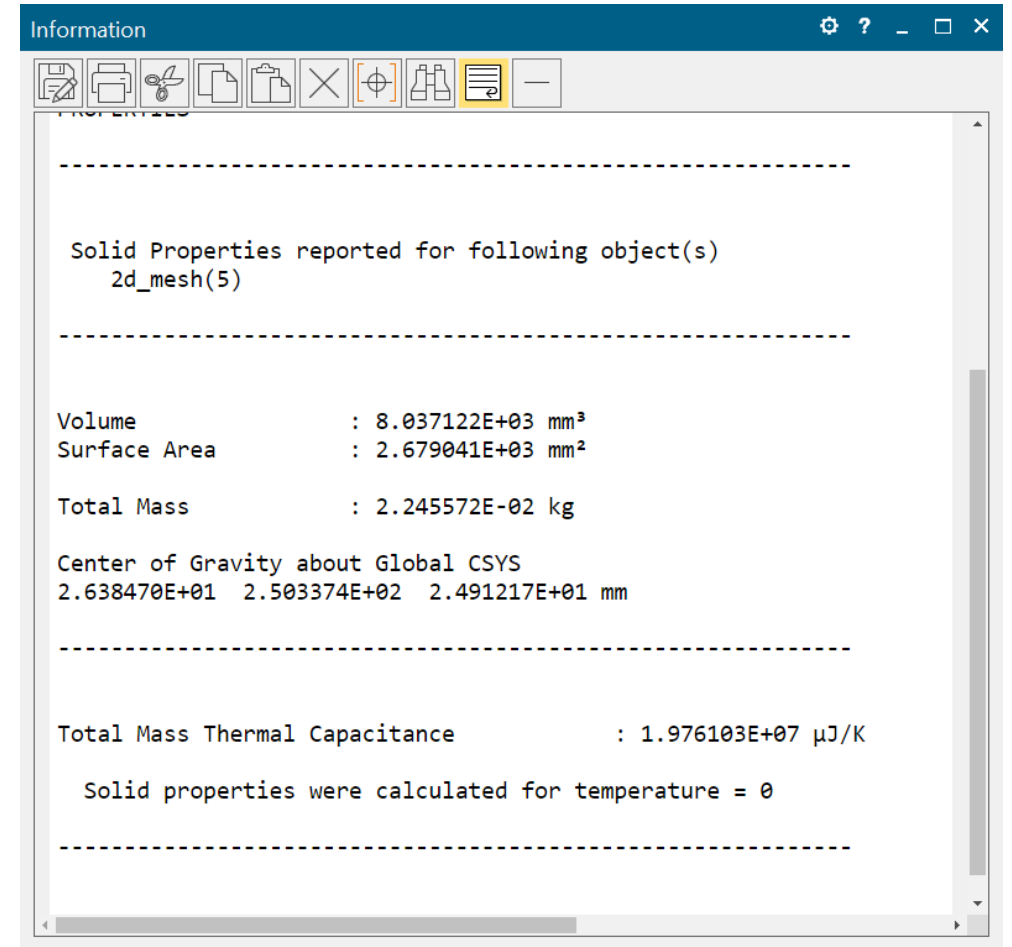


Assessing mass properties

- Perform a model mass check using the **Solid Properties Check** command to compute the surface area for convection and radiation, and the thermal capacitance, which is the model's mass multiplied by its specific heat.
- Ensure that materials with very low thermal capacitance, such as MLI, do not have mass assigned. This can cause convergence issues at solve time.
- Inspect the [Solution_name]_report.log file that contains calculation details, model parameters, elements that thermal solver created, and results summary of groups.

Temperature summary for groups

	Maximum Temp	at element	Minimum Temp	at element	Average Temp	Total Heat in	Total Capacitance	Total Mass
Group: Bot-Rad-Bus-End	20.00	3225	20.00	3225	20.00	0.00E+00	0.00E+00	0.00E+00
Group: Bot-Rad-Apayload-Enc	20.00	8236	20.00	8236	20.00	0.00E+00	0.00E+00	0.00E+00
Group: Bot-Rad-Ext-Enc	100.00	9325	-0.00	5235	7.72	0.00E+00	1.33E+11	1.48E+02



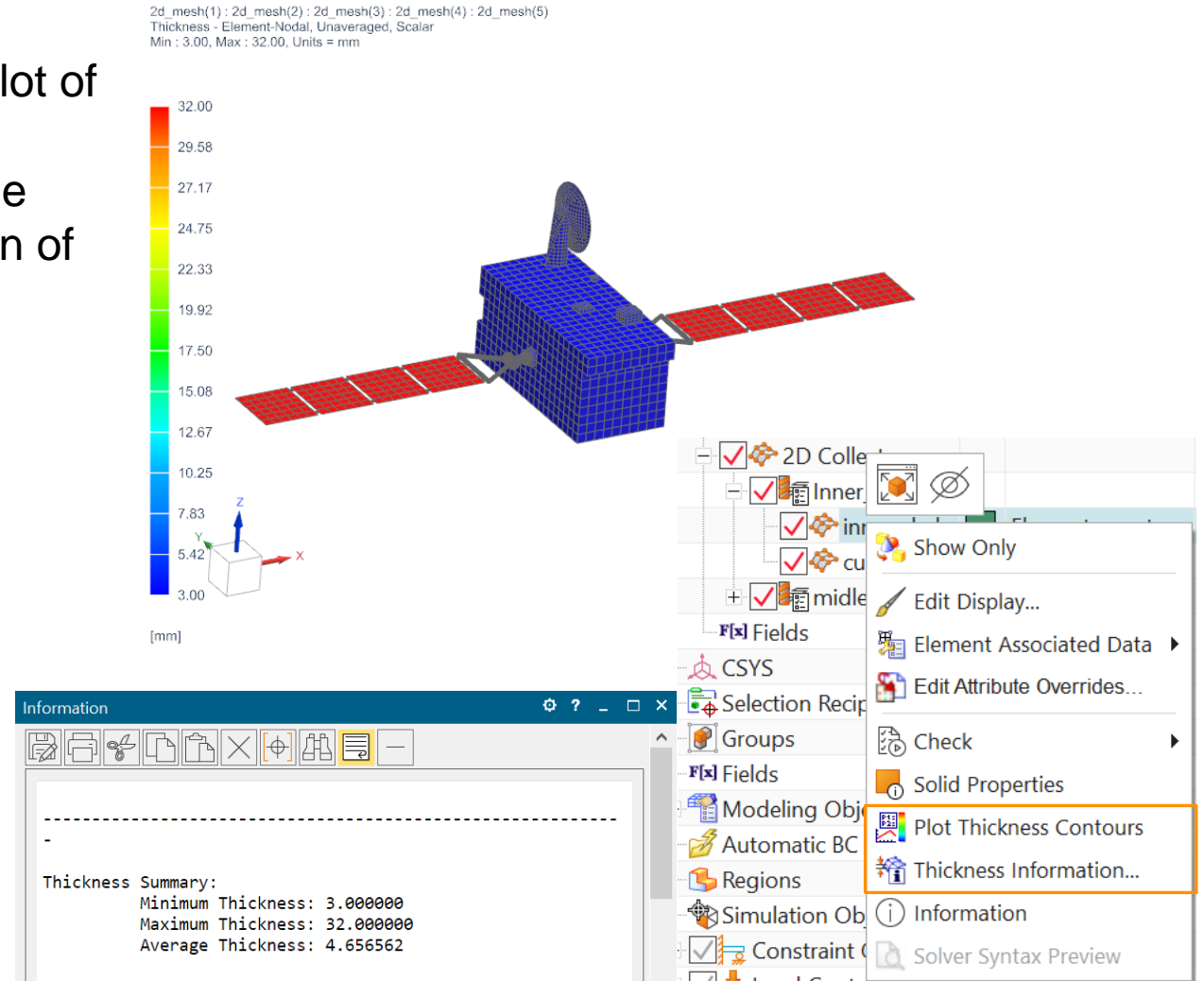
Verifying element thicknesses

You can use:

- **Plot Thickness Contours** to generate a contour plot of shell element thicknesses as a standard post view.
- **Thickness Information** to create a color-coded line display that shows the general statistical distribution of the thickness values across your 2D mesh.

You can use the thickness display to quickly identify:

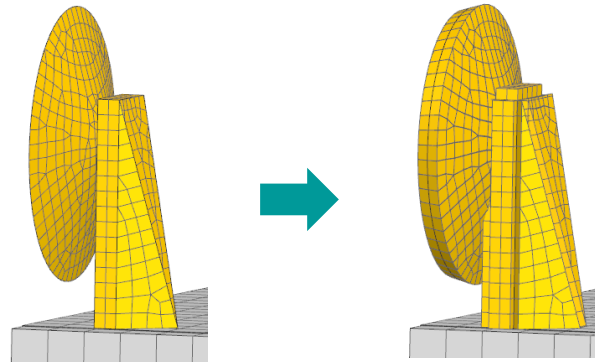
- Any sudden changes in color that may indicate incorrectly assigned thickness values.
- Elements that do not have an assigned thickness.



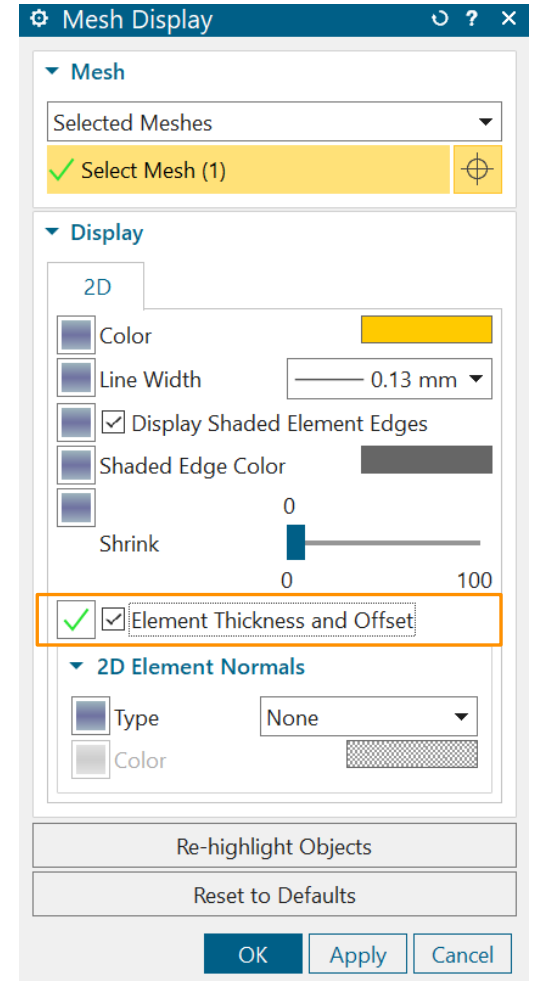
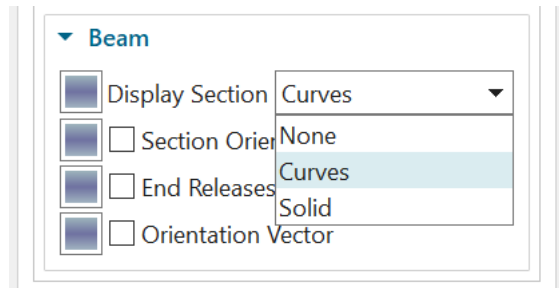
Verifying element thicknesses

You can use the **Edit Display** command (right click on mesh) to:

- View a graphical representation of the thickness and shell offset of the 2D elements.

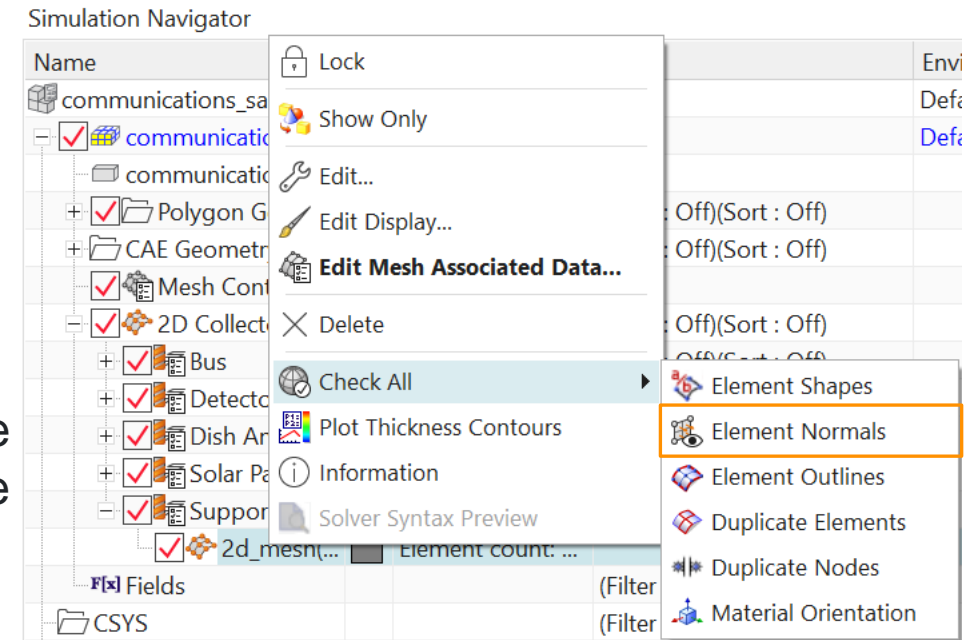


- Display beam cross sections by selecting **Curves** or **Solid** in **Display Section**.



Reviewing optical properties


- When you define thermo-optical properties for 2D meshes, you should always check the element normals using the **Element Normals** command to determine which is the top side of the mesh and which is the bottom side.
- When you specify different transmissivity values for their respective top and bottom thermo-optical modeling objects of the same mesh collector, only the transmissivity value of the top side modeling object is used. This is because the transmissivity is a property of the material, not a property of the surface, as for example, emissivity.
- Inspect the optical properties by right-clicking the mesh node and selecting **Information**.

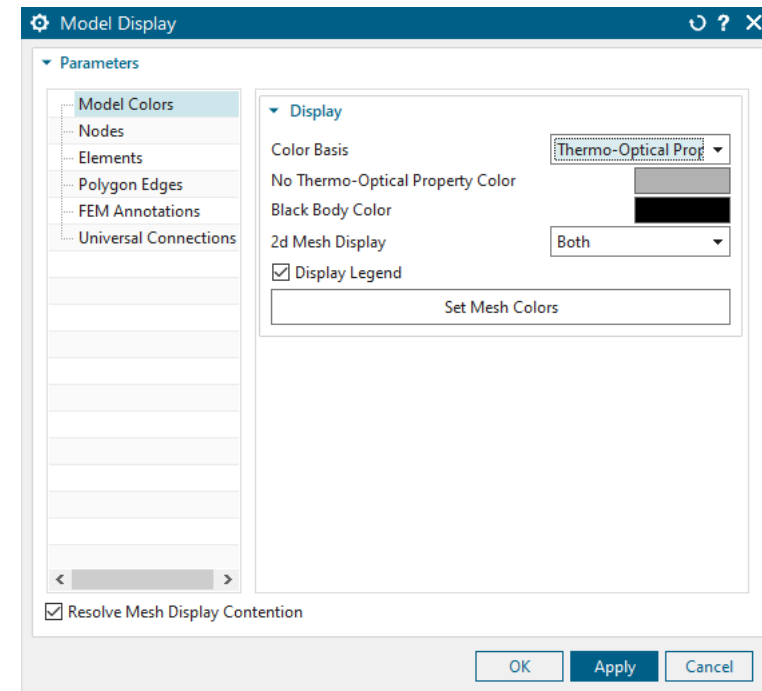
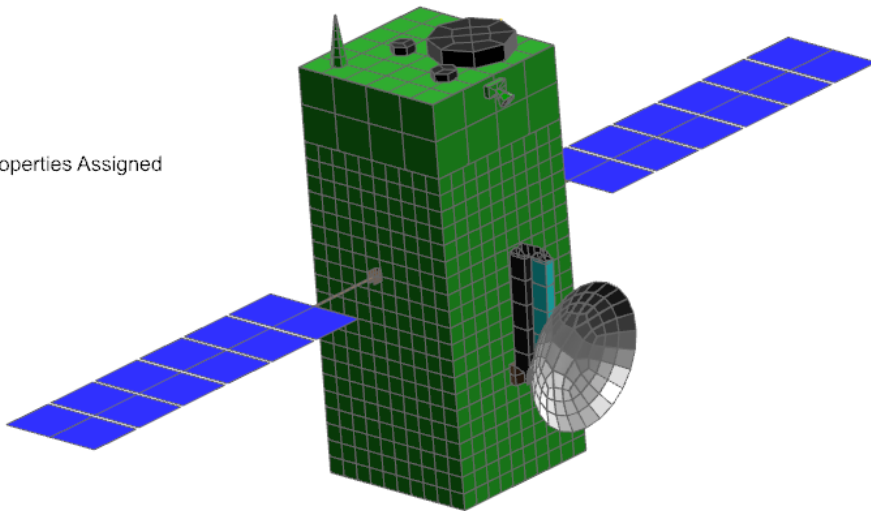


Displaying thermo-optical colors

- Set the model display preference to color-code the meshes based on their thermo-optical properties, making it easier to verify the model setup for thermo-optical properties.
- Choose **Menu**→**Preference**→**Model Display**.
- Verify that no mesh has the No Thermo-Optical Properties Assigned color

Thermo-Optical Properties

	Aluminized Kapton - EOL
	Aluminized Kapton - States
	Bare Alu - States
	Black Body
	No Thermo-Optical Properties Assigned
	Silver Teflon - States
	Solar Cells - States
	White Paint - States

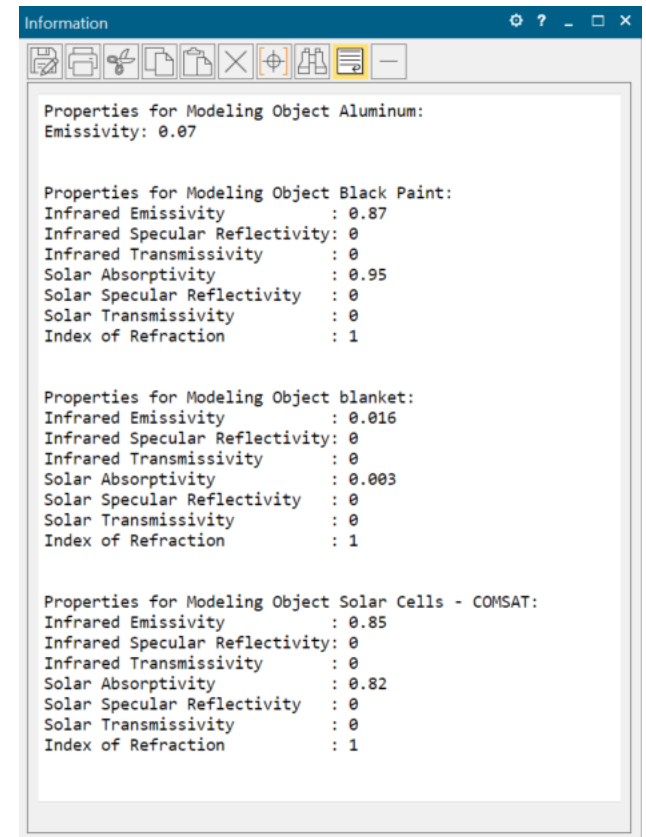
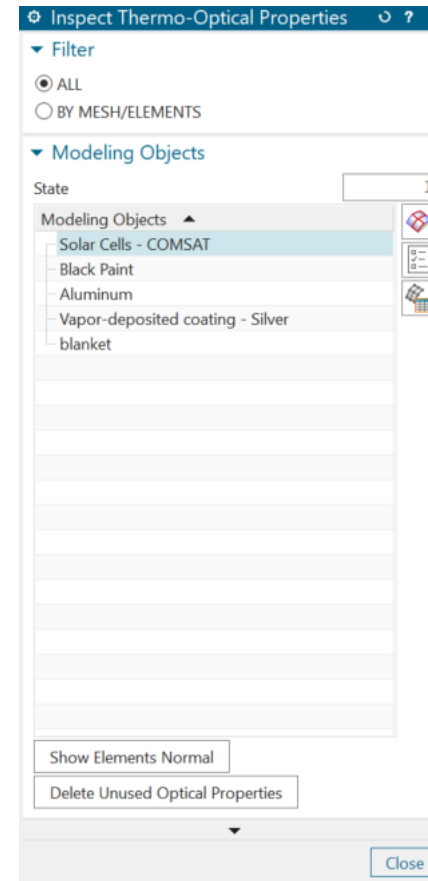


Inspecting thermo-optical properties

Use the **Inspect Thermo-Optical Properties** NX open command to easily determine the thermo-optical properties defined for different meshes in your model.

You can:

- List thermo-optical properties modeling objects for selected meshes or elements.
- Display the meshes associated to one or more thermo-optical properties modeling objects.
- Display element normals for all visible meshes as you are inspecting the thermo-optical properties.
- List thermo-optical properties for selected modeling object.



Set UGII_USER_DIR to <software installation>\nxcae_extras\tmg\customization\OpticalProperties_Tool folder before starting the software.

Inspecting thermo-optical properties

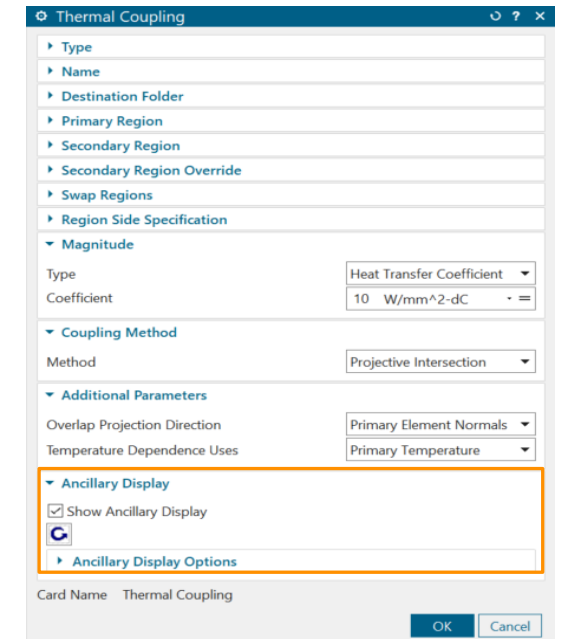
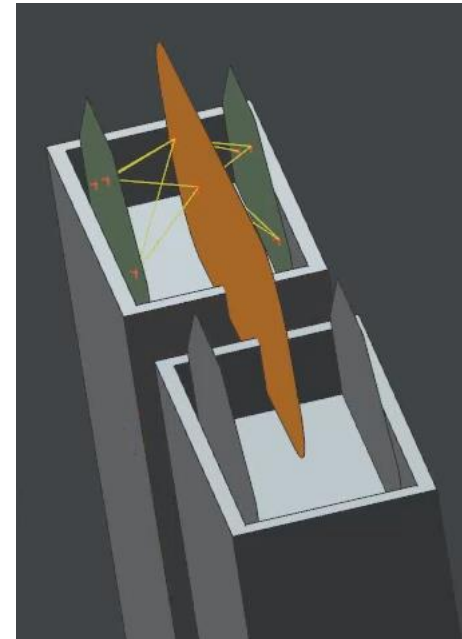
- Check that the thermo optical properties epsilon and alpha have been assigned properly, using:
 - The **Inspect Thermo-Optical Properties** command.
 - Contour incident and absorbed radiation on the analysis.

Verifying expressions

- Check expression logic and units.
- Specify a customer default so that the software issues a warning message about inconsistent units within mathematical functions.
File → **Utilities** → **Customer Defaults** → **Pre/Post** → **Expressions** → **General** tab, select the **Warn about Inconsistent Units within Mathematical Functions** check box.
- Use the expressions to set parameter values for the whole analysis. You can share these parameters in different simulation and FEM files.
- Expressions can be accessed in a tabular format by pressing Ctrl+E. You can update these expressions from an external file or linked to Excel.
- You can also use **Parameter Tables** to manage multiple expressions at once.

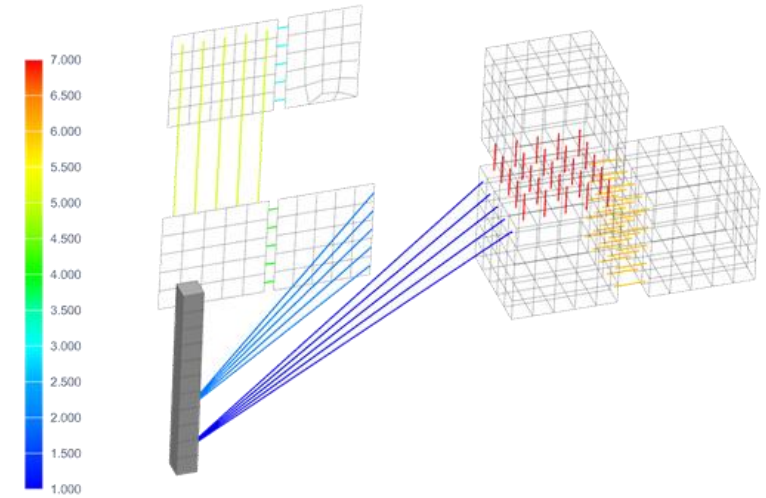
Verifying thermal couplings

- Verify that the thermal coupling setup in a physically meaningful way.
- Verify the selection region:
 - Select the smaller segment as a primary region.
 - Select the coarse mesh as a primary and fine mesh as a secondary region.
 - Note that the primary element selection does not control the direction of heat flow.
- Check thermal coupling values.
- Visualize thermal connections using the **Ancillary Display** option when using the projective intersection coupling method before solving the model.



Verifying thermal couplings

- Investigate if the primary and secondary elements are correctly connected thermally using the **Thermal Connection** result sets.
- Run conduction only simulation.
- Remove all convective and radiative boundary conditions and loads, while leaving thermal contacts and joints in the model. Apply constraints on either end of the model. This enables users to confirm thermal contacts are appropriately modeled, and can also be a check for specific heat if run transiently.
- Check warning messages.
- Use the **Report** command to investigate heat flow between components in the assembly, as well as from each component to the environment. This helps to understand where there is a large amount of heat, and where we could benefit the most by using thermal tapes or thermal standoffs in the design. The data from reports is generated in both .html and .csv formats.
- Verify and determine individual conductances of elements in a thermal coupling by inspecting scratch files. Use the **FILES MODLCF, VUFF, MODLF IN ASCII** advanced parameter to write intermediate files in ascii format.



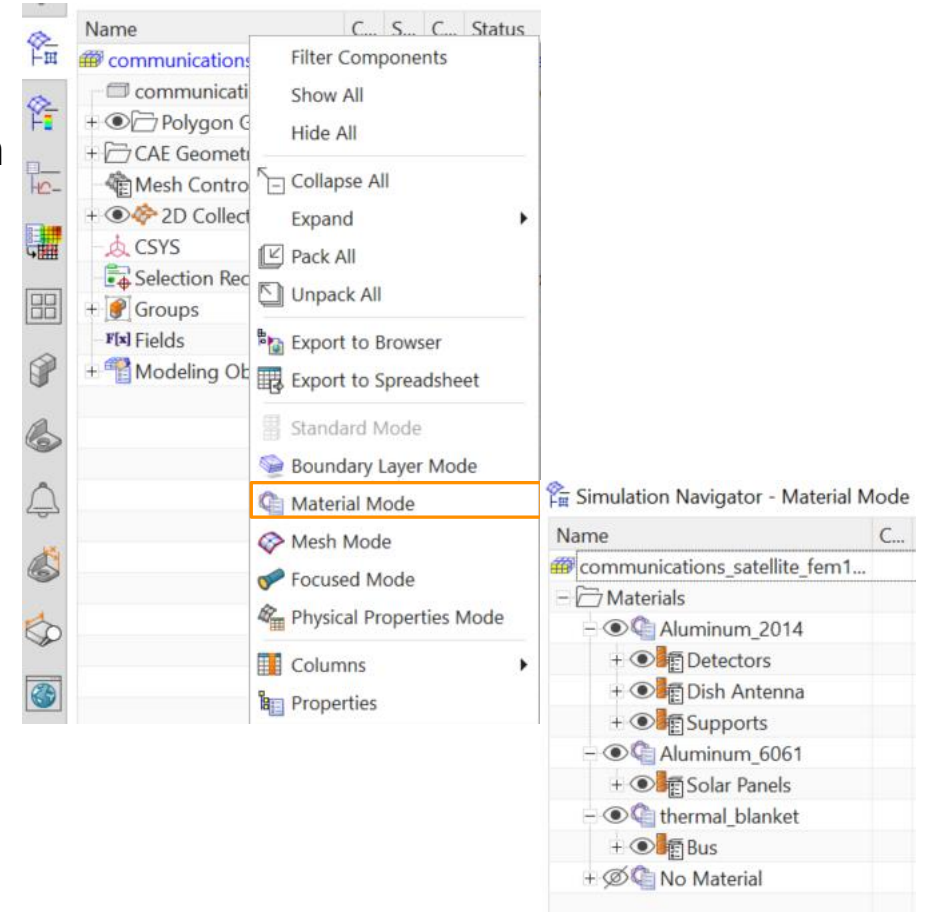
Thermal Connections –
Elemental result set

Checking radiation enclosures

- Check external and internal enclosures.
- Run radiation only test.
- Check the **View Factors Sum** result set. In an enclosure, the sum of any element's view factors should be equal to 1. You can control the precision of this calculation with the options in the **Radiation** dialog box.
- Try increasing radiation calculation accuracy (element subdivision, hemicube rendering, number of rays) and see if it impacts temperatures.

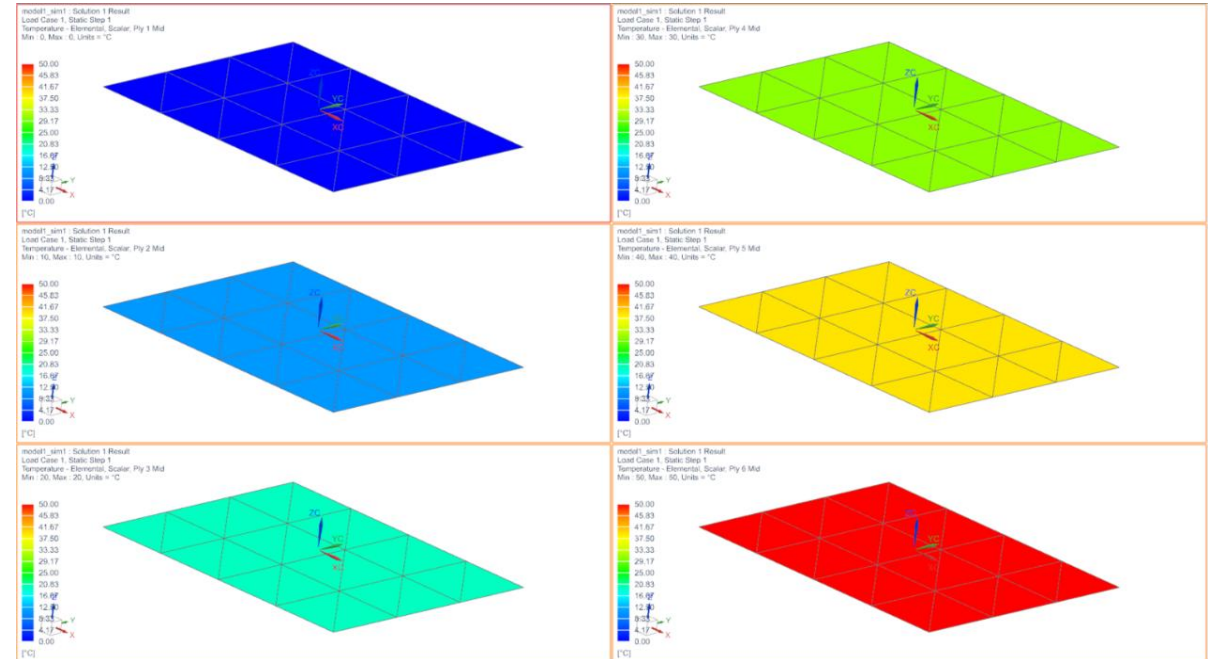
Verifying multi-layer insulation

- Check MLI setup:
 - Ensure your normal directions are correct.
 - Check if the MLI shells properties are setup correctly in the mesh collectors.
 - Show where thermal-optical properties are applied using the **Inspect Thermo-Optical Properties** command or the **Thermo-Optical Color Display**.
 - Show where material properties are applied by right-clicking the top of the navigator and selecting **Material Mode**. Make sure to open the FEM file in the window.
- Check the article on how to model MLI in Space Systems Thermal: [Multi-layer insulation modeling in Simcenter 3D Thermal](https://support.sw.siemens.com/knowledge-base/KB000122047_EN_US) (https://support.sw.siemens.com/knowledge-base/KB000122047_EN_US)



Visualizing multi-layer insulation

To visualize the thermal gradients of the MLI, set multiple viewports, and display each layer separately, as shown.



Checking heaters and heat loads

- Check heater performance in the *ThermostatReport.csv* and *[Solution_name]_report.log* file to ensure they are working as expected.
- Check the INPF file for the name of the thermostat that is used in UI. The *ThermostatReport.csv* file contains the short name.

INPF

```

THERMST Active_0 5.300000E+01 0.000000E+00 6 ID PID 1.000000E+01 1.000000E+00
NAME2 diode_h diode heat load 5W(1)
NAME diode_h 67161 67184 1
TABTYPE 5 TABLE OPERATION
TABDATA 5 7 INTERP
TABDATA 5 6 THERMST
TABTYPE 7 QNODE TIME
TABDATA 7 1.200000E+07 0.000000E+00
    
```

ThermostatReport.csv

	A	B	C	D	E	F	G	H	I
1	Time	Thermostat ID/Name	On/Off	Temp Sensor	Power	Numbers	Total Time	Total Ener	P
2	1.0000000E+00	diode_h	1	2.0000000E+00	1.2000002	1	1.0000000	1.2000000	3.9600000
3	1.0000000E+00	diode_e	1	2.0000000E+00	1.1999999	1	1.0000000	1.1999999	3.9599999
4	1.0000000E+00	diode_t	1	2.0000000E+00	1.1999998	1	1.0000000	1.1999999	3.9599999
5	1.0000000E+00	diode_	1	2.0000000E+00	1.1999998	1	1.0000000	1.1999999	3.9599999
6	1.0000000E+00	diode_l	1	2.0000000E+00	1.2000000	1	1.0000000	1.2000000	3.9600000

- Verify consistent units.
- Heat loads and setpoints of parameters can be set as variables/expressions.

Checking heaters and heat loads

- If running a steady state analysis with thermostats or heater controllers, check the Thermostat setting in the solution parameters:

The screenshot displays the 'Space Systems Thermal' configuration window. On the left is a navigation pane with the following items: Solution Details (selected), Solution Units, Ambient Conditions, Initial Conditions, Restart, Thermal, Transient Setup, and Results Options. The main area is divided into several sections:

- Description:** A text input field with a copy icon.
- Solve Options:**
 - Run Directory: Solution Name
 - Solve Co-simulation
 - Values Outside Time-dependent Tables: Periodic
- Solution Type:**
 - Solution Type: Steady State
 - Transient Thermal Data to Use for Steady State: Loads at Specified Time
 - Use Loads at Time: 0 s
 - Thermostat: Sink to Average Temperature (highlighted with an orange box)
- Advanced:** (collapsed)
- Parallel Processing:** (collapsed)
- Material Export:**
 - Invalid Material Properties Check: Error

Checking named points

- Check named point location, ensure they match the correct model location.

Verifying orbits

- Check orbit parameters:
 - Solar flux, planet IR and albedo
 - Beta angles, Altitude, etc.
- Check model orientation using the **Orbit Visualizer** command.

Verifying solid motion effects

- Check articulation setup using **Orbit Visualizer**.
- Check articulation setup in post-processing using displacement results.

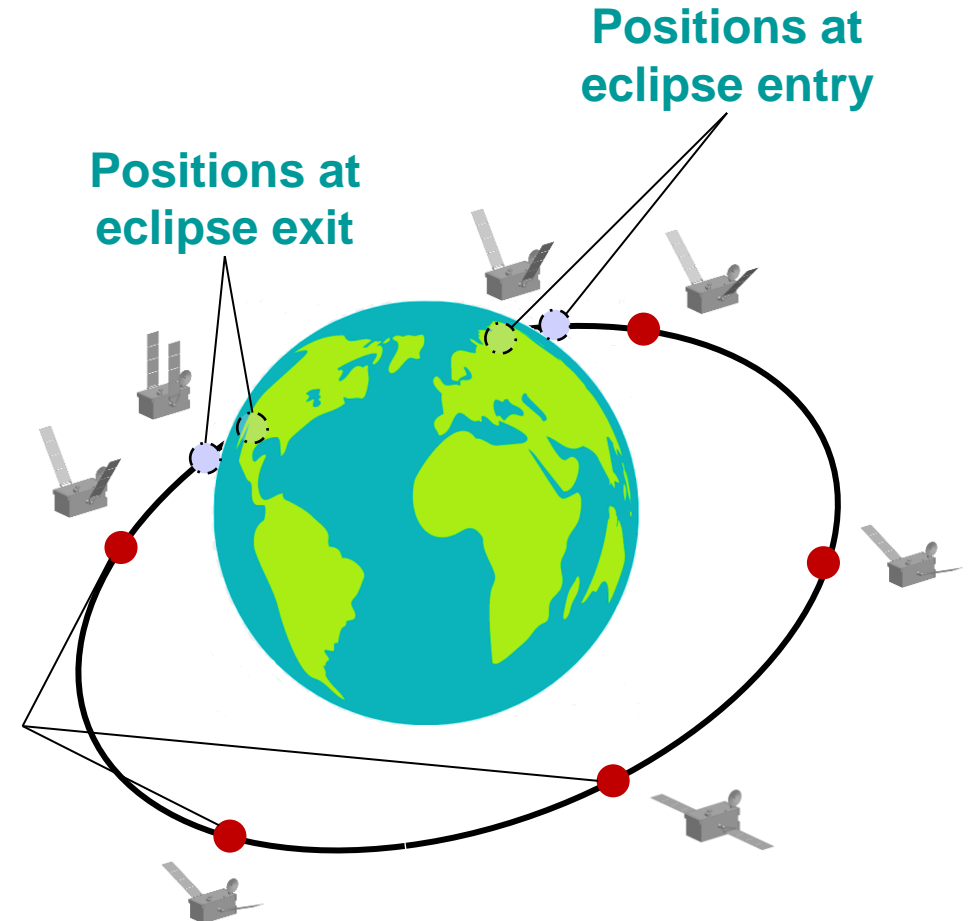
Computing articulation in the presence of an orbit

In the presence of an orbit, the solver computes the articulation positions at the orbit calculation positions to capture any sudden changes in orbital fluxes.

The number of orbit time steps depends on:

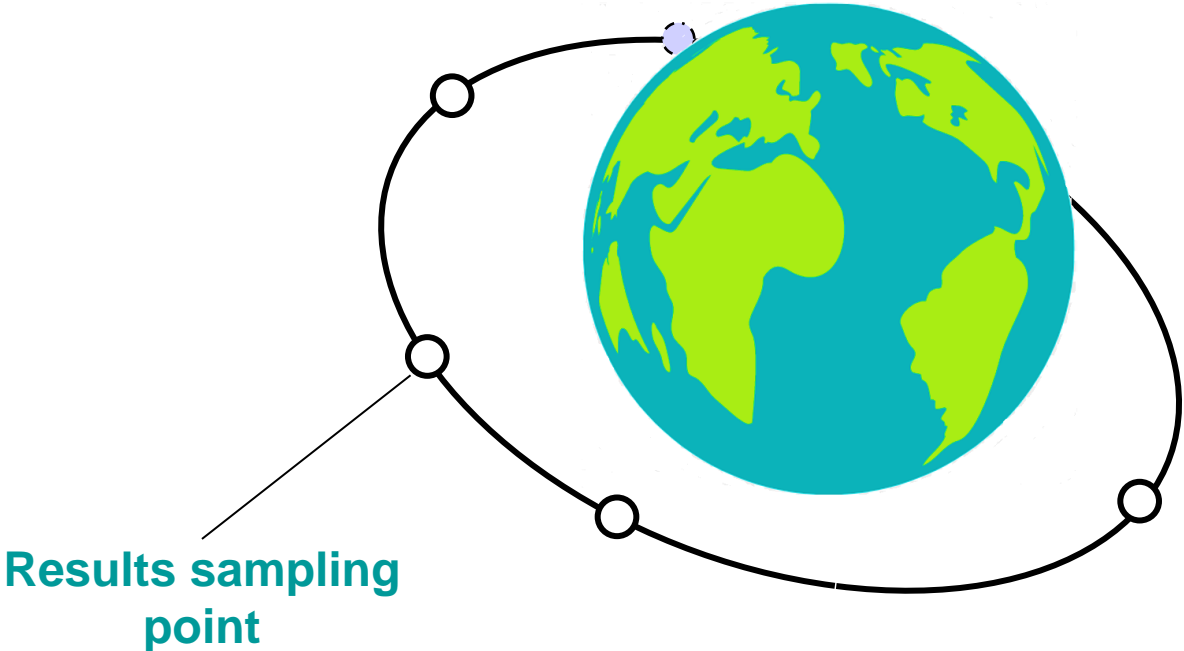
- The number of orbit positions.
- The presence of an eclipse in the orbit. The solver computes two additional calculation positions for eclipse entry and two more for eclipse exit.

Orbit positions



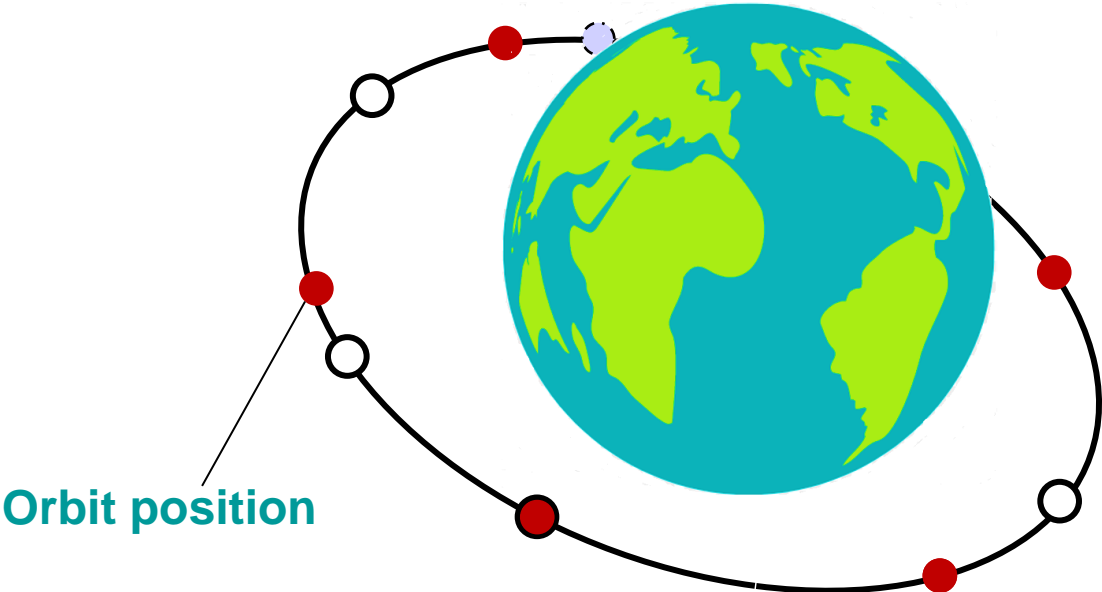
If you have the articulation rates defined using a table versus time, the solver adds the table times as extra time steps.

Post-processing articulation in the presence of an orbit



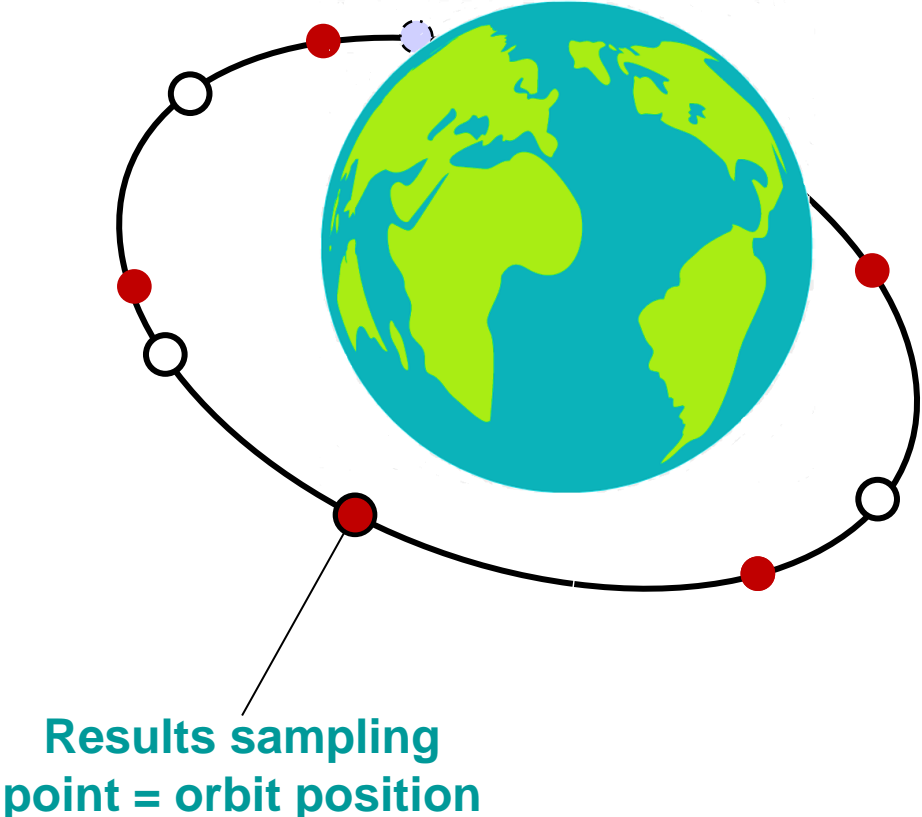
- Thermal
 - Increment 1, 0s
 - Articulation Displacement - Nodal
 - Temperature - Nodal
 - Temperature - Elemental
 - Absorbed Radiative Flux, PLANET IR - Elemental
 - Total Heat Load - Elemental
 - Total Heat Flux - Elemental
 - Environmental Gray View Factors - Elemental
 - Solar View Factor - Elemental
 - View Factors Sum - Elemental
 - Environmental View Factors - Elemental
 - Absorbed Radiative Flux, ALBEDO - Elemental
 - Absorbed Radiative Flux, SUN - Elemental
 - Earth View Factor - Elemental
 - Albedo View Factor - Elemental
 - Increment 2, 120s
 - Articulation Displacement - Nodal
 - Absorbed Radiative Flux, PLANET IR - Elemental
 - View Factors Sum - Elemental
 - Solar View Factor - Elemental
 - Environmental View Factors - Elemental
 - Absorbed Radiative Flux, ALBEDO - Elemental
 - Absorbed Radiative Flux, SUN - Elemental
 - Earth View Factor - Elemental
 - Albedo View Factor - Elemental
 - Increment 3, 200s
 - Temperature - Nodal
 - Temperature - Elemental
 - Total Heat Load - Elemental
 - Total Heat Flux - Elemental
 - Environmental Gray View Factors - Elemental

Post-processing articulation in the presence of an orbit



- Thermal
 - Increment 1, 0s
 - Articulation Displacement - Nodal
 - Temperature - Nodal
 - Temperature - Elemental
 - Absorbed Radiative Flux, PLANET IR - Elemental
 - Total Heat Load - Elemental
 - Total Heat Flux - Elemental
 - Environmental Gray View Factors - Elemental
 - Solar View Factor - Elemental
 - View Factors Sum - Elemental
 - Environmental View Factors - Elemental
 - Absorbed Radiative Flux, ALBEDO - Elemental
 - Absorbed Radiative Flux, SUN - Elemental
 - Earth View Factor - Elemental
 - Albedo View Factor - Elemental
 - Increment 2, 120s
 - Articulation Displacement - Nodal
 - Absorbed Radiative Flux, PLANET IR - Elemental
 - View Factors Sum - Elemental
 - Solar View Factor - Elemental
 - Environmental View Factors - Elemental
 - Absorbed Radiative Flux, ALBEDO - Elemental
 - Absorbed Radiative Flux, SUN - Elemental
 - Earth View Factor - Elemental
 - Albedo View Factor - Elemental
 - Increment 3, 200s
 - Temperature - Nodal
 - Temperature - Elemental
 - Total Heat Load - Elemental
 - Total Heat Flux - Elemental
 - Environmental Gray View Factors - Elemental

Post-processing articulation in the presence of an orbit



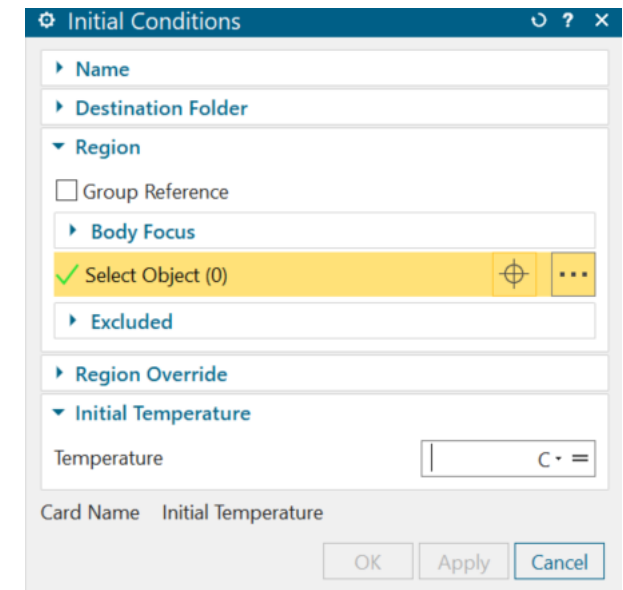
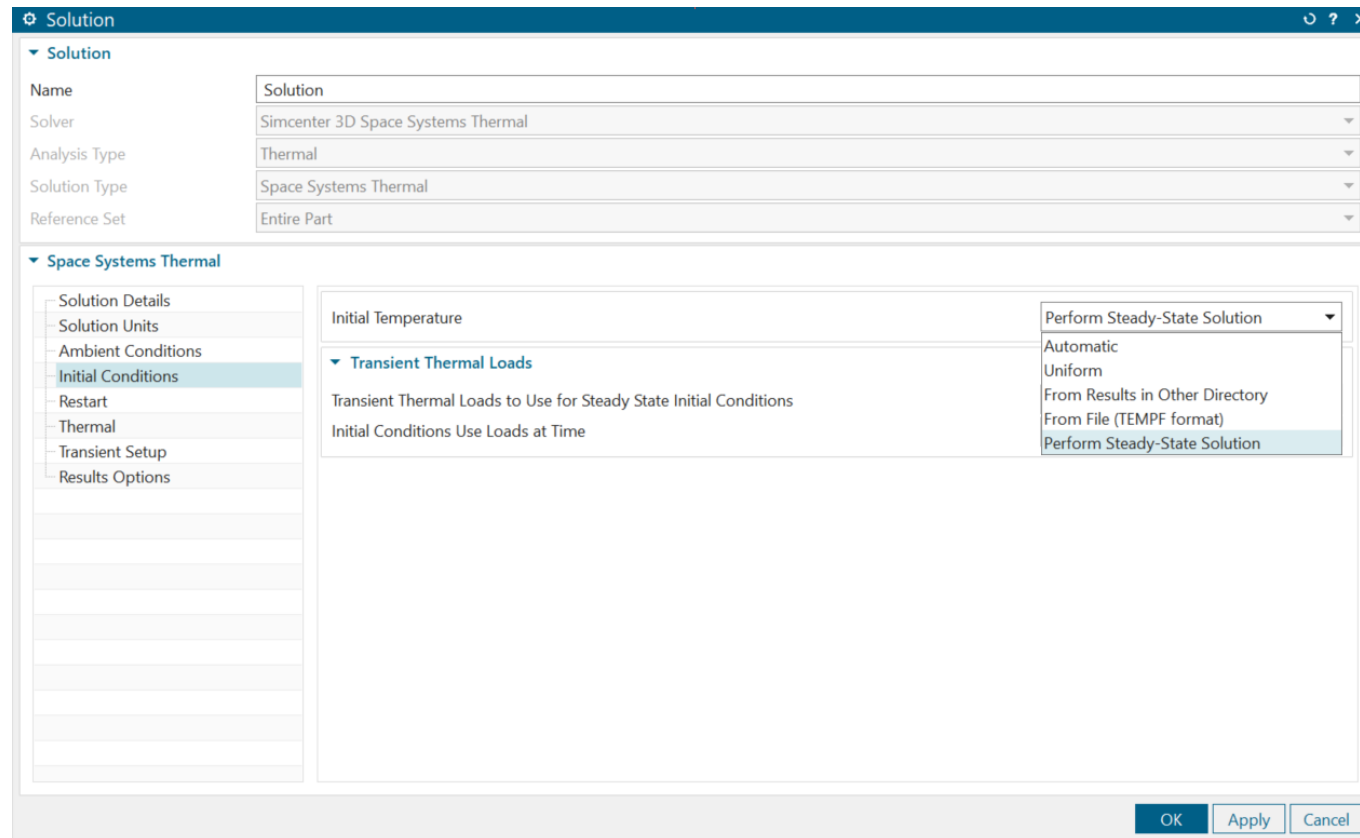
- Thermal
 - Increment 1, 0s
 - Articulation Displacement - Nodal
 - Temperature - Nodal
 - Temperature - Elemental
 - Absorbed Radiative Flux, PLANET IR - Elemental
 - Total Heat Load - Elemental
 - Total Heat Flux - Elemental
 - Environmental Gray View Factors - Elemental
 - Solar View Factor - Elemental
 - View Factors Sum - Elemental
 - Environmental View Factors - Elemental
 - Absorbed Radiative Flux, ALBEDO - Elemental
 - Absorbed Radiative Flux, SUN - Elemental
 - Earth View Factor - Elemental
 - Albedo View Factor - Elemental
 - Increment 2, 120s
 - Articulation Displacement - Nodal
 - Absorbed Radiative Flux, PLANET IR - Elemental
 - View Factors Sum - Elemental
 - Solar View Factor - Elemental
 - Environmental View Factors - Elemental
 - Absorbed Radiative Flux, ALBEDO - Elemental
 - Absorbed Radiative Flux, SUN - Elemental
 - Earth View Factor - Elemental
 - Albedo View Factor - Elemental
 - Increment 3, 200s
 - Temperature - Nodal
 - Temperature - Elemental
 - Total Heat Load - Elemental
 - Total Heat Flux - Elemental
 - Environmental Gray View Factors - Elemental

Configuring solution settings

- Check steady state or transient.
- Ensure Transient is long enough if running a periodic analysis.
- If you have heat loads with thermostats or heater controllers in the model, use transient analysis, it is more accurate and robust. Use steady state for a quick look.

Configuring solution settings – Initial Conditions

- Check the global ambient and initial conditions in the **Solution** dialog box.
- Set the local initial conditions in the **Initial Conditions** constraint.
Local overrides global conditions.



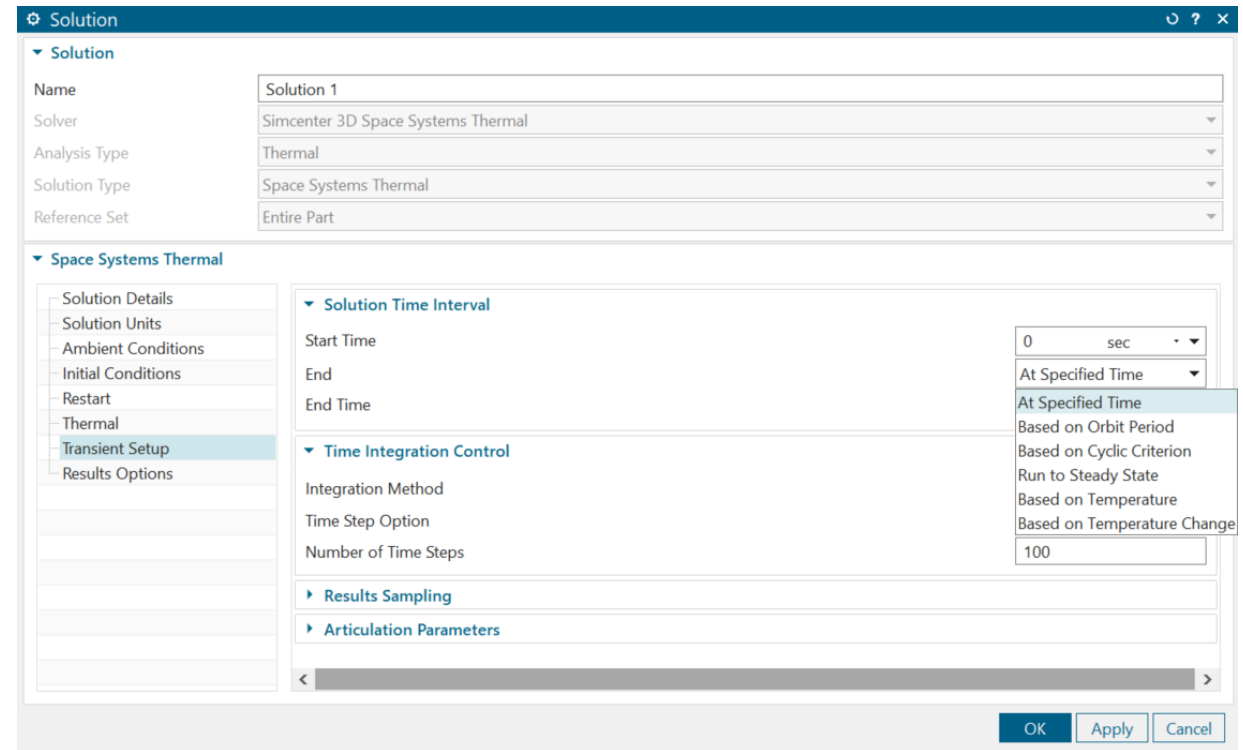
Configuring solution settings – Transient Setup

Verify transient solution, options:

- Start and end time for the transient solution.
- Time integration method, typically implicit is the recommended method.
- Time Step Option ensure time step isn't too large. A sensitivity can be run on this if time permits.

However, to speed up slow transient runs:

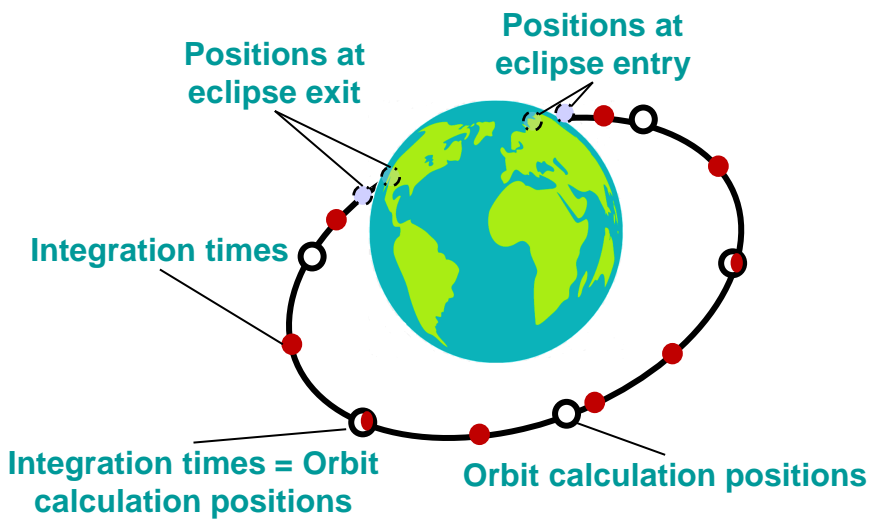
- Increase the maximum temperature difference convergence criterion.
- Select the implicit time integration method.
- Increase the integration time step.



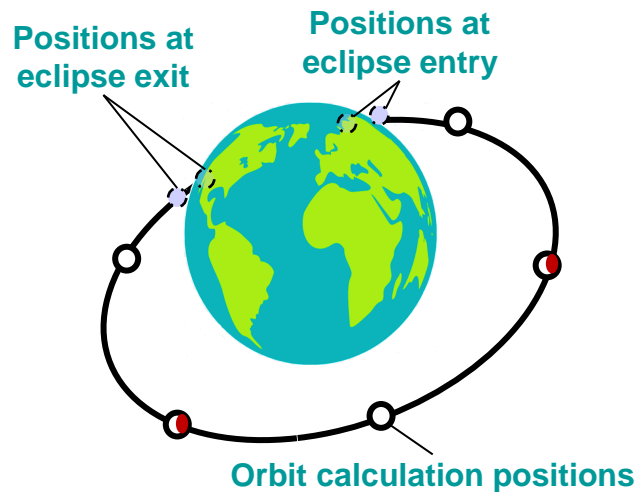
Understanding the integration time steps in the presence of an orbit

In the presence of orbit, the integration times include the orbit times, which are derived from the orbit calculation positions you specify in the **Orbital Heating** simulation object and the entry and exit positions of an eclipse. The total number of integration times depends on the number of time steps specified in the **Solution** dialog box.

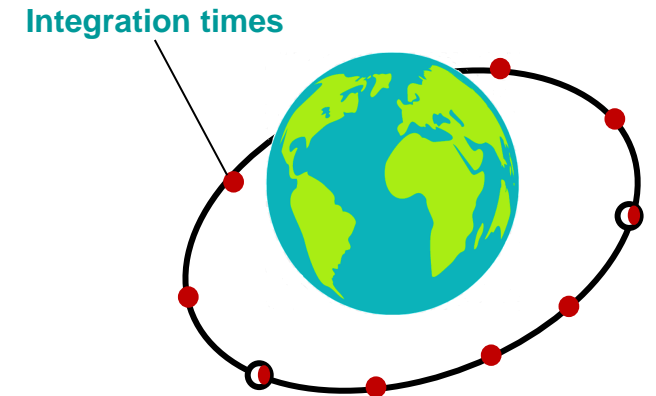
To not enforce integration at orbit times, you can use the **INTEGRATION ORBIT TIME** advanced parameter, which is located under **Thermal**→**Thermal Solver** in the catalog. The thermal solver uses the requested integration times without including the orbit calculation positions and the entry and exit of an eclipse.



Orbit and eclipse calculation positions, and requested integration time steps



Integration time steps in the presence of an orbit

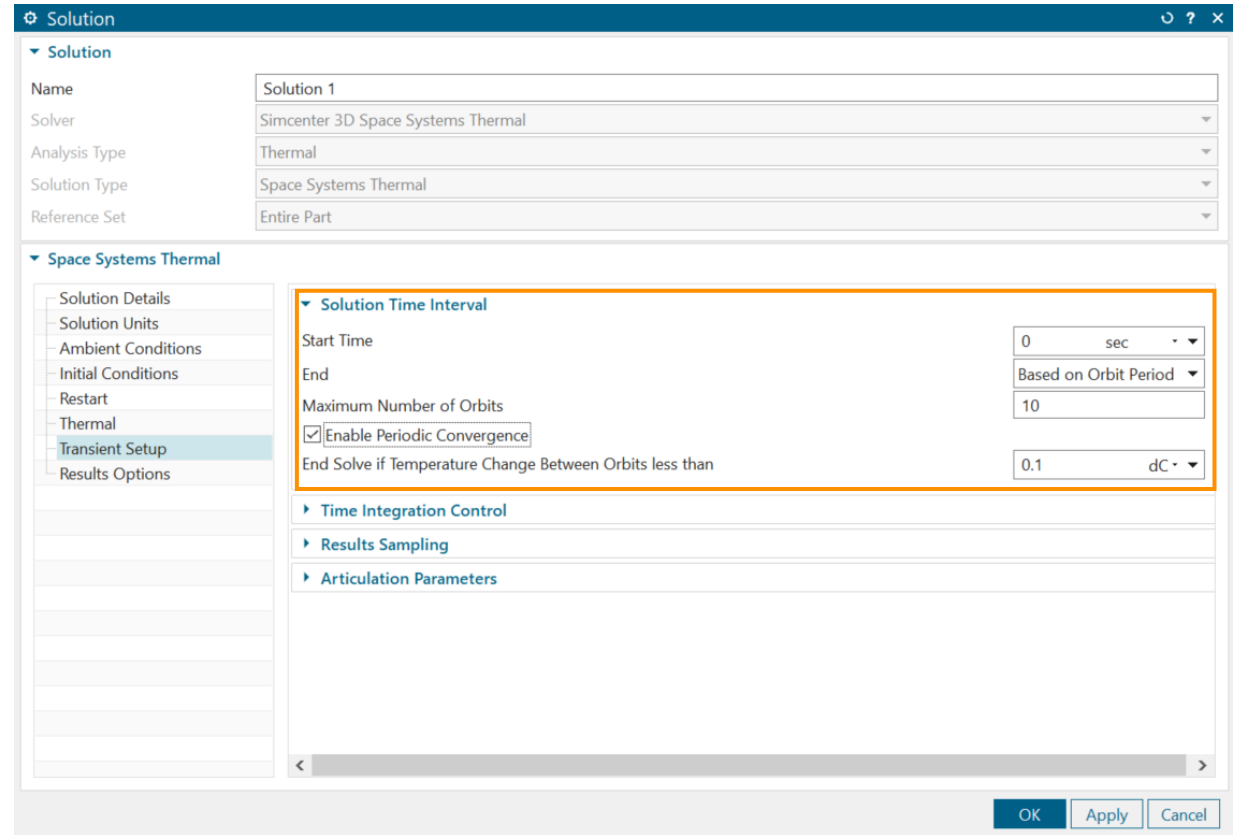


Integration time steps in the presence of an orbit with advanced parameter

Using the Enable Periodic Convergence option

For an orbital solution, when you set the **End** time to **Based on Orbit Period**, it is a good practice to select **Enable Periodic Convergence** to stop the simulation when the temperature difference of each element at the end of two successive orbits is less than the value specified in the **End Solve if Temperature Between Orbits** less than box.

It is important to check this manually if this option was not used or if the solver reached the maximum number of orbits.



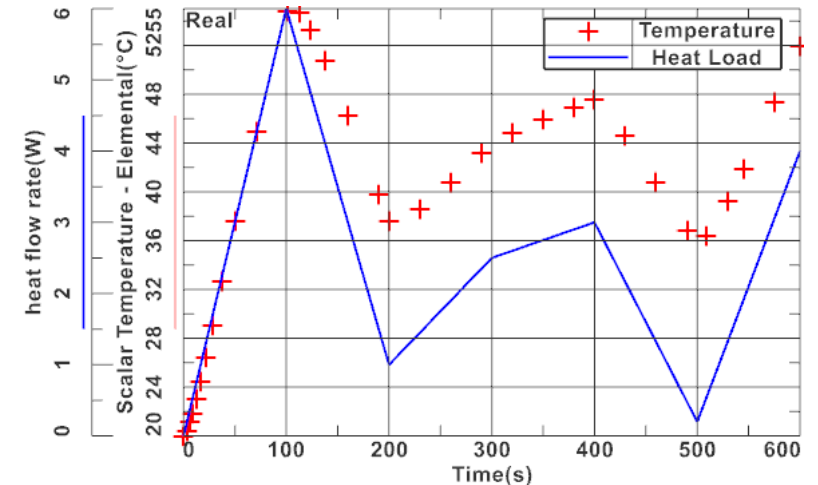
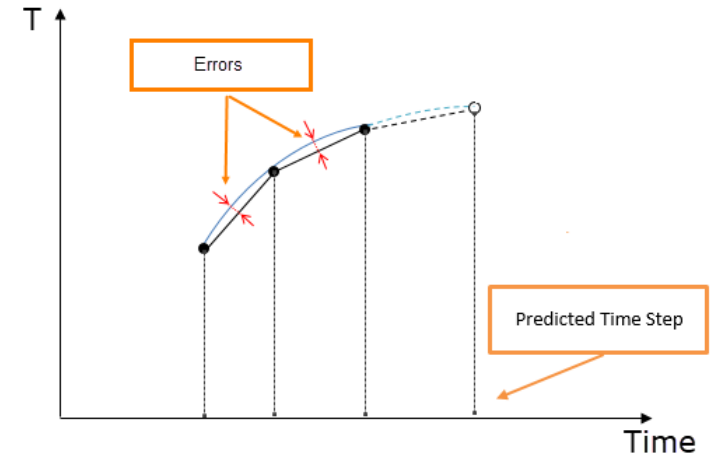
Using the automatic time step

One way to speed up a solve is to use the automatic time stepping.

The automatic time step size calculation is based on the estimated error between a quadratic fit and a linear fit through three consecutively computed temperature values for two consecutive time steps.

As shown in the graph, the adaptive time stepping scheme creates smaller time steps around the times when the abrupt changes occur.

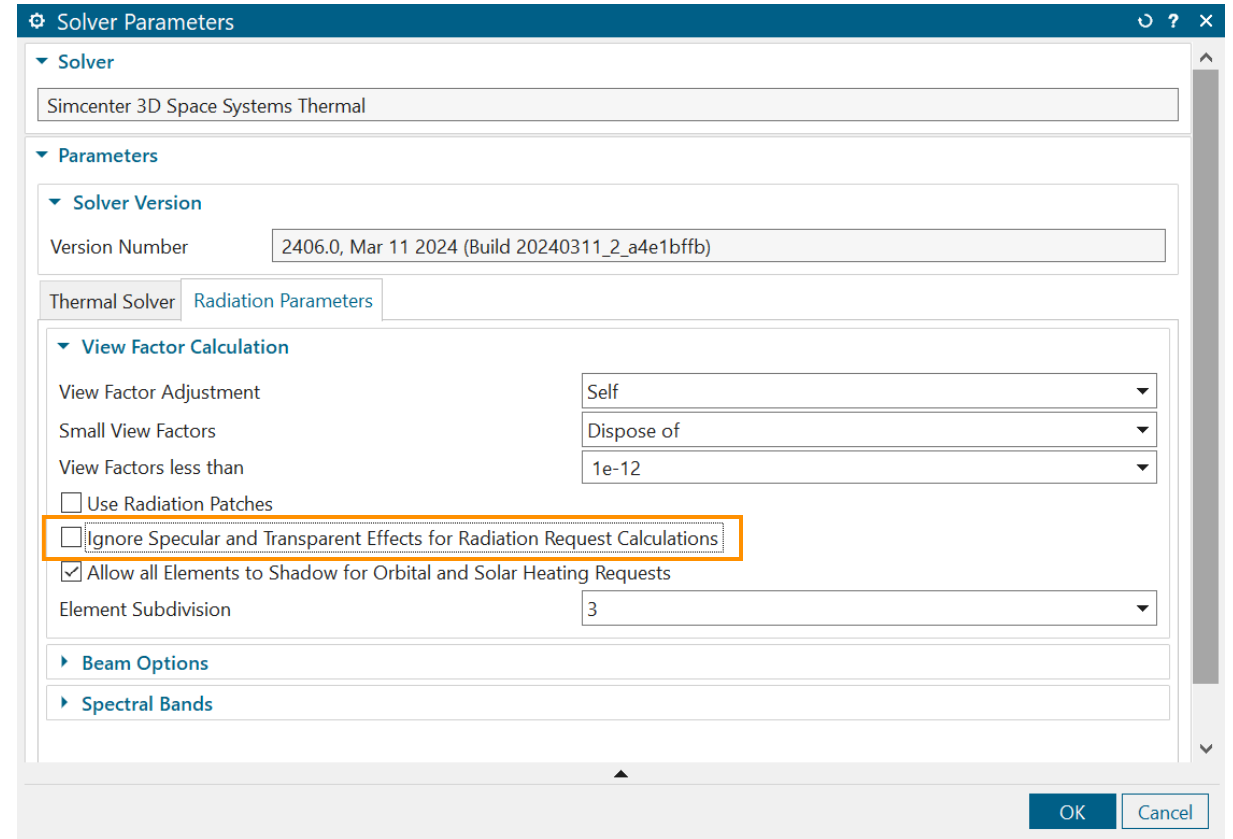
The blue curve represents the time-varying heat load that is applied to a boundary condition, and each red dot represents the temperature value at the point where the boundary condition is applied. The dots that are close to each other indicate that the time steps are smaller at those times, to better capture the changes in the heat load.



Accounting for specular and transparent effects

If you have any materials that are transparent or specular in the IR spectrum, make sure to clear the **Ignore Specular and Transparent Effects for Radiation Request Calculations** option when specular and transparent effects for black body view factors in enclosures are significant in your model.

However, do keep in mind this will most probably significantly impact the run time of the view factor calculations.

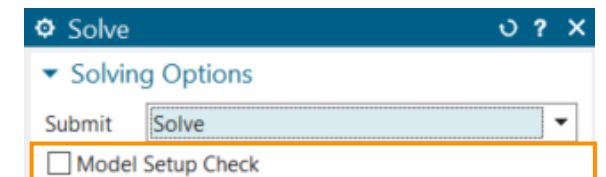
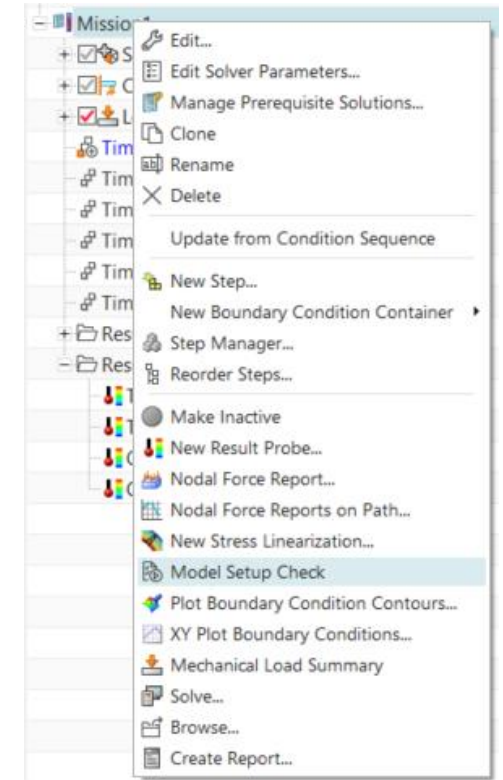


Running a model setup check

Run model setup check on solution by right-clicking **Solution** and selecting **Model Setup Check** or selecting the **Model Setup Check** checkbox in the **Solve** dialog box. Look if there are any errors or warnings.

Model Setup Check outputs model checks to **Information** window on:

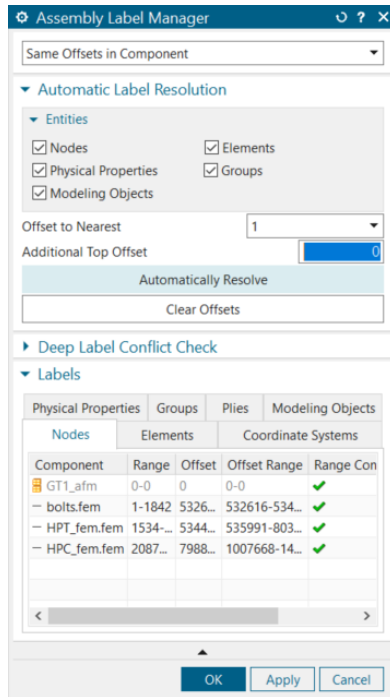
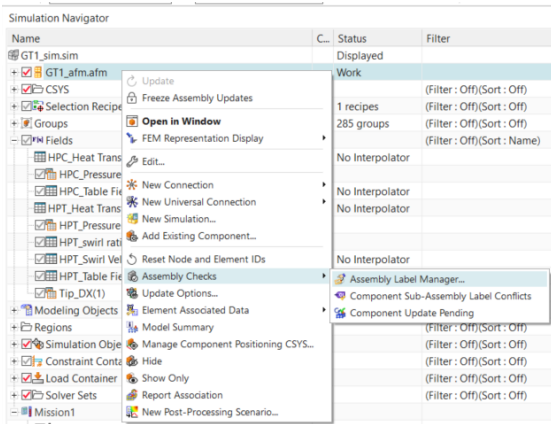
- Assembly fem label conflicts
- Simulation label conflicts
- Mesh / Materials / Physical properties check
- Groups
- Loads, Constraints, Boundary Conditions (invalid selections/values)
- Solution



Resolving ID conflicts

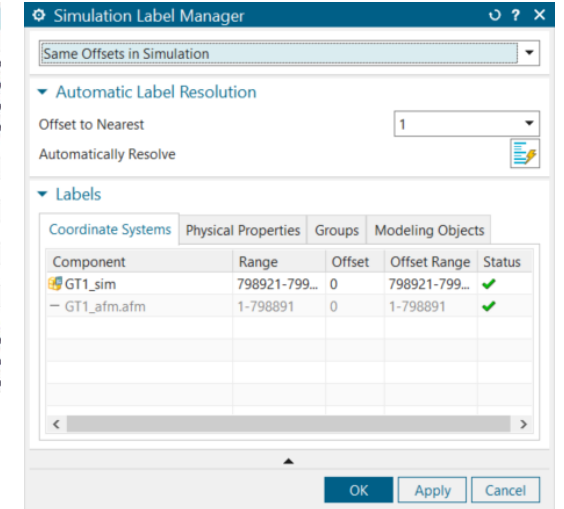
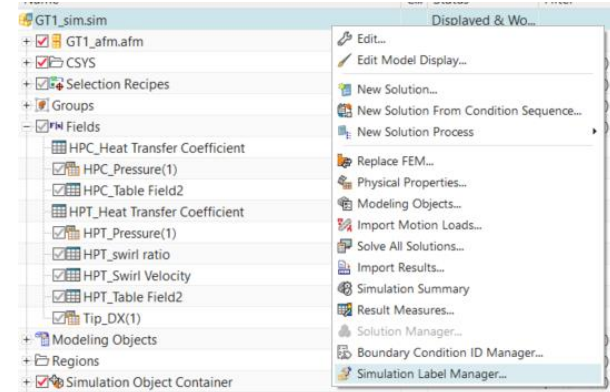
AFM label conflicts

Right-click the active assembly FEM file →
Assembly Checks → **Assembly Label Manager**



Simulation label conflicts

Right-click the active Simulation file →
Simulation Label Manager



Inspecting log files

This is the **first place to look** in the case of a solver crash. This log file may contain some specific details on why the model crashed.

Check the following:

- Warnings or error messages.
- Convergence data.
- Heat flow summary at end of log file if running a steady state analysis.

For more information about the files, see [Overview of thermal solver files and how to use them](https://support.sw.siemens.com/en-US/product/289054037/knowledge-base/KB000128451_EN_US) (https://support.sw.siemens.com/en-US/product/289054037/knowledge-base/KB000128451_EN_US).

```
Time= 10000.0000000    Integration timestep= 900.000
Cpu time in ANALYZER module= 222.5

Minimum temperature      = 288.150 at element 1296635
Maximum temperature     = 569.913 at element 1136454
Average temperature     = 295.980

Heat Flow+Load Summary Into Different Sink Entities:

      Sink Entity      Temperature      Heat      Energy absorbed
                          Flow+Load      since start
-----
HPT_Duct_Inlet_Temp_1    3.933E+02    -4.694E-07    -3.995E+01
HPT_Duct_Inlet_Temp_2    3.931E+02    -5.655E-10     3.941E-05
Sink elements with no entity names:  3.673E+02    -7.620E+07    -5.311E+12

...done.

-----
|                               END                               |
-----

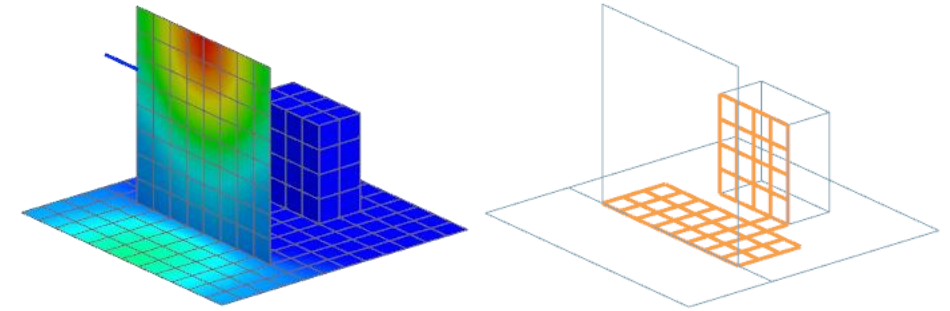
Solution elapsed time: 05 min 06 sec

Solve completed at:
=====
Time: Tue Aug 29 16:30:46 2023
```

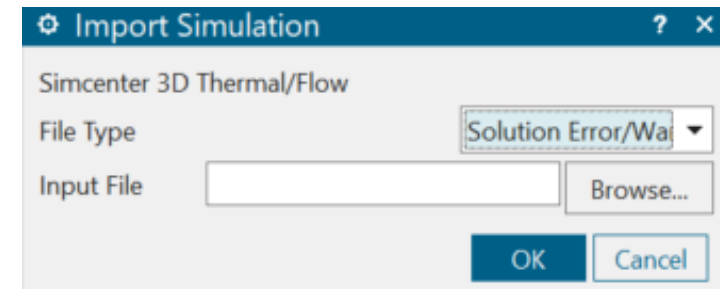
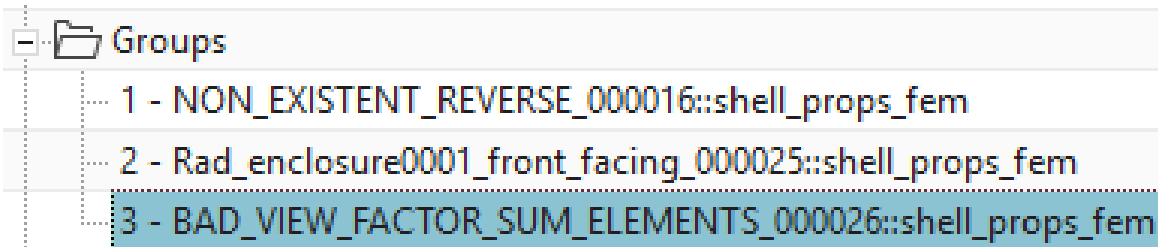
Resolving a warning

Review the messages in the Solution Monitor or the log file for the thermal and flow solvers:

```
** WARNING 4316 **
** The view factor sum error of the following 41 element(s)
** exceeds 20%. The total number elements with non-zero
** view factor sums is 267. Incomplete enclosures may
** exist in the model. A complete element list can be
** found in file groups.unv under the group name:
** BAD_VIEW_FACTOR_SUM_ELEMENTS_000026
**      422      423      424      425      426      427      428      429
```



Import the solution warning groups and observe the failed elements. Choose **File** → **Import** → **Simulation**, select **Simcenter 3D Thermal/Flow**.



Analyzing results

- Check for:
 - Large temperature gradients.
 - Sudden changes in temperatures.
 - Unreasonable looking min/max temperatures.

Using Thermal Coupling Report tool

- An excel file to generate table of thermal coupling data for a model.
- **This is not commercial grade software! It is provided as is and not supported by us.**
- Select all thermal coupling in the model right-click on them and click on Information. Save the information window to a text file and you can import this text file into the excel sheet.

	A	B	C	D	E	F	G	H	I
1	Name	Description	Type	Value					Thermal Coupling List File: D:\users\Test.txt
2	TC-Hotbox-2	3.5mm bolts, small stiff surface 2.37 C/W x2	Total Conductance	1.68776371308017 W/°C					
3	Regulator-PCB	Perfect contact	Total Conductance	1000 W/°C			Read File		
4	Regulator-PCB-2	Perfect contact	Total Conductance	1000 W/°C					
5	Regulator-PCB-3	Perfect contact	Total Conductance	1000 W/°C					
6	Regulator-PCB-4	Perfect contact	Total Conductance	1000 W/°C					
7	GapPad	Sil-pad	Conductive Gap	0.9 W/(m*K)					
8	PCDU-PCB-HeatSink	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					
9	PCDU-PCB-HeatSink(2)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					
10	PCDU-PCB-HeatSink(3)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					
11	PCDU-PCB-HeatSink(4)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					
12	PCDU-PCB-HeatSink(5)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					
13	PCDU-PCB-HeatSink(6)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					
14	PCDU-PCB-HeatSink(7)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					
15	PCDU-PCB-HeatSink(8)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C					

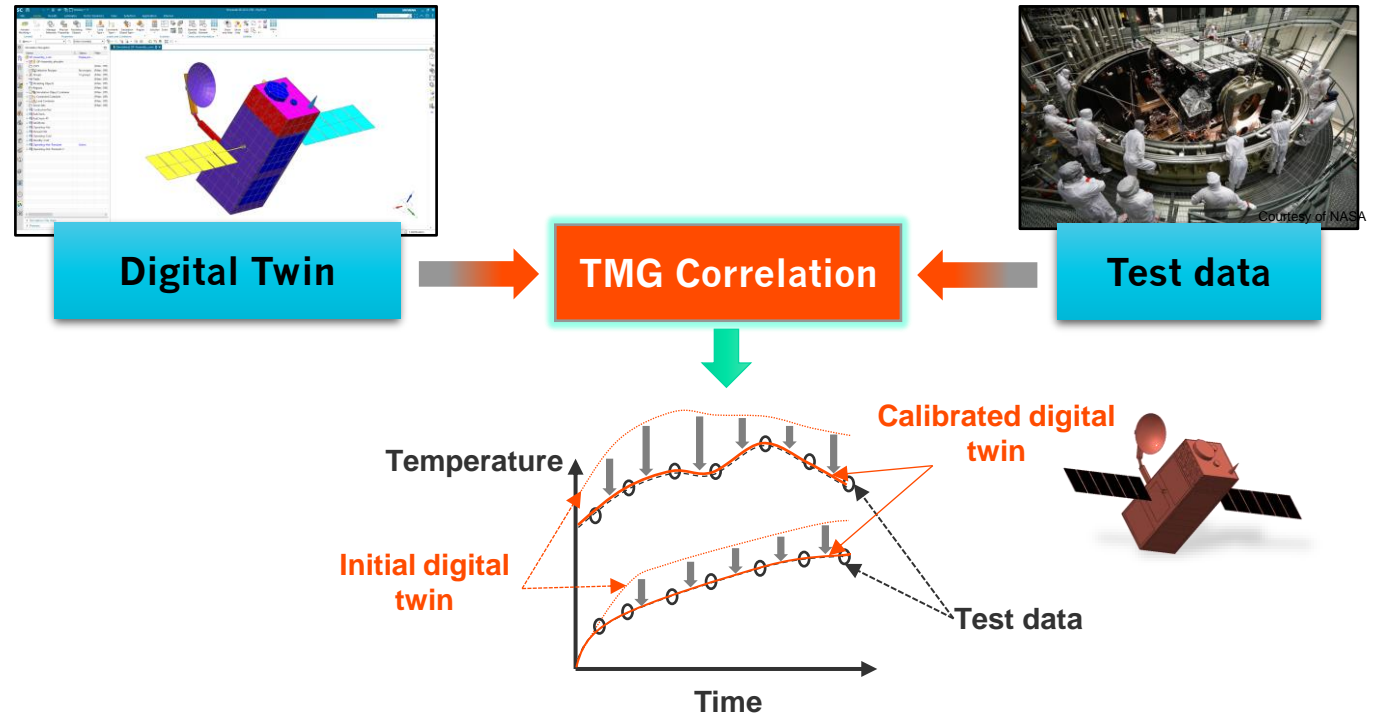
See the ThermalCoupling.xlsm file linked to this Knowledge Base Article.

Using the TMG Correlation tool

With extensive applications in space systems, gas turbine, electronics, automotive and cold chain, TMG Correlation provides unique and state of the art solver capabilities for fast and easy correlation of thermal models to test or analysis data. https://help.mayahtt.com/tmg_correlation/v1/index.html

TMG Correlation:

- **Correlate** your digital twin in **days** instead of months.
- Calibrate **steady-state** and **transient** problems with **hundreds of design variables**.
- Apply **weighting factors** to put more emphasis on critical sensors.



Best practices

- Add descriptions to boundary conditions.
- Leave formula for conductance calculations.
- Use descriptive names for solution/simulation objects.
- Clean model with no unused materials or Modeling Objects.
- Avoid using perfect contacts.