# 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.

Solution		৩ <b>?</b>	x
▼ Solution			
Name	Solution 3		ור
Solver	Simcenter 3D Space	e Systems Thermal	
Analysis Type	Thermal		
Solution Type	Space Systems The	rmal	
Reference Set	Entire Part		
Snace Systems Therma	1		-
Space Systems merina	•		
Solution Details	<ul> <li>Solution Units</li> </ul>	Í Í	
- Ambient Conditions	Solve-Time Units	Current Part 🔹	
- Initial Conditions	The line is the later of the line of the line of the later of the late		
Restart	Units information		
- Thermal	Mass	lbf-s^2/in	
Transient Setup	Length	in	
Results Options	Time	second	
	Power	lbf-in/s	
	Heat Flux	lbf/in-s	
	Energy	Ibf-in	
	Velocity	in/s	
	Pressure	lbf/in^2	
	Viscosity	lbf-s/in^2	
	Density	lbf-s^2/in^4	
< >>	Specific Heat	in^2/s^2-F	
-	-		



Apply || Cancel

# **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.

Library Materials						
Libraries						
<ul> <li>Materials</li> </ul>						
Name	Category	Туре	Mass Density (	Thermal Conductivity (K)	Specific Heat (CP)	
ABS	PLASTIC	Isotropic	1.05e-06 kg/m	170 microW/mm-dC	180000000 mi	^
ABS-GF	PLASTIC	Isotropic	1.05e-06 kg/m	170 microW/mm-dC	180000000 mi	
Acetylene_C2H2_Gas	OTHER	Fluid	Tabular Data:	Tabular Data:	Tabular Data:	
Acetylene_C2H2_Liquid	OTHER	Fluid	Tabular Data:	Tabular Data:	Tabular Data:	
Acrylic	PLASTIC	Isotropic	1.2e-06 kg/mm	215.62 microW/mm-dC	150000000 mi	
Air	OTHER	Fluid	1.2041 kg/m^3	26.3 microW/mm-dC	1007000000 mi	
Air_Temp-dependent_Gas	OTHER	Fluid	Tabular Data:	Tabular Data:	Tabular Data:	
AISI_310_SS	METAL	Isotropic	7.92781e-06 kg	Tabular Data:	50000000 micr	
AISI_410_SS	METAL	Isotropic	7.73377e-06 kg	Tabular Data:	46000000 micr	
AISI_SS_304-Annealed	METAL	Isotropic	7.9e-06 kg/mm	16300 microW/mm-dC	50000000 micr	
AISI_Steel_1005	METAL	Isotropic	7.872e-06 kg/	56000 microW/mm-dC	481000000 micr	
AISI_Steel_1008-HR	METAL	Isotropic	7.872e-06 kg/	65200 microW/mm-dC	481000000 micr	
AISI_Steel_4340	METAL	Isotropic	7.85e-06 kg/m	44500 microW/mm-dC	475000000 micr	
AISI_Steel_Maraging	METAL	Isotropic	8e-06 kg/mm^3	25300 microW/mm-dC	476000000 micr	
AL	A APTAL	testinate.	2 704. OC 1/	T.L. I. D.A.	000000001	*

r Import/Export	-		
E B P	<b>1</b>		
Selection for Ex	port		
Name	Label	Туре	
Painted Steel	1	Thermo-Optical Properties	
Steel	2	Thermo-Optical Properties	
Pyrex	3	Thermo-Optical Properties	
Filament	4	Thermo-Optical Properties	
Polished Alumin	5	Thermo-Optical Properties	
Polished Alumin	6	Thermo-Optical Properties	
Select All			
Resolve Name O	Conflict		
Overwrite Value	es for Existing	g Entities	
Rename Impor	ted Thermo-	Optical Properties	



# 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.



#### **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**.







# **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.



Element Quality		<u> </u>
✓ Elements to Check		
Selected	•	
关 Select Object (0)		<b>↔</b> …
Element Labels		
Limits Check		
General Geometry Checks		
<ul> <li>Solver Specific Geometry Checks</li> </ul>		
Use Element Type Specific Value	S	
Select All Deselect All		
Lacobian Batio >	20	_
lacobian 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 Elem	ents	
Correct Failed E	lements	
		Close



#### **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





# **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**.

#### Envi Name 🖽 communications sa Defa 🌺 Show Only communicati Defa 🗇 communicatio 🔑 Edit... + 🗸 🗁 Polygon G 🧹 Edit Display... Off)(Sort : Off) Off)(Sort : Off) CAE Geometr 🕼 Edit Mesh Associated Data... Mesh Cont $= \checkmark$ 2D Collect $\times$ Delete Off)(Sort : Off) ✓ ■ Bus Check All Element Shapes V 률 Detecto 🕂 🗸 🛱 Dish Ar 🔛 Plot Thickness Contours 🎉 Element Normals 🗄 🗸 🌆 Solar Pa (i) Information Element Outlines 🖃 🗸 🎼 Suppor 🐘 Solver Syntax Preview Ouplicate Elements √ 🌮 2d mesr ୶ Duplicate Nodes F[x] Fields (Filter 🛕 Material Orientation CSYS (Filter

Simulation Navigator



#### **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



Model Display		ა? X
▼ Parameters		
<ul> <li>Model Colors</li> <li>Nodes</li> <li>Elements</li> <li>Polygon Edges</li> <li>FEM Annotations</li> <li>Universal Connections</li> </ul>	<ul> <li>Display</li> <li>Color Basis</li> <li>No Thermo-Optical Property Color</li> <li>Black Body Color</li> <li>2d Mesh Display</li> <li>Display Legend</li> <li>Set Mesh Col</li> </ul>	Thermo-Optical Prog
Resolve Mesh Display Cont	tention	
	ОК	Apply Cancel





# **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 thermooptical 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.

စံ Inspect Thermo-Optical Properties ပ ? ×	Information 🗘 ? _ 🗆 ×
<ul> <li>▼ Filter</li> <li>● ALL</li> <li>○ BY MESH/ELEMENTS</li> </ul>	Properties for Modeling Object Aluminum:
Modeling Objects State      Modeling Objects      Solar Cells - COMSAT      Black Paint      Aluminum      Vapor-deposited coating - Silver      blanket	Emissivity: 0.07 Properties for Modeling Object Black Paint: Infrared Emissivity : 0.87 Infrared Transmissivity : 0 Solar Absorptivity : 0.95 Solar Specular Reflectivity : 0 Solar Transmissivity : 0 Index of Refraction : 1 Properties for Modeling Object blanket: Infrared Emissivity : 0.016 Infrared Emissivity : 0.016 Infrared Transmissivity : 0 Solar Absorptivity : 0.003 Solar Specular Reflectivity : 0 Solar Specular Reflectivity : 0 Infrared Transmissivity : 0 Solar Absorptivity : 0 Solar Specular Reflectivity : 0 Index of Refraction : 1
Show Elements Normal Delete Unused Optical Properties	Properties for Modeling Object Solar Cells - COMSAT: Infrared Emissivity : 0.85 Infrared Specular Reflectivity: 0 Infrared Transmissivity : 0 Solar Absorptivity : 0.82 Solar Specular Reflectivity : 0 Solar Transmissivity : 0 Index of Refraction : 1

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 Coupling	0 ?
• Туре	
• Name	
Destination Folder	
Primary Region	
Secondary Region	
Secondary Region Override	
Swap Regions	
Region Side Specification	
<ul> <li>Magnitude</li> </ul>	
Туре	Heat Transfer Coefficient 🔹
Coefficient	10 W/mm^2-dC · =
<ul> <li>Coupling Method</li> </ul>	
Method	Projective Intersection
<ul> <li>Additional Parameters</li> </ul>	
Overlap Projection Direction	Primary Element Normals 🔻
Temperature Dependence Uses	Primary Temperature 🔹
<ul> <li>Ancillary Display</li> </ul>	
Show Ancillary Display	
C	
<ul> <li>Ancillary Display Options</li> </ul>	
Card Name Thermal Coupling	
	OK Const



# **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.



6.500

4.500

4.000

3.500

2.500

1.500

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.



Parameters Model Colors Nodes Elements Polygon Edges FEM Annotations Universal Connections Thickness 0.35 mm Line Style Endpoint Marker Stitched Edges Color Thickness 0.35 mm Line Style Endpoint Marker Stitched Edges Color Thickness 0.35 mm Endpoint Marker Stitched Edges Color Thickness 0.35 mm Color Thickness 0.35 mm Color Thickness 0.35 mm Line Style Endpoint Marker Stitched Edges Color Thickness 0.35 mm Line Style Endpoint Marker Color Thickness 0.35 mm Color Thickness 0.35 mm Line Style Endpoint Marker Color Thickness 0.35 mm Line Style Endpoint Marker Color Thickness 0.35 mm Line Style Color Thickness 0.35 mm Line Style Color Thickness 0.35 mm Color Thickness 0.35 mm Line Style Color Thickness 0.35 mm Line Style Color Thickness 0.35 mm Line Style Color Thickness 0.35 mm Line Style Color Thickness 0.35 mm Line Style Color	Model Display	ა? x
Model Colors          Nodes          Elements          Polygon Edges       Color         FEM Annotations          Universal Connections          Universal Connections          Visit Connections          Image: Color          Visit Connections          Image: Color          Visit Ched Edges          Display Stitched Edges          Color          Image: Color	Parameters	
Resolve Mesh Display Contention           OK         Apply         Cancel	Model Colors Nodes Elements Polygon Edges FEM Annotations Universal Connections	<ul> <li>Free Edges</li> <li>✓ Display Free Edges</li> <li>Color</li> <li>Thickness</li> <li>0.35 mm ✓</li> <li>Line Style</li> <li>✓ Stitched Edges</li> <li>✓ Display Stitched Edges</li> <li>✓ Color</li> <li>✓ Thickness</li> <li>0.35 mm ✓</li> <li>Line Style</li> <li>✓ Ine Style</li> <li>✓ Endpoint Marker</li> <li>✓ Ine Style</li> <li>✓ ✓</li> </ul>
OK Apply Cancel	Resolve Mesh Display Contention	
		OK Apply Cancel



## **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: <u>Multi-layer insulation modeling in Simcenter 3D Thermal</u> (<u>https://support.sw.siemens.com/knowledge-base/KB000122047\_EN\_US</u>)



# **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	at	Minimum	at	Average	Total	Total	Total	
	Temp	element	Temp	element	Temp	Heat in	Capacitance	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-	-Apayloa	d-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	



#### **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.

_	
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	

INPF

#### ThermostatReport.csv

	A	В	С	D	E	F	G	н	1
1	Time	Thermostat ID/Name	0n/Off	Temp Sensor	Power	Numbers	Total Time	Total Ener	Р
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	1.1999999	3.959999
4	1.0000000E+00	diode_t	1	2.0000000E+0	1.1999998	1	1.000000	1.1999998	3.959999
5	1.0000000E+00	diode	1	2.0000000E+0	1.1999998	1	1.000000	1.1999998	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:
 Space Systems Thermal

Ð
-
•
•
-
•
•
•



# **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  $\rightarrow$  Utilities  $\rightarrow$  Customer Defaults  $\rightarrow$  Pre/Post  $\rightarrow$  Expressions  $\rightarrow$  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.

**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





#### SIEMENS

#### Post-processing articulation in the presence of an orbit





#### SIEMENS

### Post-processing articulation in the presence of an orbit







# **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.

Solution		৩ ?
Solution		
Name	Solution	
Solver	Simcenter 3D Space Systems Thermal	
Analysis Type	Thermal	
olution Type	Space Systems Thermal	
Reference Set	Entire Part	
Space Systems Ther	mal	
Solution Details	Initial Temperature	Perform Steady-State Solution
- Solution Units - Ambient Conditions - Initial Conditions	s Transient Thermal Loads	Automatic Uniform
Restart	Transient Thermal Loads to Use for Steady State Initial Conditions	From Results in Other Directory
Transient Setup	Initial Conditions Use Loads at Time	Perform Steady-State Solution
Results Options		
		OK Apply Cap

Initial Conditions	ა? ×
▶ Name	
Destination Folder	
▼ Region	
Group Reference	
Body Focus	
✓ Select Object (0)	<b>•</b> •••
Excluded	
Region Override	
<ul> <li>Initial Temperature</li> </ul>	
Temperature	C·=
Card Name Initial Temperature	
0	K Apply Cancel



### **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.

Name	Solution 1	
Solver	Simcenter 3D Space Systems Thermal	
Analysis Type	Thermal	
Solution Type	Space Systems Thermal	
Reference Set	Entire Part	
<ul> <li>Space Systems Thermal</li> </ul>		
Solution Details	<ul> <li>Solution Time Interval</li> </ul>	
- Solution Units	Start Time	0
- Ambient Conditions	Start Time	o sec t
Initial Conditions	End	At Specified Time
Restart	End Time	At Specified Time
- Thermal		Based on Orbit Period
Transient Setup	<ul> <li>Time Integration Control</li> </ul>	Based on Cyclic Criterion
Results Options	Integration Method	Run to Steady State
		Based on Temperature
	Time Step Option	Based on Temperature C
	Number of Time Steps	100
	Results Sampling	
	Articulation Parameters	



#### 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



#### **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.

Solution		ა?
<ul> <li>Solution</li> </ul>		
Name	Solution 1	
Solver	Simcenter 3D Space Systems Thermal	
Analysis Type	Thermal	
Solution Type	Space Systems Thermal	
Reference Set	Entire Part	
<ul> <li>Space Systems Thermal</li> </ul>		
Solution Details	<ul> <li>Solution Time Interval</li> </ul>	
Solution Units	Start Time	0
Ambient Conditions	State Time	o sec
- Initial Conditions	End	Based on Orbit Period 🔻
Thermal	Maximum Number of Orbits	10
Transient Setun	Enable Periodic Convergence	
Results Options	End Solve if Temperature Change Between Orbits less than	0.1 dC
	Time Integration Control	
	Results Sampling	
	Articulation Parameters	
	· Alticulation Parameters	
	<	



#### 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.

Solver Parameters	ు ? x
▼ Solver	^
Simcenter 3D Space Systems Thermal	
▼ Parameters	
▼ Solver Version	
Version Number 2406.0, Mar 11 2024 (Build 202403	311_2_a4e1bffb)
Thermal Solver Radiation Parameters	
▼ View Factor Calculation	
View Factor Adjustment	Self
Small View Factors	Dispose of 🔹
View Factors less than	1e-12 💌
Use Radiation Patches	
Ignore Specular and Transparent Effects for Radiation Req	uest Calculations
Allow all Elements to Shadow for Orbital and Solar Heatin	g Requests
Element Subdivision	3
Beam Options	
<ul> <li>Spectral Bands</li> </ul>	
	~
	OK Cancel



#### **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



Solve	ა? ×						
✓ Solving Options							
Submit	Solve						
Model Setup Check							

#### SIEMENS

#### **ID conflicts**

#### **AFM Label Conflicts**

With AFM Active:

# Right-click afm file $\rightarrow$ Assembly Checks $\rightarrow$ Assembly Label Manager

					Assembly Lat		anag	51
Simulation Navigator	1				Same Offsets in C	ompor	nent	
Name		c.	Status	Filter				1
個GT1_sim.sim			Displayed		<ul> <li>Automatic La</li> </ul>	idel K	esolu	U
+ C = GT1_atm.atm + CSYS + CSYS + CSYS + Coups - PM Fields - HPC_Heat Trans - HPC_Table Field - HPT_Heat Trans	Volate     Preze Assembly Updates     Freeze Assembly Updates     Pown in Window     FEM Representation Dapkay     // Edit.     New Connection     New Universal Connection     New One Internet		Work 1 recipes 285 groups No Interpolator No Interpolator No Interpolator	(Filter : Off)(Sort : Off) (Filter : Off)(Sort : Off) (Filter : Off)(Sort : Off) (Filter : Off)(Sort : Name)	Entities     Nodes     Physical Prop     Modeling Ob Offset to Nearest Additional Top Of	erties jects fset		6
HPT_Pressure	Add Existing Component				riduitional top of	Auto	matic	a
HPT_Swirl Vel	Reset Node and Element IDs		No Interpolator					-
PILI HPT_table he	B) Assembly Checks		Assembly Label	Manager			Clear	0
+ Modeling Objects	Update Options  Bernent Associated Data  Model Summary	•	Component Sult	a-Assembly Label Conflicts date Pending	Deep Label C	Conflic	t Che	20
+ 2 Simulation Obje	<ul> <li>Manage Component Positioning CSYS.</li> <li>Hide</li> </ul>			(Filter : Off)(Sort : Off) (Filter : Off)(Sort : Off) (Filter : Off)/Sort : Off)	<ul> <li>Labels</li> </ul>			
+ Load Container	Show Only			(Filter : Off)(Sort : Off)	Physical Properti	ies G	rouns	
+ Colver Sets	# Report Association			(Filter : Off)(Sort : Off)				
- II Mission1	Kan New Post-Processing Scenario				Nodes	Elen	nents	
- PRA			-		Component	Range	e Off	se
					GT1_afm	0-0	0	
					- bolts.fem	1-184	2 532	6
					- HPT fem.fem	1534-	. 534	4
					- HPC_fem.fem	2087	798	8

	ompone	ent							
Automatic La	bel Res	olutio	n						
✓ Entities									
✓ Nodes ✓ Elements									
Physical Properties Groups									
Modeling Ob	jects								
ffset to Nearest			1						
dditional Top Of	fset								
	A		Develo						
	Auton	natically	Kesolv	e					
	C	lear Off	sets						
Deep Label C	anflict	Charle							
Deep Label C	Conflict	Check							
Deep Label C Labels	Conflict	Check							
Deep Label C Labels Physical Properti	Conflict	Check	Plies	Mode	ling Objects				
Deep Label C Labels Physical Properti Nodes	ies Gro Eleme	Check oups ents	Plies	Mode	ling Objects Systems				
Deep Label C Labels Physical Properti Nodes Component	ies Gro Eleme Range	Check oups ents Offset	Plies Coo Offset	Model ordinate Range	ling Objects Systems Range Cor				
Deep Label C Labels Physical Properti Nodes Component	Conflict ies Gro Eleme Range 0-0	Check	Plies Coo Offset 0-0	Model ordinate Range	ling Objects Systems Range Cor				
Deep Label C Labels Physical Properti Nodes Component GGT1_afm – bolts.fem	Conflict ies Gro Eleme 0-0 1-1842	Check oups ents Offset 0 5326	Plies Coo Offset 0-0 53261	Model ordinate Range 6-534	ling Objects Systems Range Cor				
Deep Label C Labels Physical Properti Nodes Component GGT1_afm bolts.fem HPT_fem.fem	Conflict ies Gro Eleme 0-0 1-1842 1534	Check oups ents Offset 0 5326 5344	Plies Coo Offset 0-0 53261 53599	Model ordinate Range 6-534 1-803	ing Objects Systems Range Cor				
Deep Label C Labels Physical Properti Nodes GOTLafm - bolts.fem - HPT_fem.fem - HPC_fem.fem	Conflict ies Gro Eleme 0-0 1-1842 1534 2087	Check oups ents Offset 0 5326 5344 7988	Plies Cod 0-0 532610 53599 10076	Model ordinate Range 6-534 1-803 68-14	ing Objects Systems Range Cor • • • •				
Deep Label C Labels Physical Properti Nodes Gomponent GT1_afm – bolts.fem – HPT_fem.fem	Conflict ies Gro Eleme 0-0 1-1842 1534 2087	Check oups ents Offset 0 5326 5344 7988	Plies Cod Offset 0-0 532610 53599 100760	Model ordinate Range 6-534 1-803 68-14	ing Objects Systems Range Cor				
Deep Label C Labels Physical Properti Nodes Component GTL_afm - bolts.fem - HPT_fem.fem - HPC_fem.fem	Conflict ies Gro Eleme 0-0 1-1842 1534 2087	Check oups ents Offset 0 5326 5344 7988	Plies Coo 0-0 53261 53599 10076	Model ordinate Range 6-534 1-803 68-14	ing Objects Systems Range Cor				

#### **Simulation Label Conflicts**

With Simulation Active:

# Right-click Simulation file $\rightarrow$ Simulation Label Manager

iT1_sim.sim	Displayed & Wo						
🛛 🖷 GT1_afm.afm	B Edit						
CSYS	🖌 Edit Model Display	)(So					
Selection Recipes	Mew Solution	)(So					
Groups	New Solution From Condition Seque	nce. )(So					
Fields	New Solution Process	, )(So					
HPC_Heat Transfer Coefficient		Simulation Label	Manag	er			ა?
HPC_Pressure(1)	Replace FEM						
HPC_Table Field2	Physical Properties	Same Offsets in Simul	lation				
HPT_Heat Transfer Coefficient	1 Modeling Objects						
HPT_Pressure(1)	Minport Motion Loads	<ul> <li>Automatic Label</li> </ul>	Resolut	tion			
HPT_swirl ratio	Solve All Solutions	Offset to Nearest				1	,
HPT_Swirl Velocity	Import Results	onset to rearest					F
HPT_Table Field2	Simulation Summary	Automatically Resolve					
Tip_DX(1)	Result Measures						
Modeling Objects	Solution Manager	<ul> <li>Labels</li> </ul>					
Regions	Boundary Condition ID Manager	Coordinate Systems	Dhusica	Desparties		Medaling Object	10
Simulation Object Container	Simulation Label Manager	Coordinate Systems	Physica	roperties 0	sroups	modeling Objec	ts
-		Component		Range	Offset	Offset Range	Status
		🛃 GT1_sim		798921-799	0	798921-799	<ul> <li>Image: A set of the set of the</li></ul>
		- GT1_afm.afm		1-798891	0	1-798891	1



#### 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 <u>Overview of thermal solver</u> <u>files and how to use them</u> (<u>https://support.sw.siemens.com/en-</u> <u>US/product/289054037/knowledge-base/KB000128451\_EN\_US</u>).</u>

Time= 10000.0000000 Int	tegration	tim	estep=	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		
t Flow+Load Summary Into Dif	ferent Sin	nk E	ntities:		
Sink Entity		Ten	perature	e Heat	Energy absorbed
				Flow+Load	since start
Duct_Inlet_Temp_1		з.	933E+02	-4.694E-07	-3.995E+01
_Duct_Inlet_Temp_2		з.	931E+02	-5.655E-10	3.941E-05
c elements with no entity name	nes:	з.	673E+02	-7.620E+07	-5.311E+12
done.					
	END				
tion elansed time: 05 min 00	6 580				
re completed at:					
5. IUG MUY 29 ID.30.40 2023					
	Time= 10000.000000 Int Cpu time in ANALYZER modules Minimum temperature Maximum temperature Average temperature t Flow+Load Summary Into Diff Sink Entity 	Time= 10000.000000 Integration Cpu time in ANALYZER module= 222.5 Minimum temperature = Maximum temperature = Average temperature = t Flow+Load Summary Into Different Sin Sink Entity	Time= 10000.000000 Integration tim Cpu time in ANALYZER module= 222.5 Minimum temperature = Maximum temperature = Average temperature = t Flow+Load Summary Into Different Sink E Sink Entity Tem _Duct_Inlet_Temp_1 3. _Duct_Inlet_Temp_2 3. k elements with no entity names: 3. done. END ution elapsed time: 05 min 06 sec ve completed at: E: Tue Aug 29 16:30:46 2023	Time= 10000.000000 Integration timestep= Cpu time in ANALYZER module= 222.5 Minimum temperature = 288.150 Maximum temperature = 569.913 Average temperature = 295.980 t Flow+Load Summary Into Different Sink Entities: Sink Entity Temperature _Duct_Inlet_Temp_1 3.933E+02 _Duct_Inlet_Temp_2 3.931E+02 k elements with no entity names: 3.673E+02 done. END ution elapsed time: 05 min 06 sec ve completed at: e: Tue Aug 29 16:30:46 2023	Time= 10000.000000 Integration timestep= 900.000 Cpu time in ANALYZER module= 222.5 Minimum temperature = 288.150 at element Maximum temperature = 569.913 at element Average temperature = 295.980 t Flow+Load Summary Into Different Sink Entities: Sink Entity Temperature Heat Flow+Load Duct_Inlet_Temp_1 3.933E+02 -4.694E-07 Duct_Inlet_Temp_2 3.931E+02 -5.655E-10 k elements with no entity names: 3.673E+02 -7.620E+07 done. END END Lion elapsed time: 05 min 06 sec ve completed at: E: 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 follow	ving 41	element(s)		
**	exceeds 20%. Th	e total num	ber elements	with non-zero			
**	view factor sum	is is 💈	267. Incomple	ete enclosures	may		
**	exist in the mo	del. A compl	lete element	: list can be	-		
**	found in file g	roups.unv u	nder the grou	up name:			
**	BAD_VIEW_FACTOR	_SUM_ELEMENT	TS_000026				
	422	423 4	424 425	5 426	427	428	429



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







# **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		fx								
	А	В	с	D	E	F	G	н	I	
1	Name	Description	Туре	Value					Thermal Coupling List File:	D:\users\Test.txt
T	C-Hotbox-2	3.5mm bolts, small stiff surface 2.37	Total Conductance	1.68776371308017 W/°C						
2		C/W x2								
3 R	egulator-PCB	Perfect contact	Total Conductance	1000 W/°C			Road	d Eilo		
4 R	egulator-PCB-2	Perfect contact	Total Conductance	1000 W/°C			inear	urne		
5 R	egulator-PCB-3	Perfect contact	Total Conductance	1000 W/°C						
6 R	egulator-PCB-4	Perfect contact	Total Conductance	1000 W/°C						
7 G	apPad	Sil-pad	Conductive Gap	0.9 W/(m*K)						
8 P	CDU-PCB-HeatSink	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
9 P	CDU-PCB-HeatSink(2)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
10 P	CDU-PCB-HeatSink(3)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
11 P	CDU-PCB-HeatSink(4)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
12 P	CDU-PCB-HeatSink(5)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
13 P	CDU-PCB-HeatSink(6)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
14 P	CDU-PCB-HeatSink(7)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
15 P	CDU-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. <u>https://help.mayahtt.com/tmg\_correlation/v1/index.html</u>



# 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.