Model checks in Space Systems Thermal

SIEMENS

Why do model checks?

- Detect errors and inconsistencies in the simulation setup.
- Reveal incorrect simplifications.
- Check that the boundary conditions are correctly defined and represent the real-word 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.

Model checklist

Check units
Check material and optical properties
Verify elements quality and mesh
Check for duplicate nodes
Display all thermal couplings
Check all warning messages for thermal couplings and consistent units
Check radiation enclosures
Check MLI set up
Do model mass check
Check heaters and heat loads, verify consistent units
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

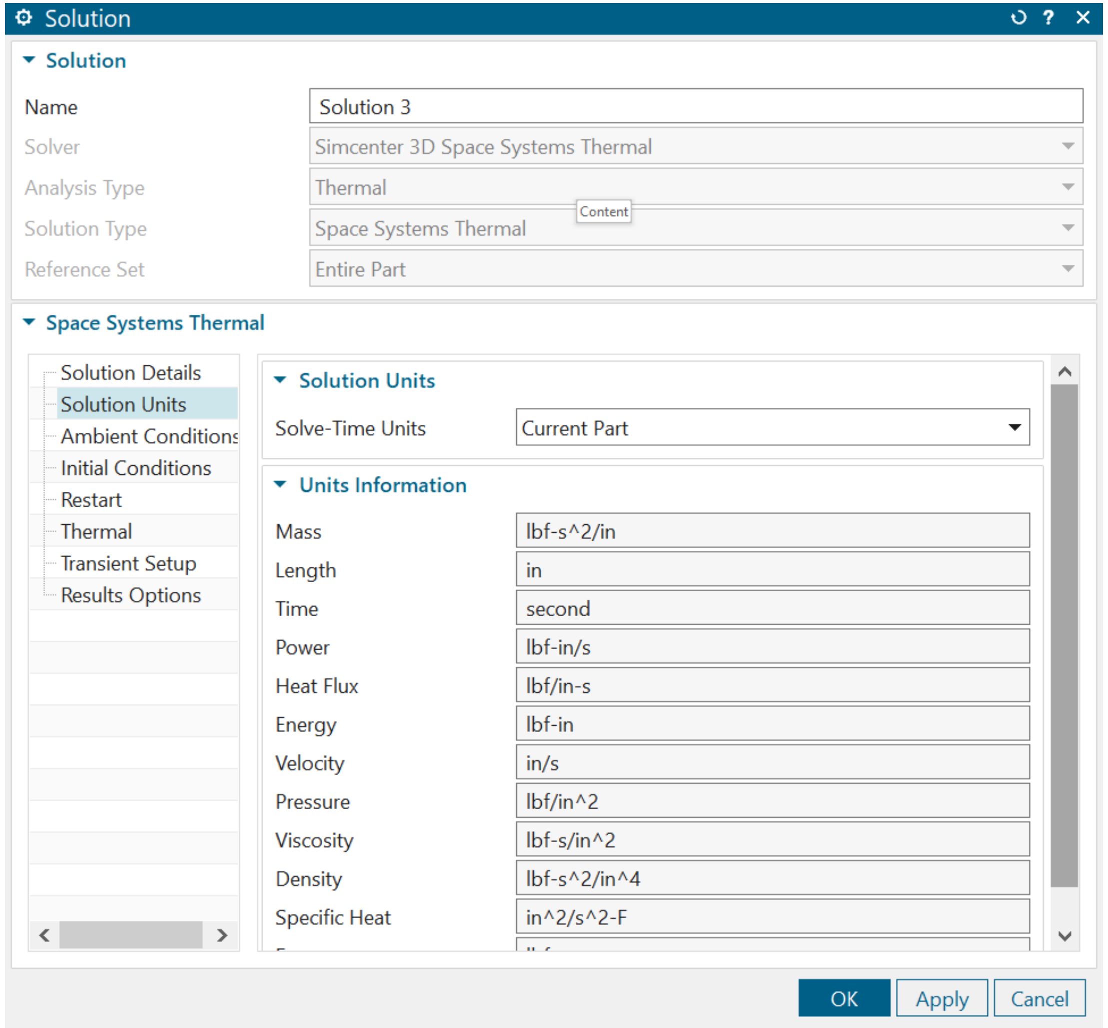
Most common issues

- Units
- Material and optical property assignments
- Thermal coupling setup
- Radiation enclosure setup

Units

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

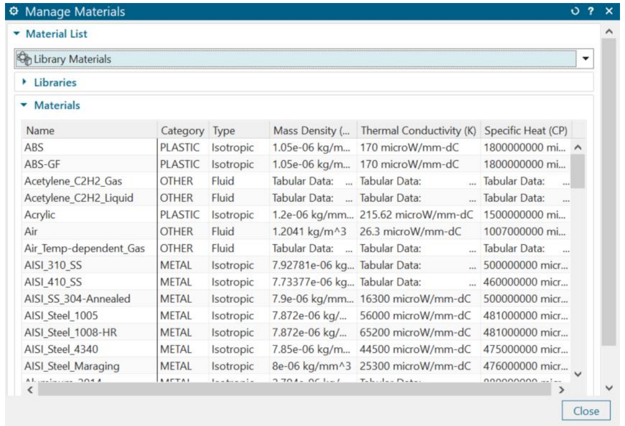
- Check user logic (expressions) for problems that could arise from changing units.
- Make units consistent.
- Do one of the following, since not all units are case insensitive.
  - Use the auto-complete suggestion for the unit.
  - Use the case for the unit as shown in the **Units Manager** dialog box.

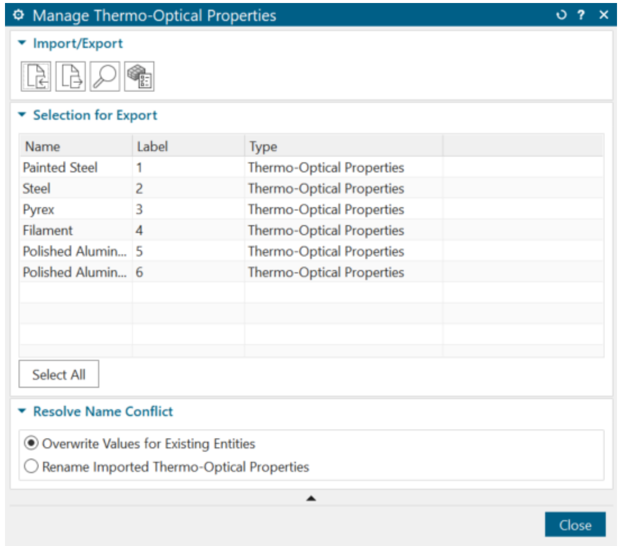


SIEMENS

Material

- 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 Cp 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.





SIEMENS

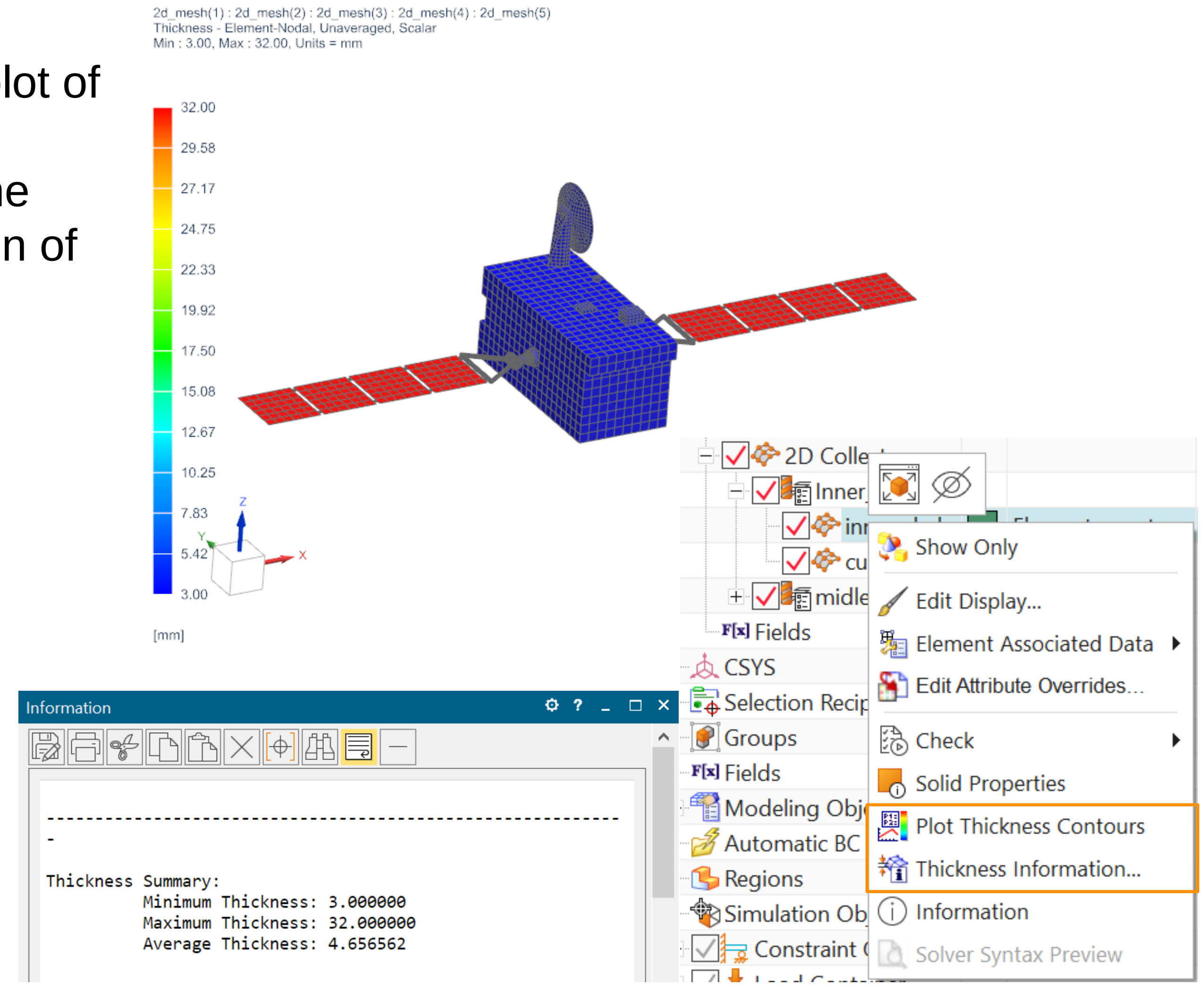
Thickness

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 a 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.

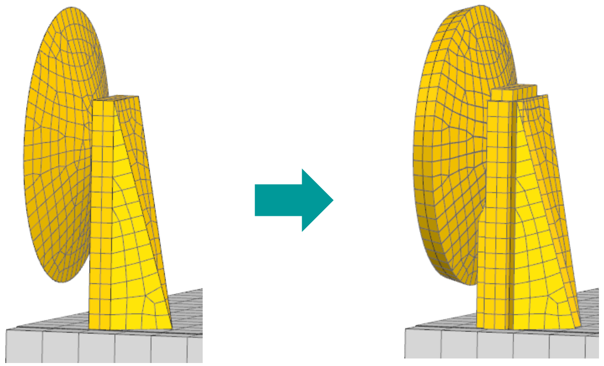


SIEMENS

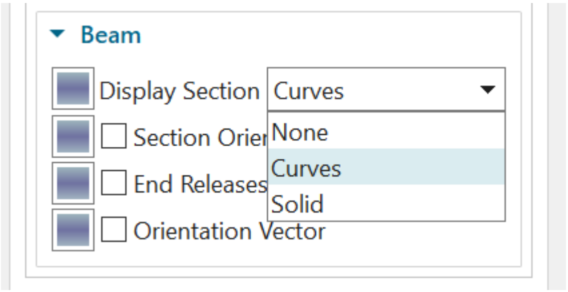
Thickness

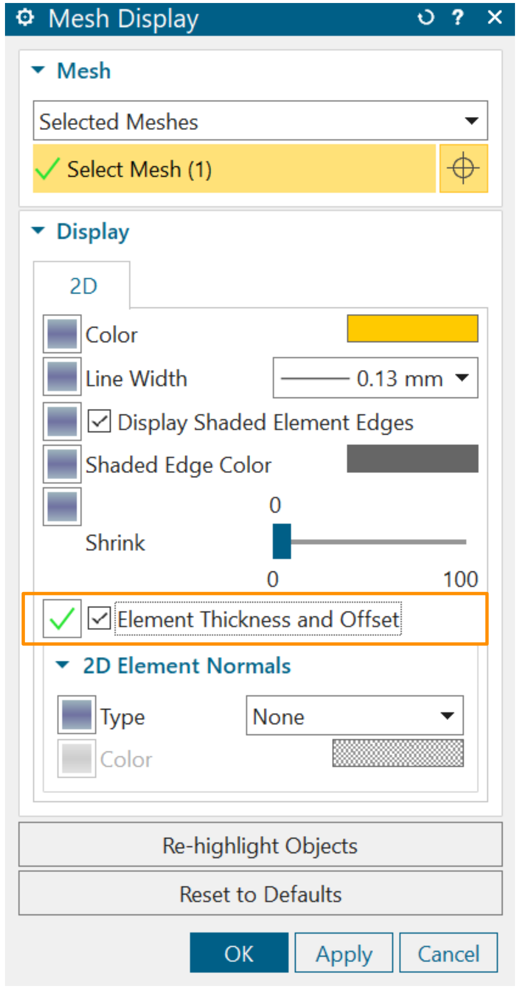
You can use the **Edit Display** command (right click on mesh) to:

- 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**.





SIEMENS

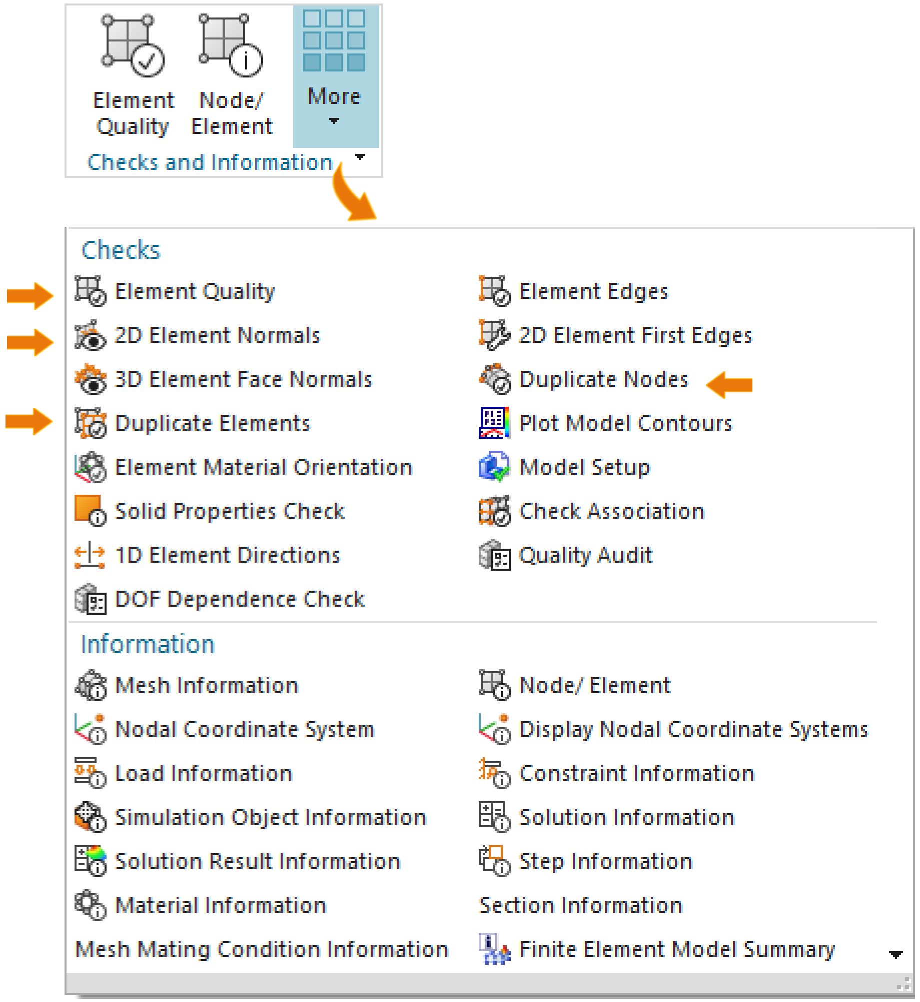
Mesh Density

- Check visually for appropriate mesh density.
  - Are there enough elements to capture temperature gradients? A common mistake is to create meshes which are much too fine.
- Perform a mesh sensitivity study to assess the adequacy of a mesh. 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.

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.

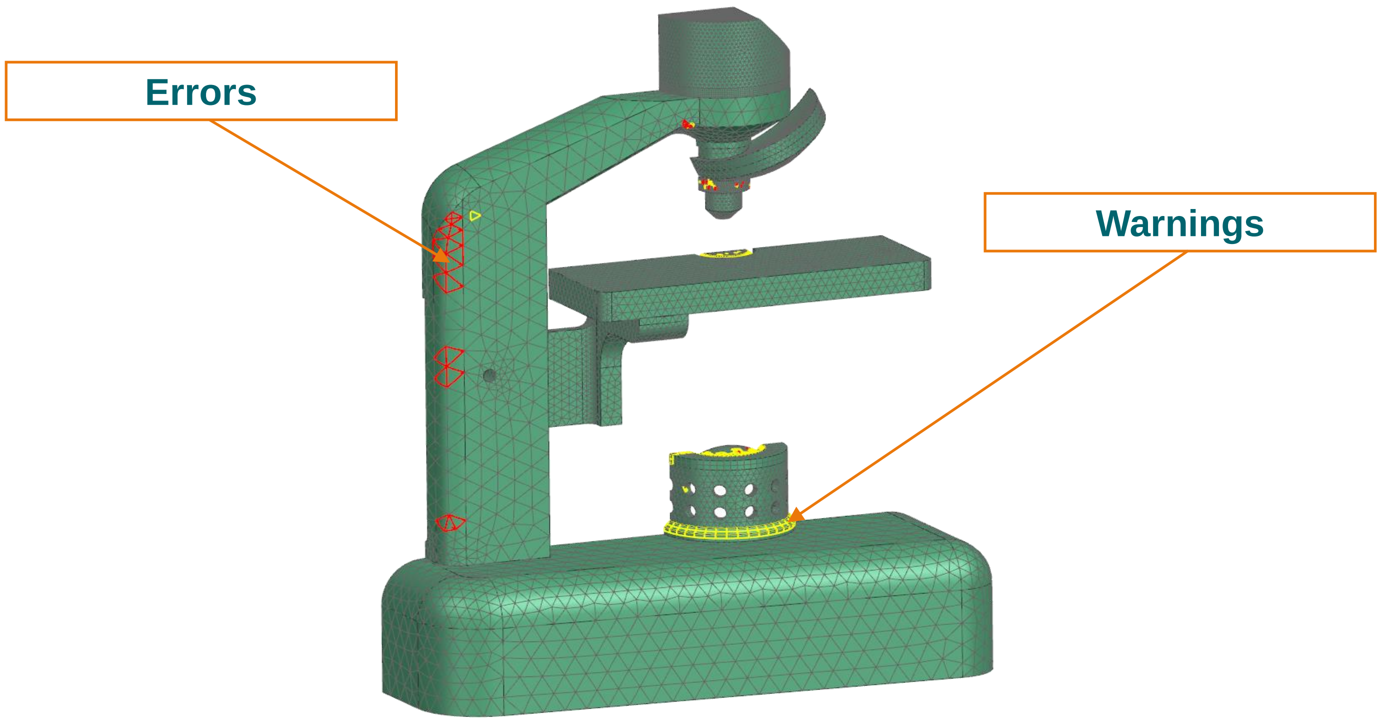


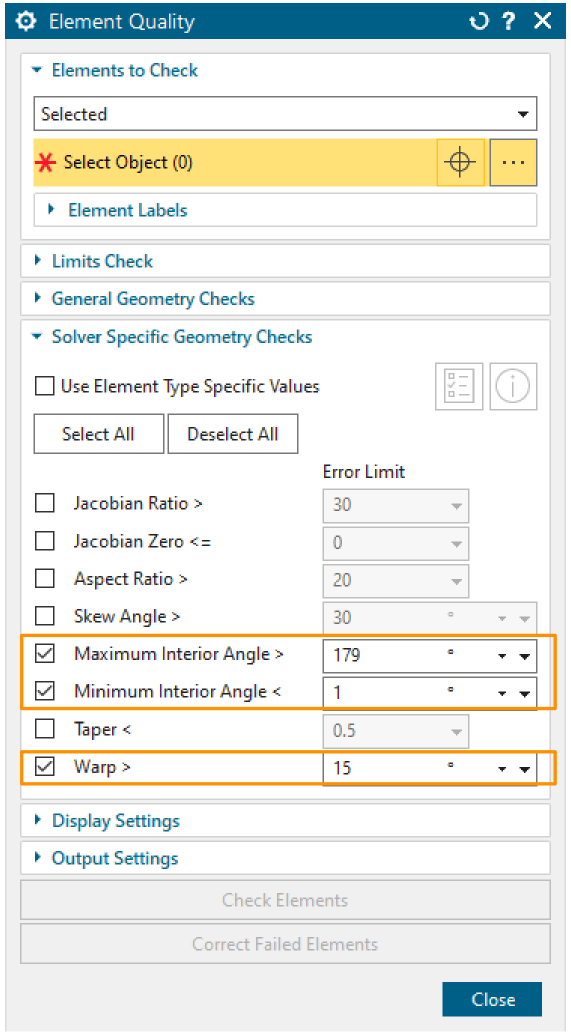
SIEMENS

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°<α<179°	β<15°





SIEMENS

Coincident nodes

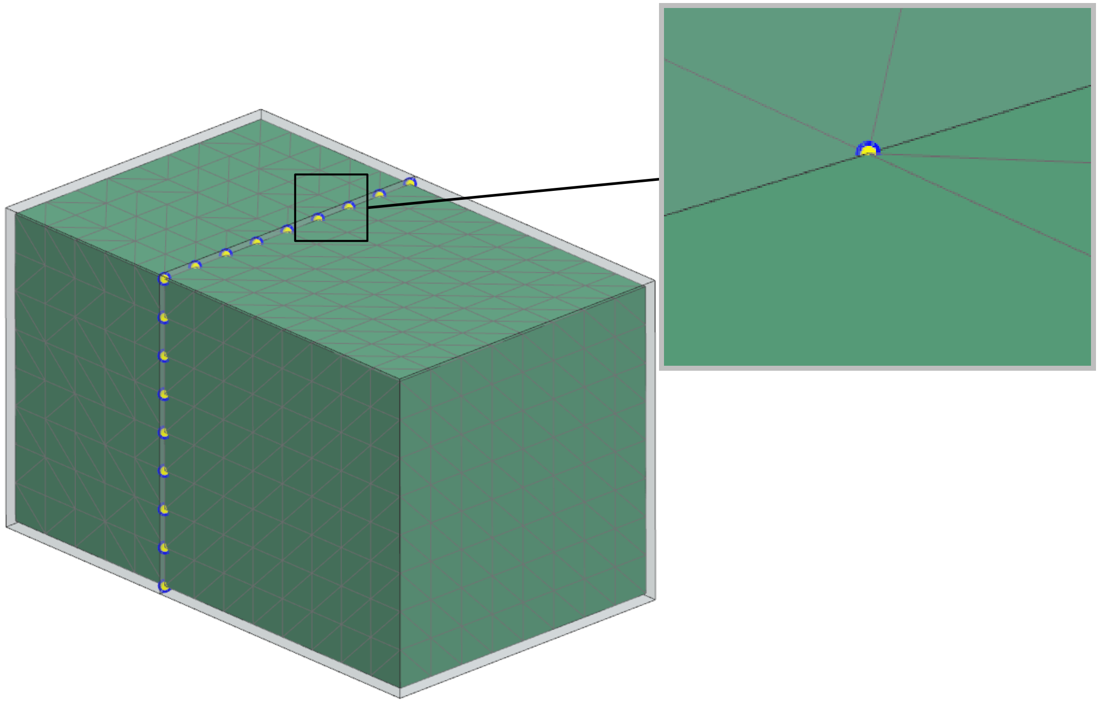
Check for coincident nodes which are duplicate nodes lying on top of each other.

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



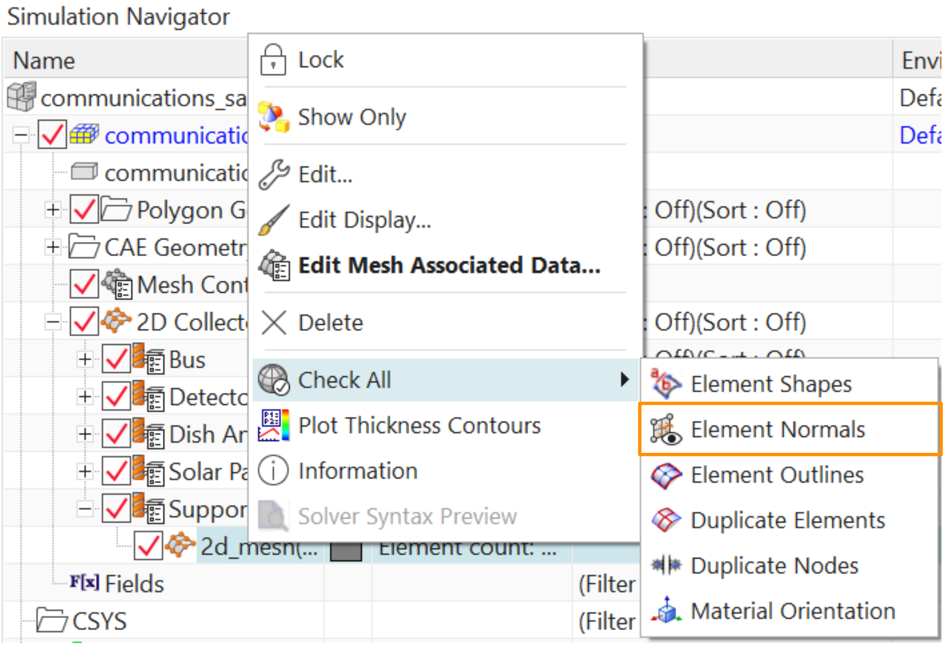
SIEMENS

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.

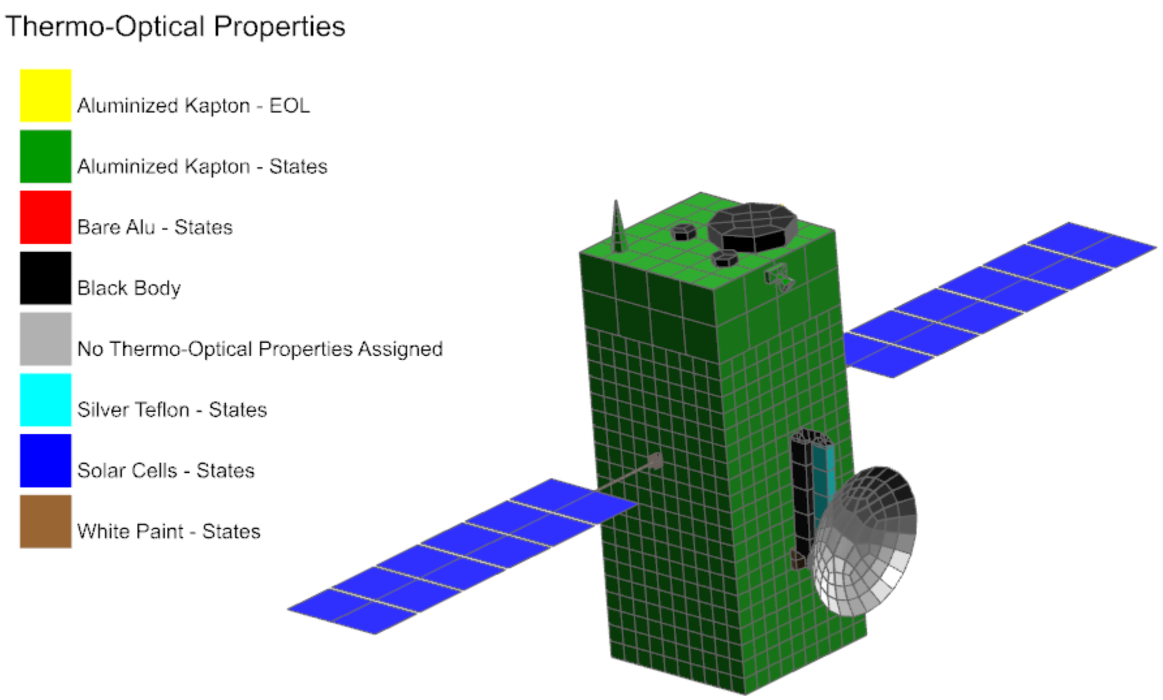
- Inspect the optical properties by right-clicking the mesh node and selecting **Information**.

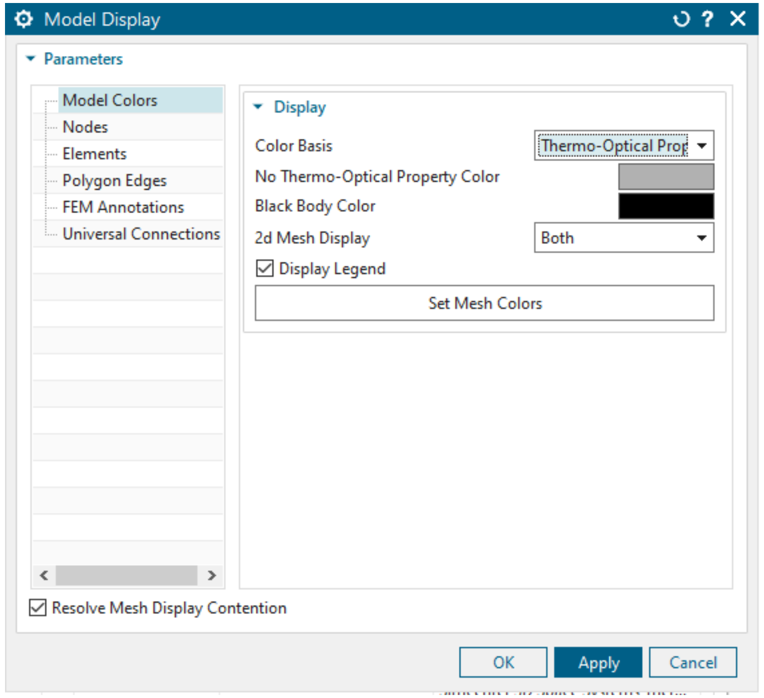


SIEMENS

Thermo-optical color display

- 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





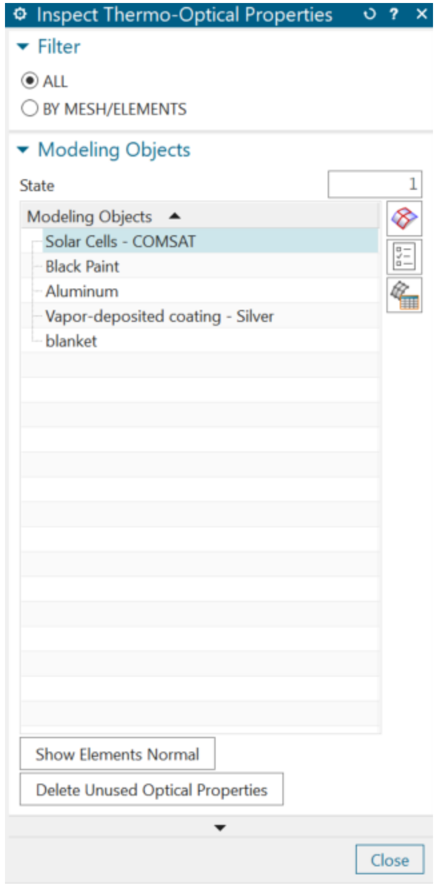
SIEMENS

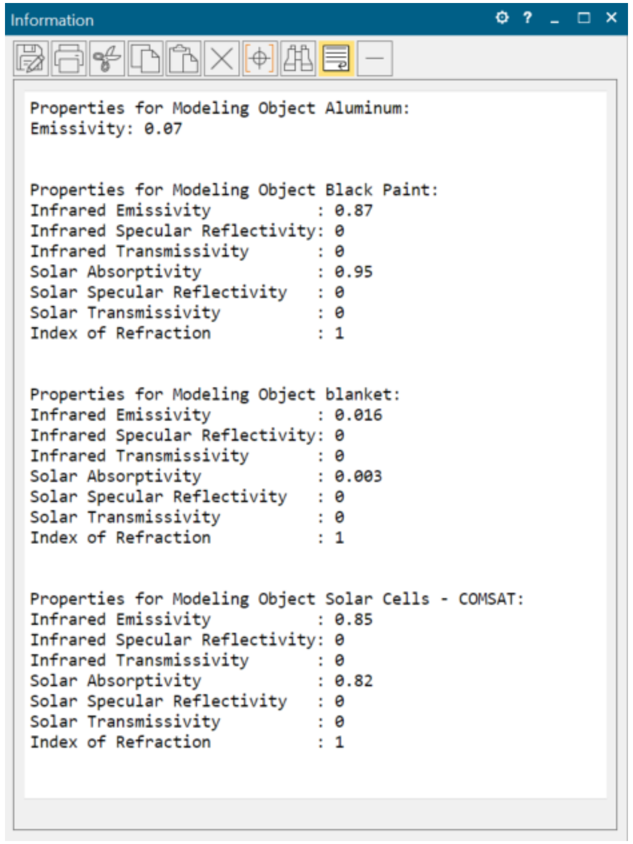
Inspecting thermo-optical properties

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.





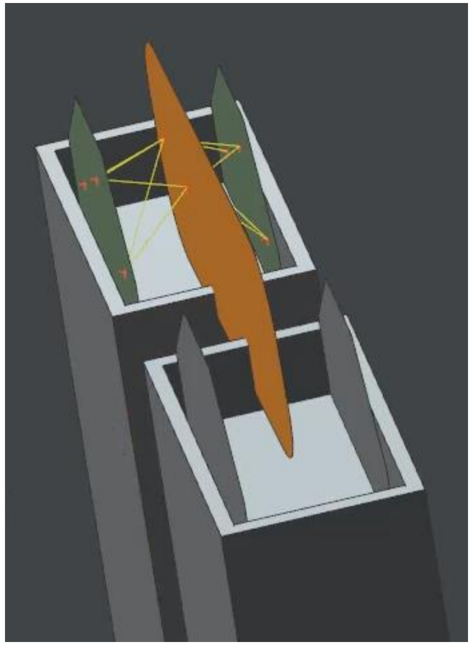
Set UGII_USER_DIR to <software installation>\nxcae_extras\tmg\customization\OpticalProperties_Tool folder before starting the software.

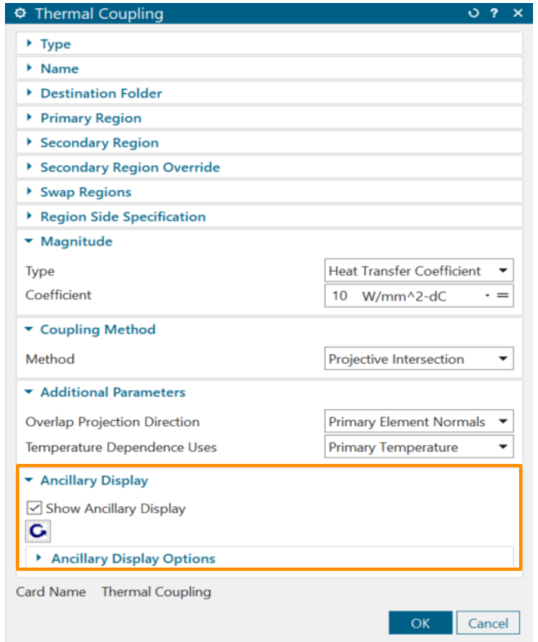
Inspecting thermo-optical properties

- 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.

Thermal Couplings

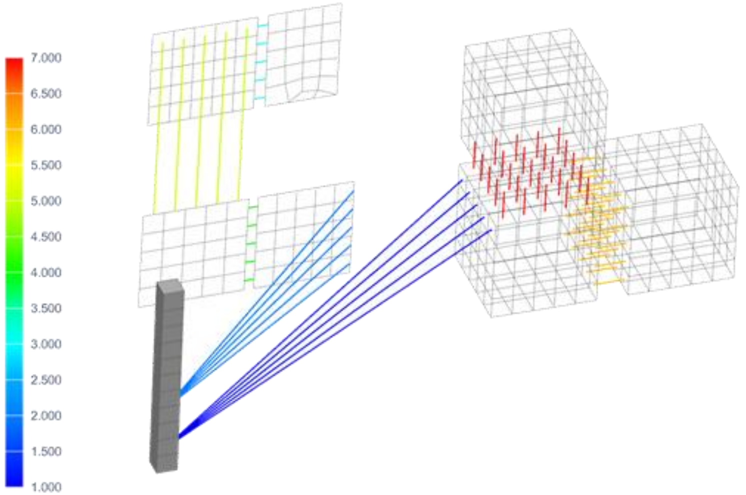
- Verify that the thermal coupling setup in a physically meaningful way.
- Verify the selection region:
  - Select the smaller segment as a primary region.
  - Select the coarse mesh as a primary and fine mesh as a secondary region.
  - Note that the primary element selection does not control the direction 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.





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.



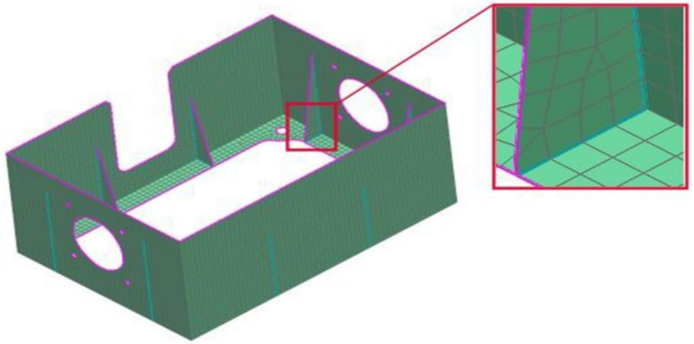
Thermal Connections – Elemental result set

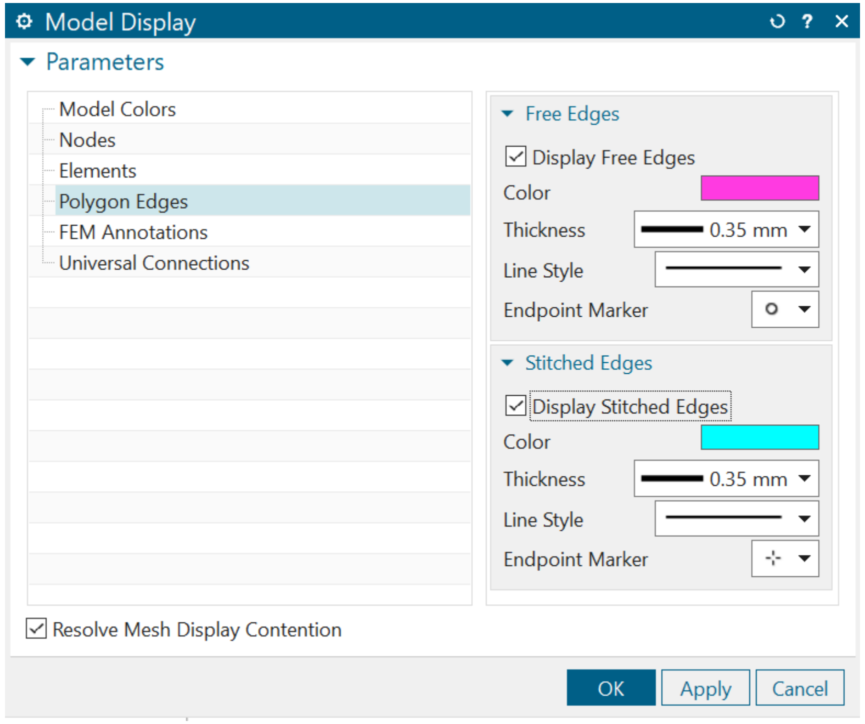
Radiation enclosures

- 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.

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** command, 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.

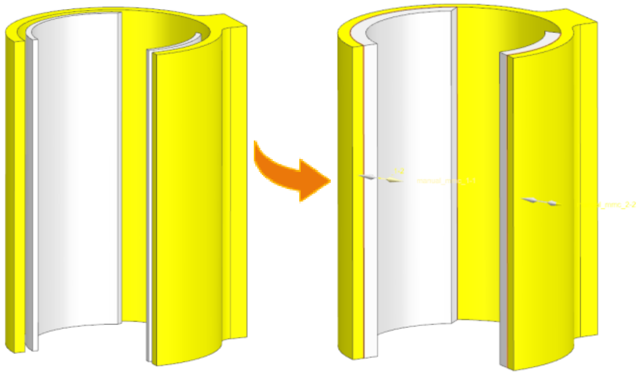




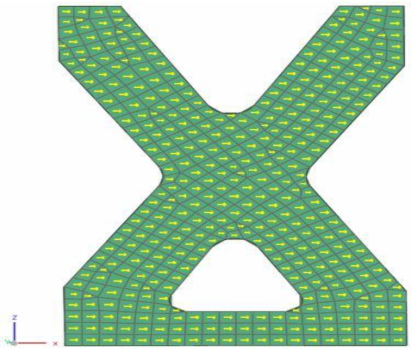
SIEMENS

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.



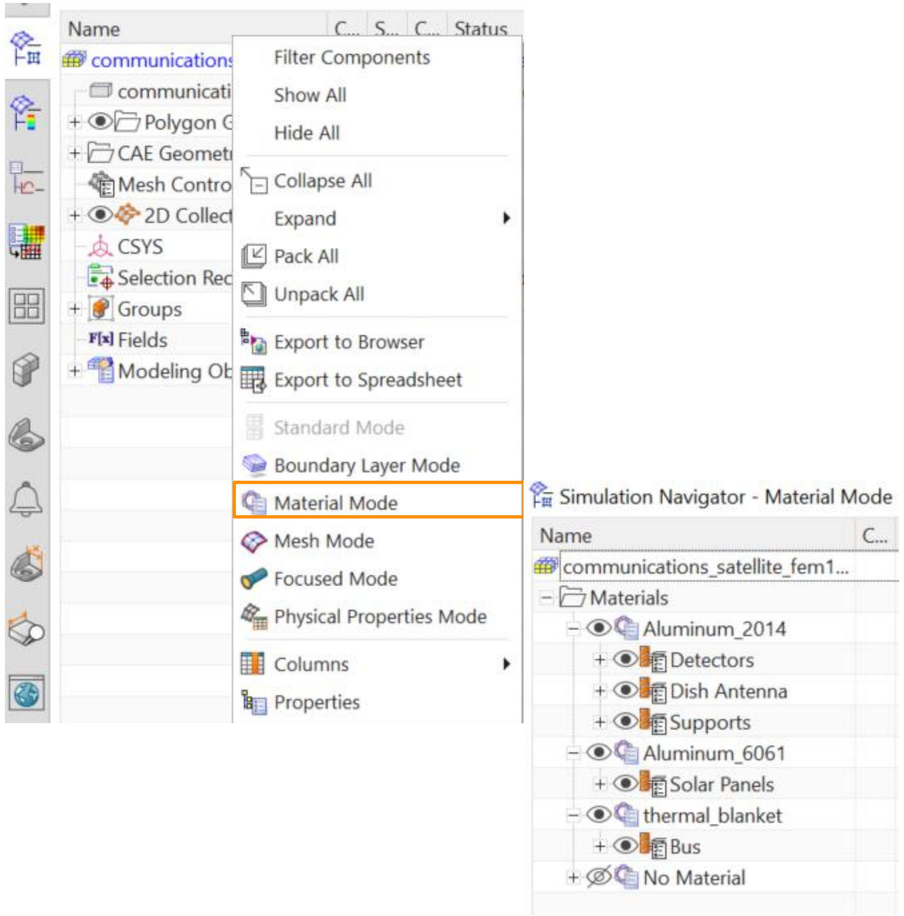
- Display the material orientation of 2D or 3D elements in your model using the **Element Material Orientation** command.



- Check for duplicate bodies or faces.

Multi-layer insulation

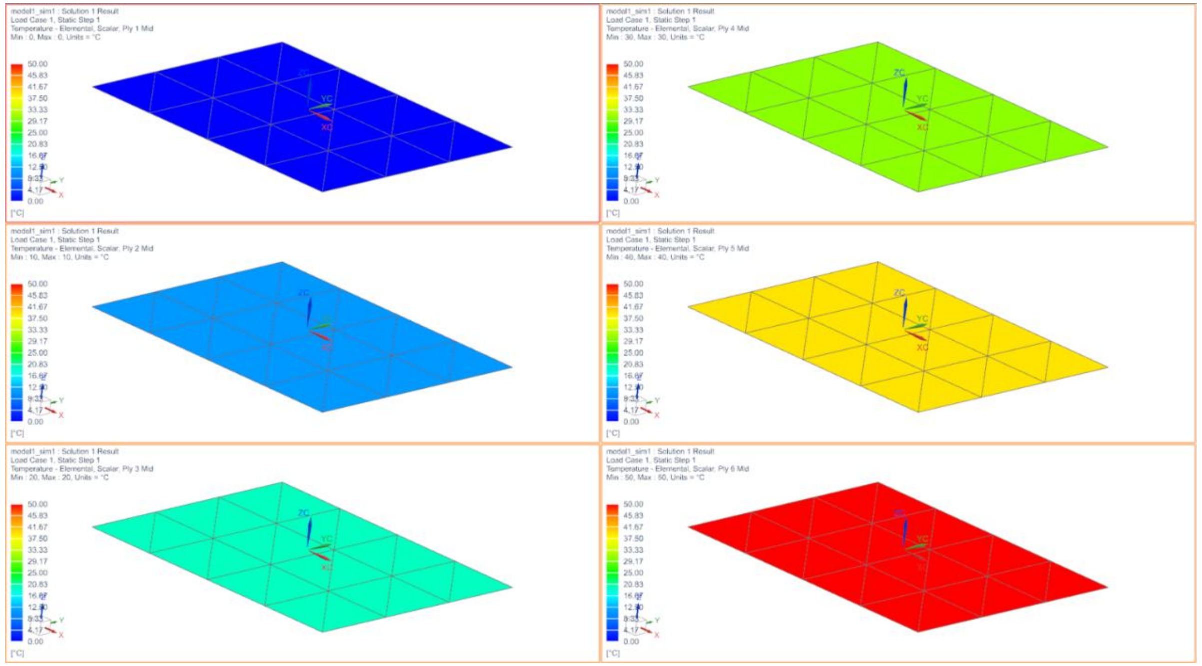
- Check MLI setup:
  - Ensure your normal directions are correct.
  - Check if the MLI shells properties are setup correctly in the mesh collectors.
  - Show where thermal-optical properties are applied using the **Inspect Thermo-Optical Properties** command or the **Thermo-Optical Color Display**.
  - 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*)



SIEMENS

Multi-layer insulation post-processing

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

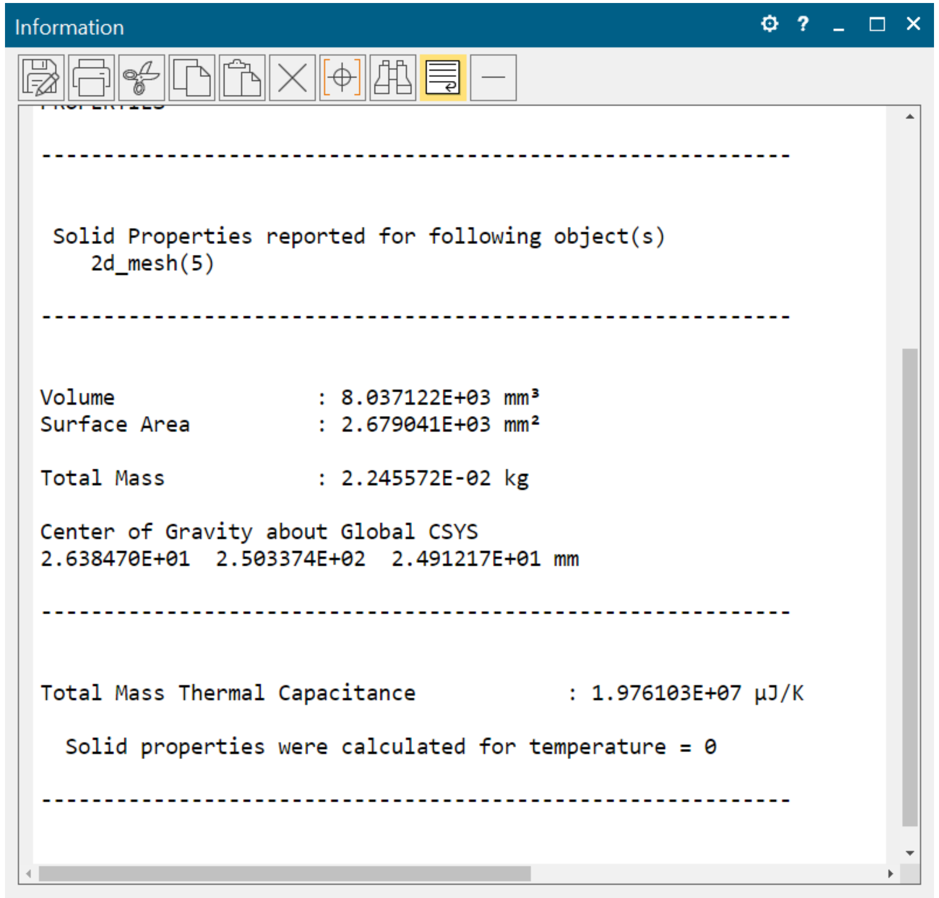


Mass

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

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


Information

```
-----------------------------------------------------------

 Solid Properties reported for following object(s)
    2d_mesh(5)

-----------------------------------------------------------

Volume                 : 8.037122E+03 mm³
Surface Area           : 2.679041E+03 mm²

Total Mass             : 2.245572E-02 kg

Center of Gravity about Global CSYS
2.638470E+01  2.503374E+02  2.491217E+01 mm

-----------------------------------------------------------

Total Mass Thermal Capacitance          : 1.976103E+07 µJ/K

  Solid properties were calculated for temperature = 0

-----------------------------------------------------------
```


SIEMENS

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
```

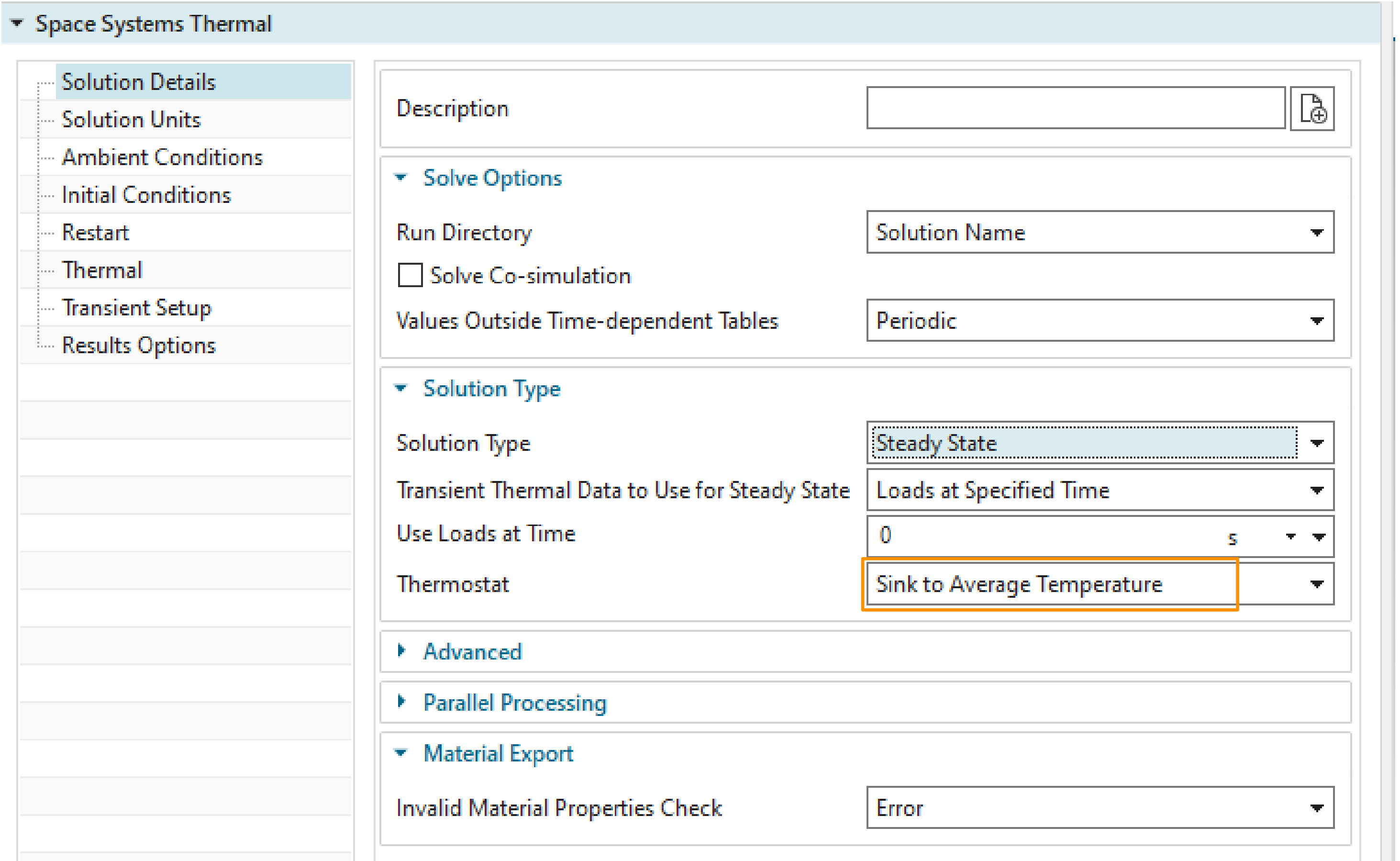
ThermostatReport.csv

	A	B	C	D	E	F	G	H	I
1	Time	Thermostat ID/Nam	On/Off	Temp Sensor	Power	Numbers	Total Time	Total Ener	P
2	1.0000000E+00	diode_h	1	2.0000000E+0	1.2000002	1	1.000000	1.200000	3.960000
3	1.0000000E+00	diode_e	1	2.0000000E+0	1.1999999	1	1.000000	1.199999	3.959999
4	1.0000000E+00	diode_t	1	2.0000000E+0	1.1999998	1	1.000000	1.199999	3.959999
5	1.0000000E+00	diode__	1	2.0000000E+0	1.1999998	1	1.000000	1.199999	3.959999
6	1.0000000E+00	diode_l	1	2.0000000E+0	1.2000000	1	1.000000	1.200000	3.960000

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

Heaters and heat loads

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



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 sim and FEM files.
- Expressions can be accessed in a tabular format by hitting Ctrl+E. These expressions can be updated from an external file or linked to Excel.
- You can also use **Parameter Tables** to manage multiple expressions at once.

Named Points

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

Orbits

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

Solid motion effects

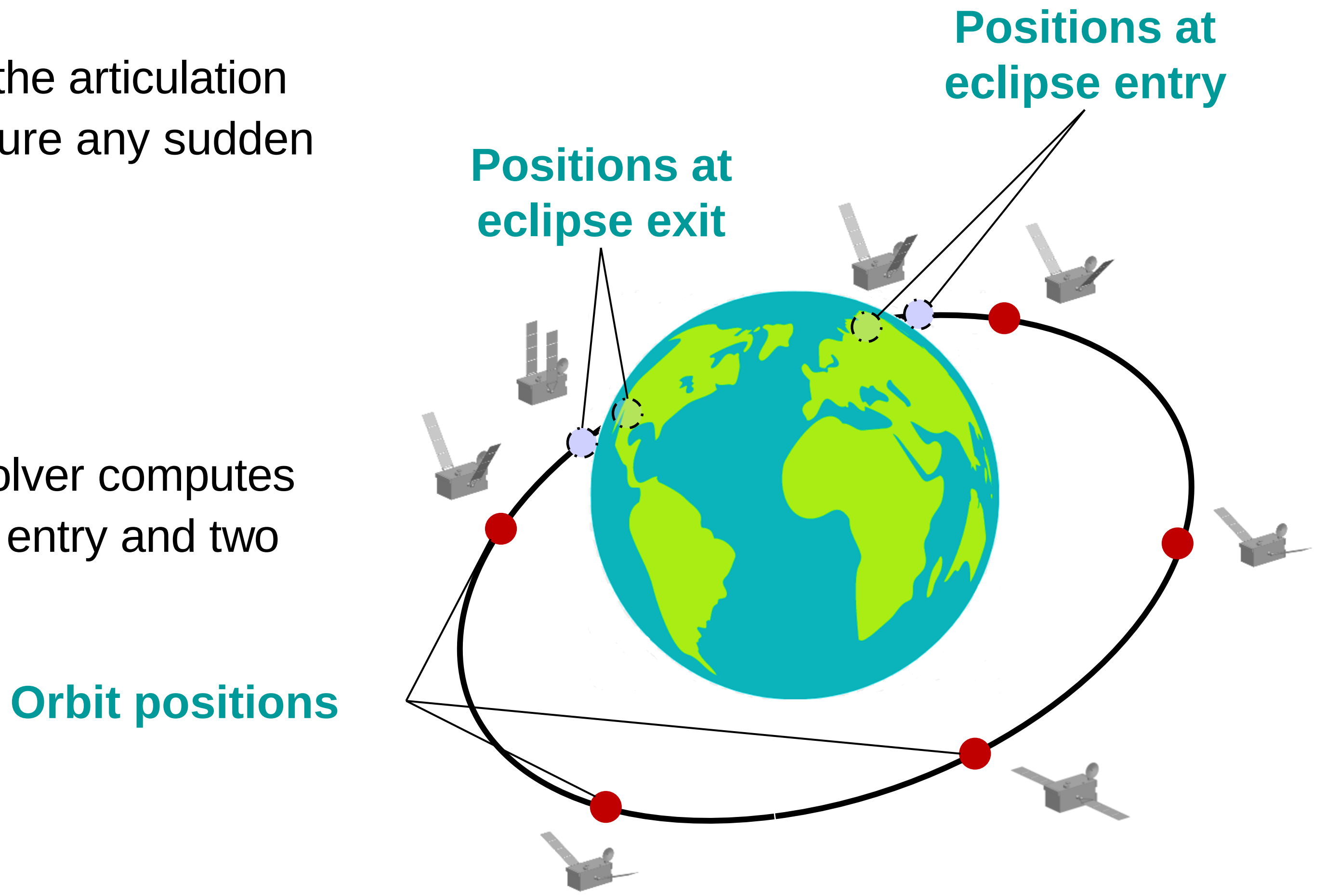
- Check articulation setup using **Orbit Visualizer**.
- Check articulation setup in post-processing using displacement results.

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.

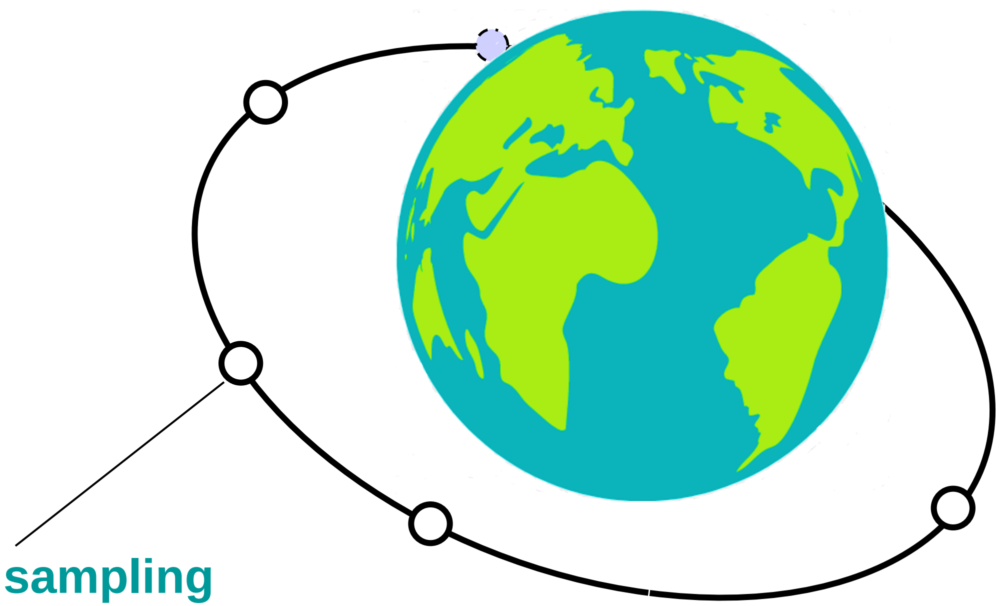
The number of orbit time steps depends on:

- 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.

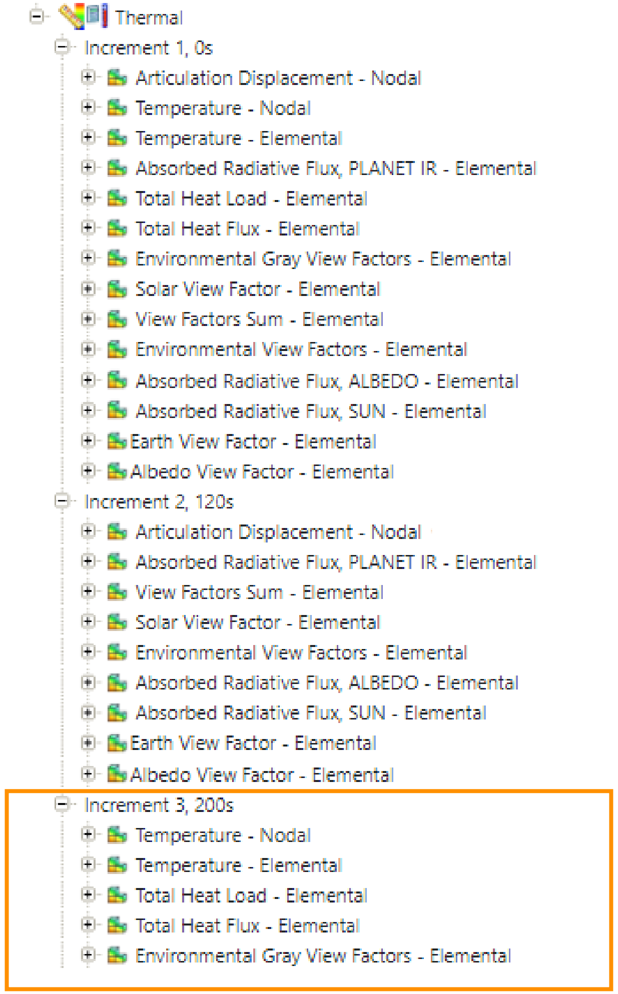


If you have the articulation rates defined using a table versus time, the solver adds the table times as extra time steps.

Post-processing articulation in the presence of an orbit

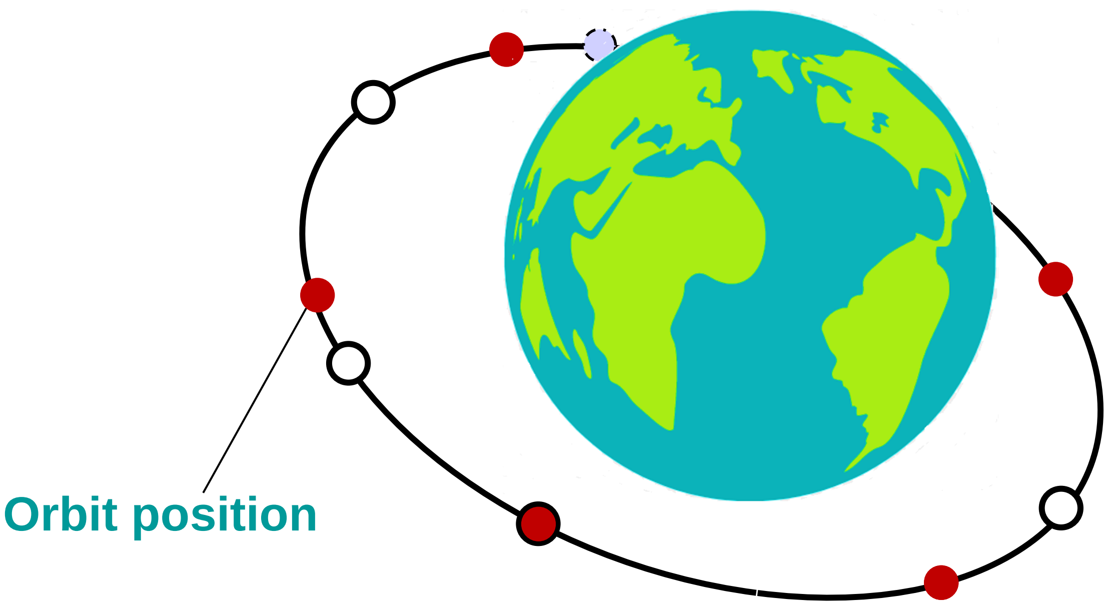


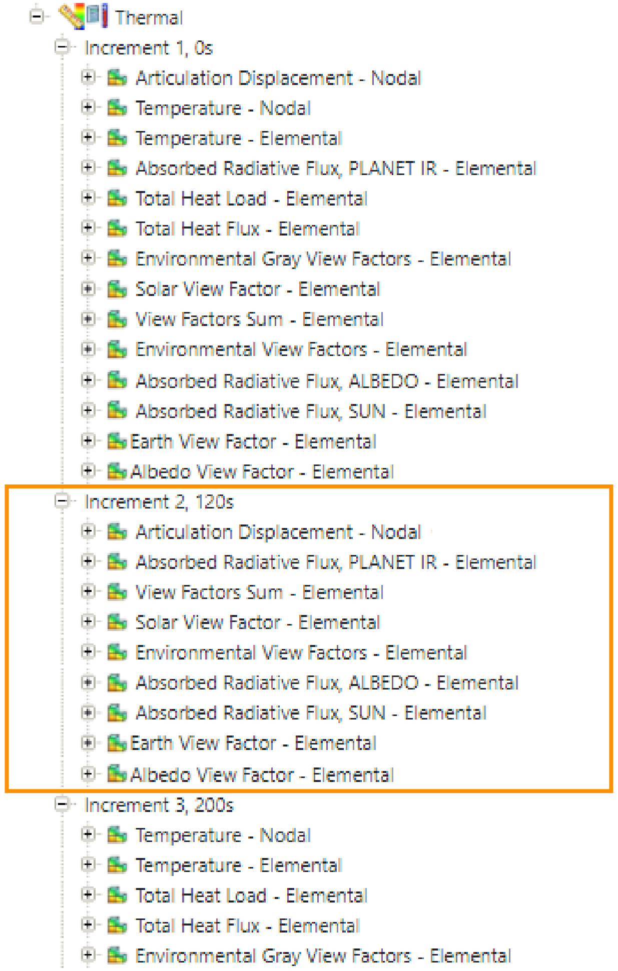
Results sampling point



SIEMENS

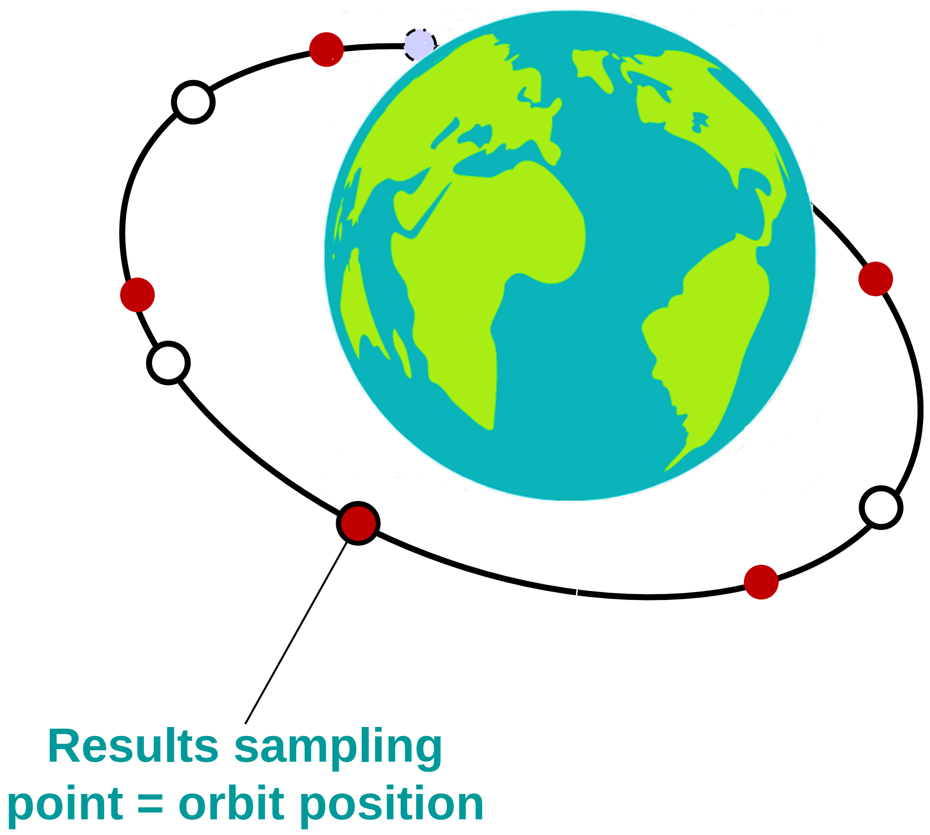
Post-processing articulation in the presence of an orbit





SIEMENS

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

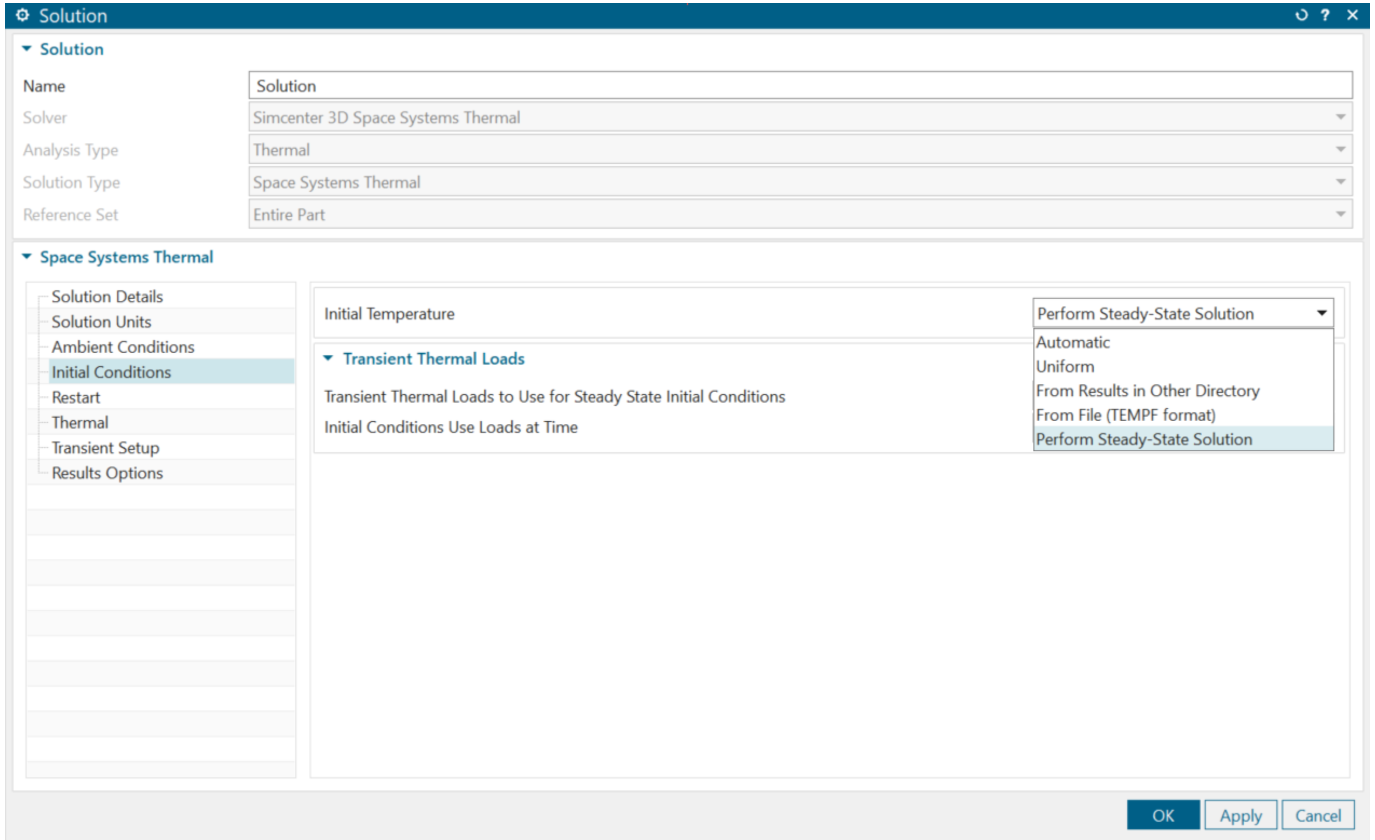
SIEMENS

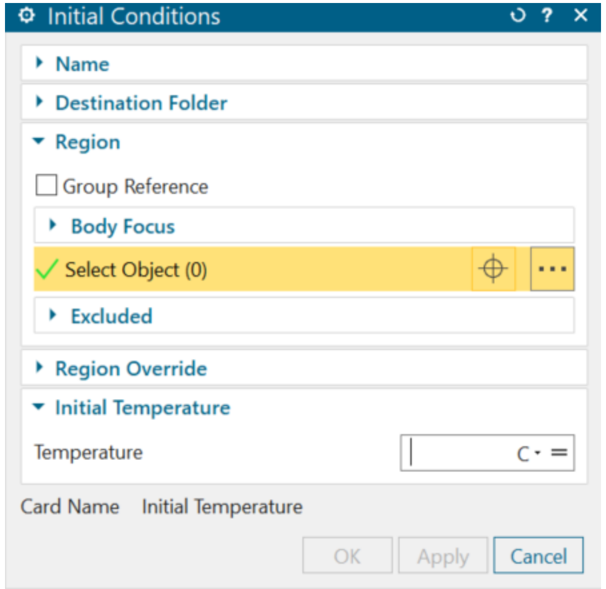
Solution settings

- 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.

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.





SIEMENS

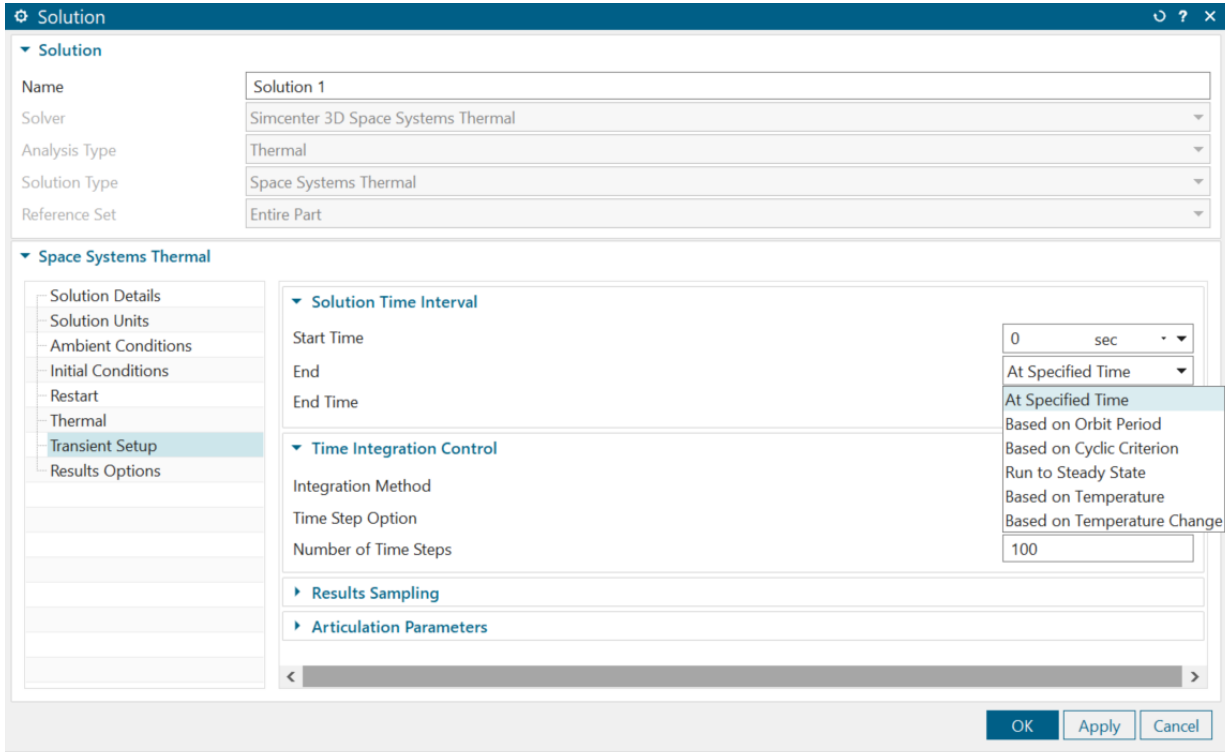
Solution settings – Transient Setup

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.

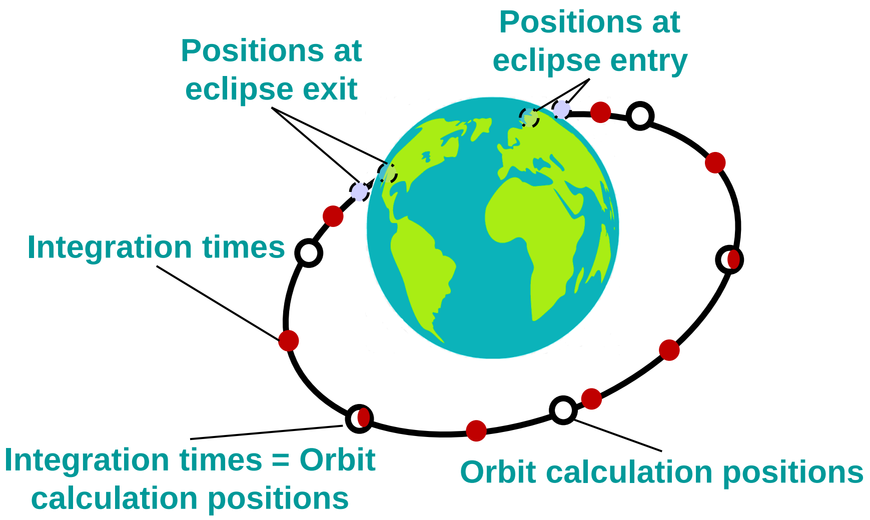


SIEMENS

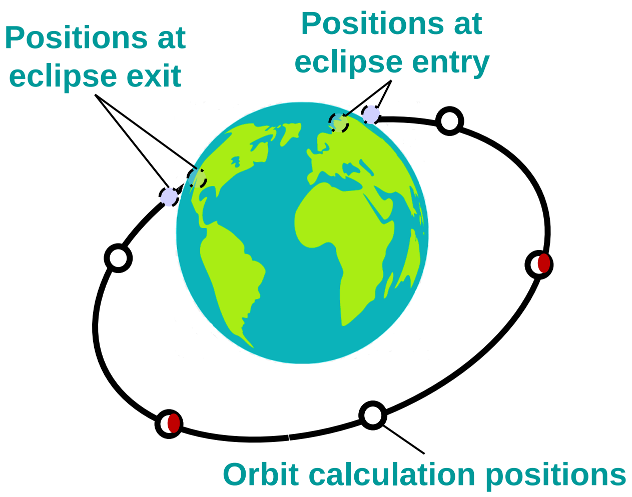
Understanding the integration time steps in the presence of an orbit

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.

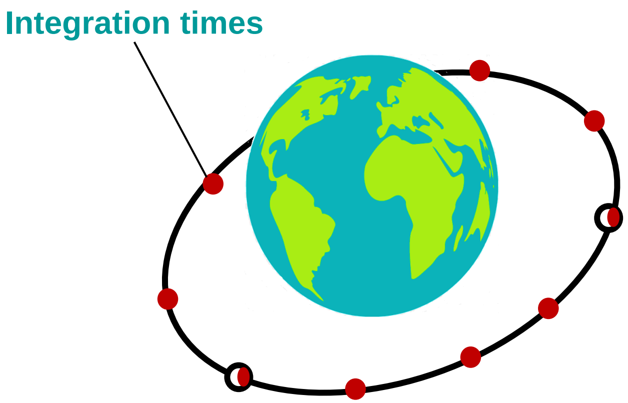
To not enforce integration at orbit times, you can use the **INTEGRATION ORBIT TIME** advanced parameter, which is located under **Thermal→Thermal Solver** in the catalog. The thermal solver uses the requested integration times without including the orbit calculation positions and the entry and exit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



Orbit and eclipse calculation positions, and requested integration time steps



Integration time steps in the presence of an orbit



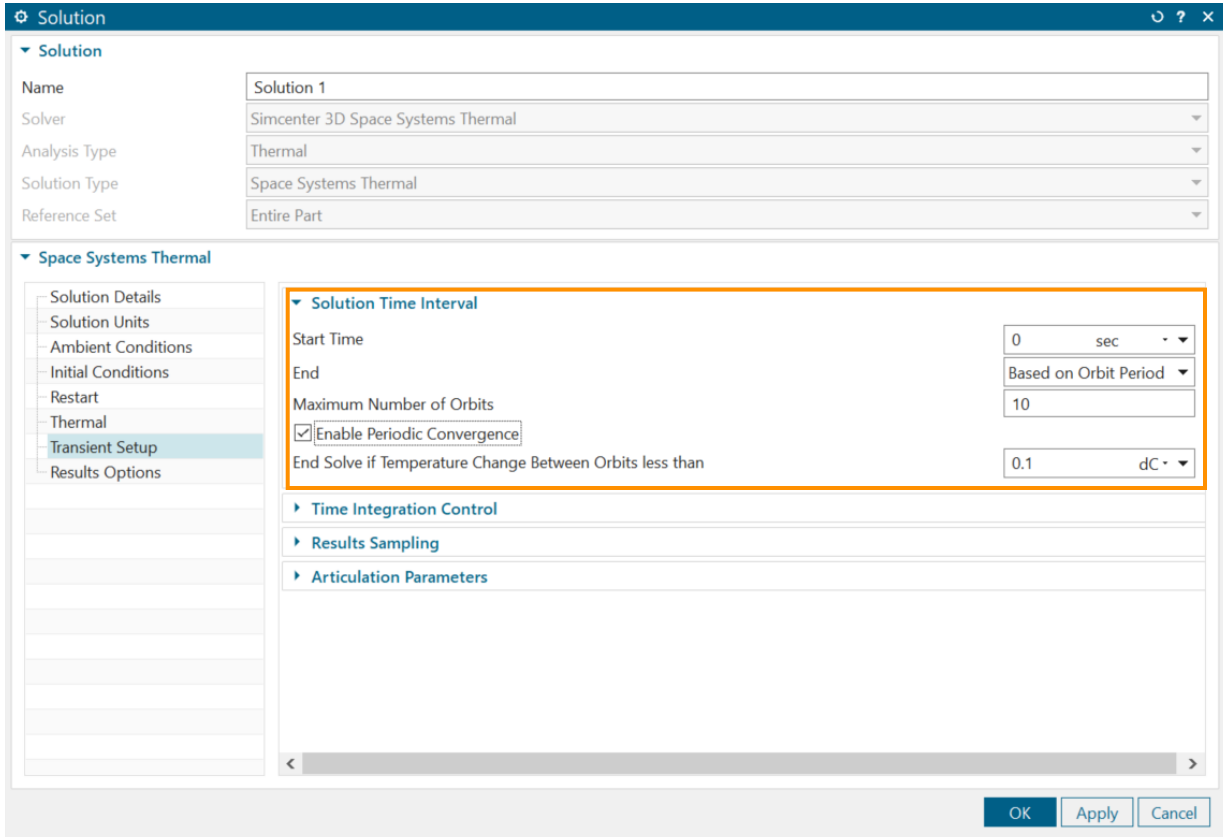
Integration time steps in the presence of an orbit with advanced parameter

SIEMENS

Check Orbital solution convergence

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.



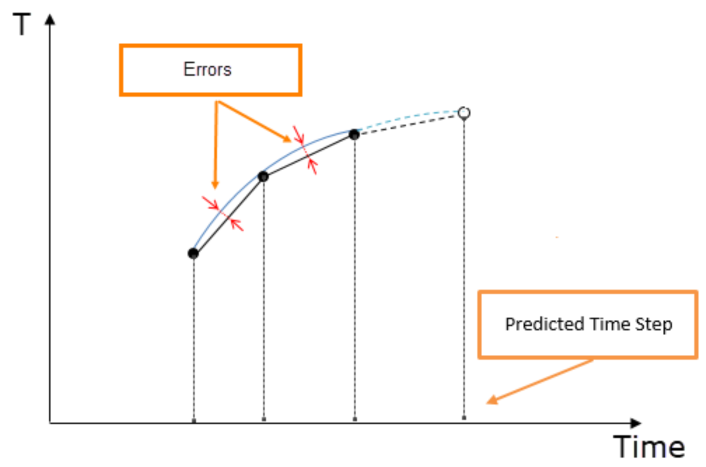
Automatic time step

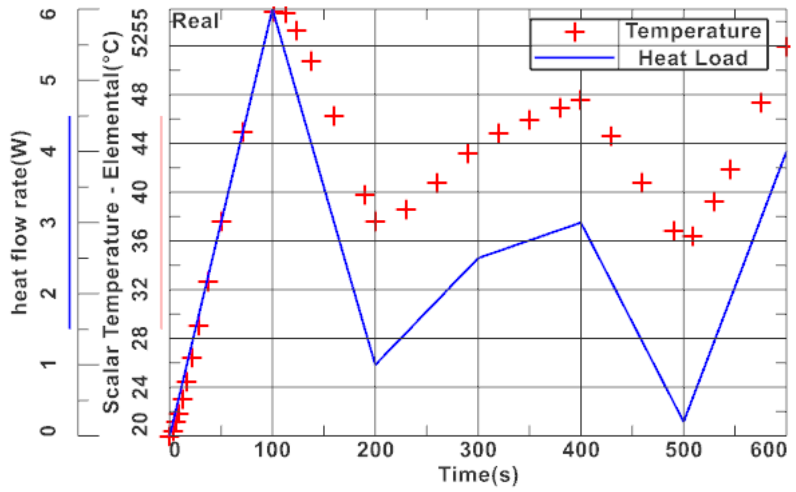
One way to speed up a solve is to use the automatic time stepping.

The automatic time step size calculation is based on the estimated error between a quadratic fit and a linear fit a linear variation 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.



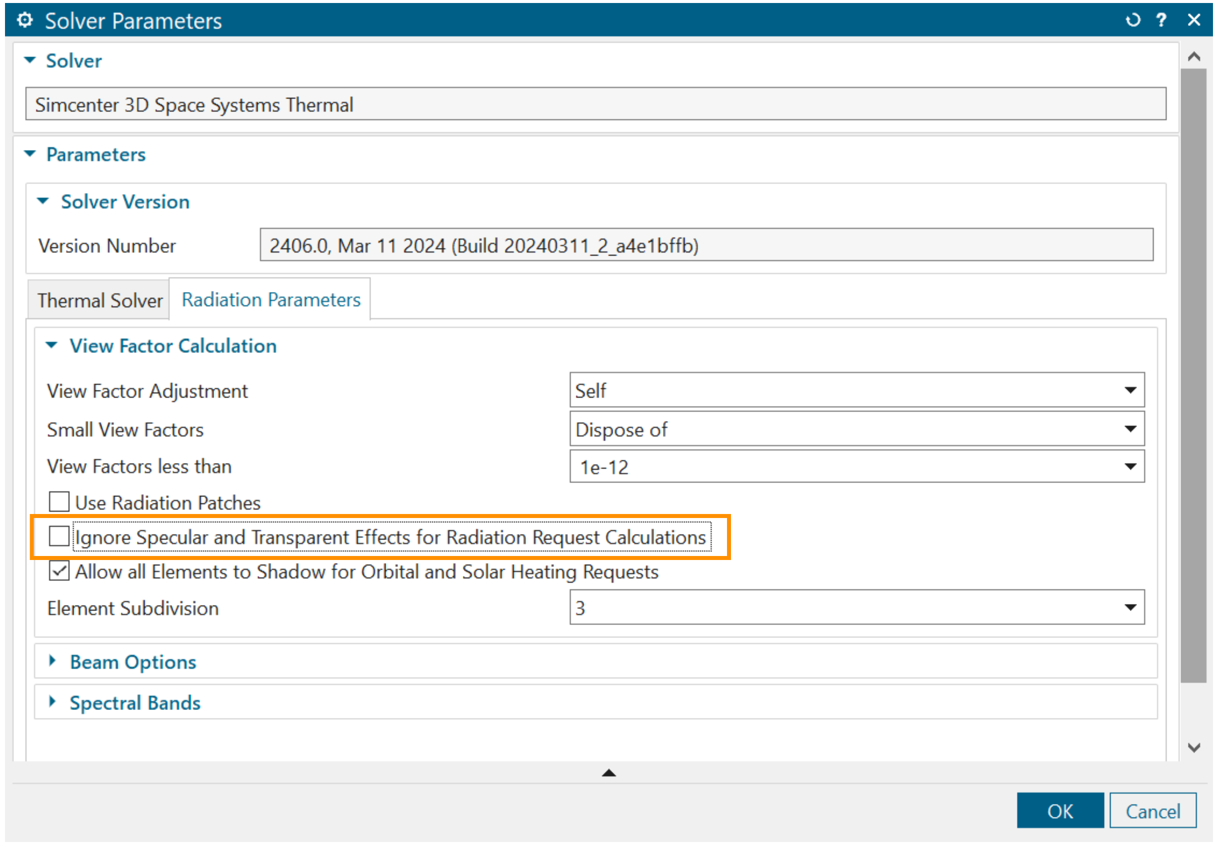


SIEMENS

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.



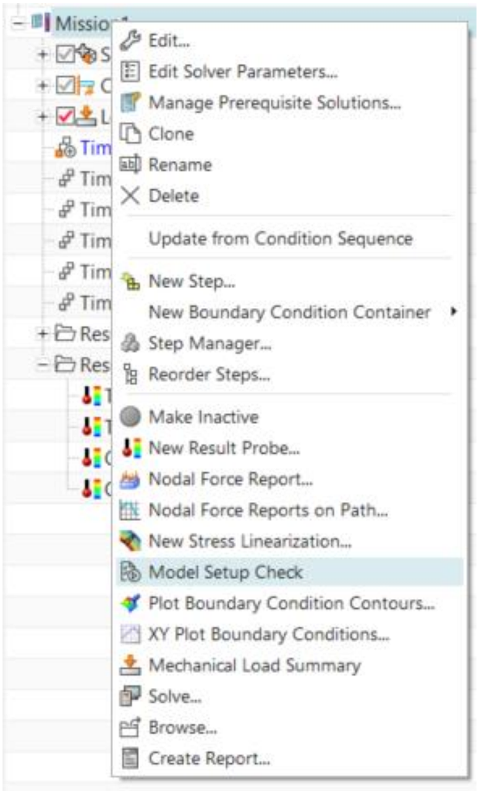
SIEMENS

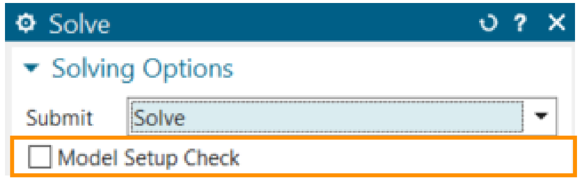
Model Setup Check

Run model setup check on solution by right-clicking **Solution** and selecting **Model Setup Check** or selecting the **Model Setup Check** ckeck box 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





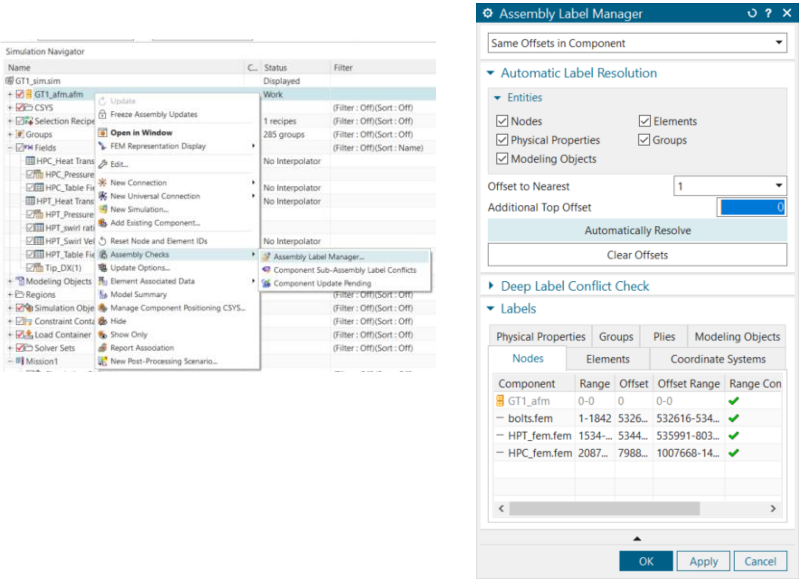
SIEMENS

ID conflicts

AFM Label Conflicts

With AFM Active:

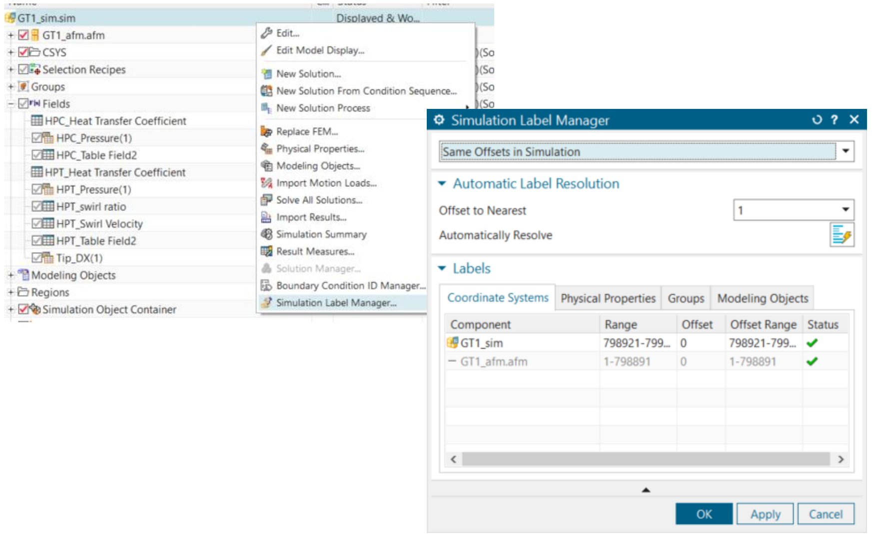
Right-click afm file → **Assembly Checks** → **Assembly Label Manager**



Simulation Label Conflicts

With Simulation Active:

Right-click Simulation file → **Simulation Label Manager**



SIEMENS

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).

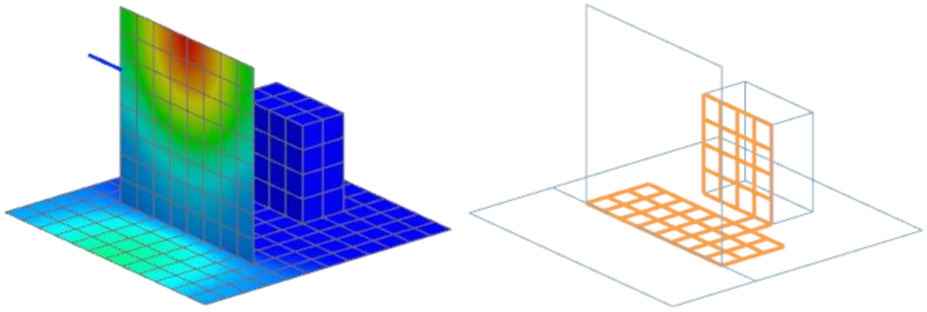
```
Time=  10000.0000000     Integration timestep=  900.000

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
```


SIEMENS

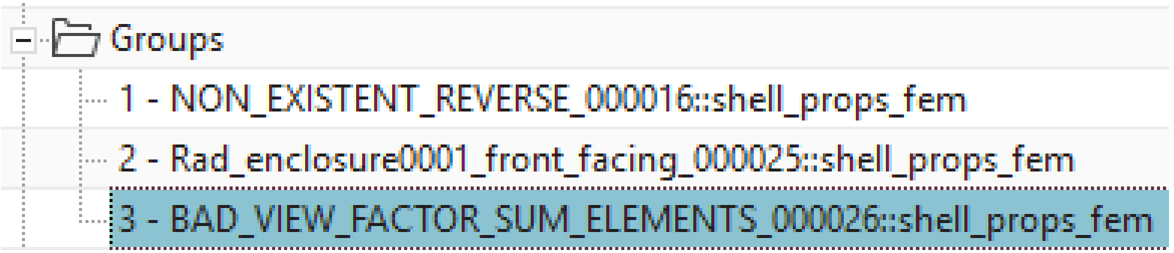
Resolving a warning

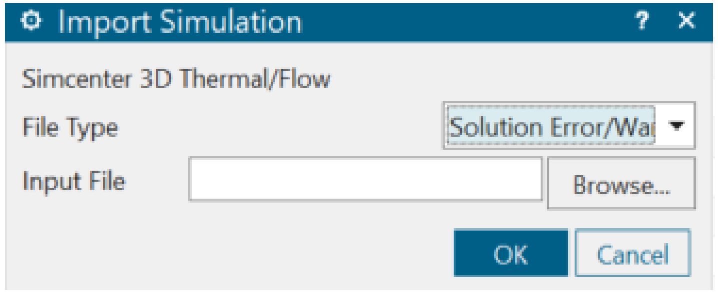
Review the messages in the Solution Monitor or the log file for the thermal and flow solvers:

```
**  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**.





SIEMENS

Results

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

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.

Useful tool – Thermal Coupling Report

- 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.

G13

Thermal Coupling List File: D:\users\Test.txt

Read File

	Name	Description	Type	Value
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
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

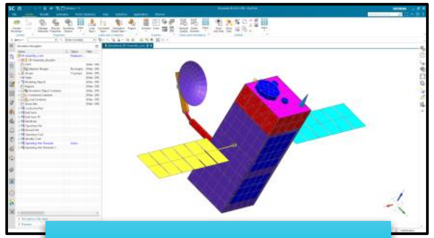
See the ThermalCoupling.xlsm file linked to this Knowledge Base Article.

TMG Correlation

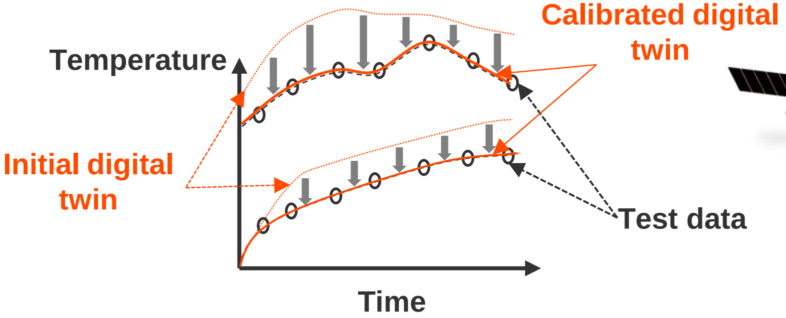
Quickly produce an accurate and reliable digital twin

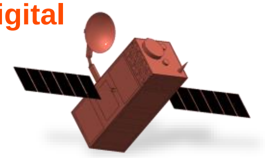
Build confidence, save time, and innovate faster

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



Digital Twin → TMG Correlation ← Test data





Compare 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.