## Model checks and troubleshooting in turbomachinery

SIEMENS



#### 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-word conditions accurately.
- Identify issues such as mesh distortion, element quality, or excessive skewness.
- Verify the material 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 properties
- $\checkmark\,$  Verify elements quality and mesh
- $\checkmark\,$  Verify mesh normals
- $\checkmark\,$  Check for duplicate nodes
- $\checkmark\,$  Do model mass check
- $\checkmark\,$  Verify the element thickness
- Generate a boundary condition contour plot to display defined quantities
- ✓ Display all thermal couplings
- $\checkmark\,$  View thermal results on model
- $\checkmark\,$  Check log file for errors and warnings
- $\checkmark\,$  Run model setup check
- ✓ Check thermal connections

- ✓ Resolve ID conflicts
- Inspect various result quantities associated to boundary conditions using BCDATA PLOT and TABLES
- Review a graph illustrating the dependencies in the boundary conditions
- $\checkmark\,$  Check convective thickness and area factors
- ✓ Inspect coupling areas in scratch files
- $\checkmark\,$  Check pressures on walls
- $\checkmark\,$  Perform adiabatic check
- Run conduction only solution to check thermal contacts
- $\checkmark\,$  Check mass flow and stream nominal directions
- $\checkmark\,$  Check temperature gradients at shutdown conditions
- $\checkmark\,$  Use the traceback patch

#### **Most common issues**

- Units
- Material 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 expression for potential issues that could arise from changing units.
- Make units consistent. Select the List Expression and Units Inconsistency Warnings customer default to display the warning when inconsistent units are used.
- Do one of the following, since not all units are case insensitive.
  - Use the auto-complete suggestion for the unit.
  - Use the unit case as displayed in the Units Manager dialog box.

For more information, see <u>Case sensitivity in expressions</u>.

 Use the Plot Contours command to view the values used in the boundary conditions. This applies to non-solver evaluated boundary condition definitions.

# Solution Solution Name Solution Solver Sincenter 3D Multiphysics Analysis Type Coupled Thermal-Structural 2D Solid Option None Solution Type Thermal-Structural (SOL 401 Multi-Step Nonlinear) Reference Set Entire Part

✓ Automatically Create Step or Subcase

#### Thermal-Structural (SOL 401 Multi-Step Nonlinear)

General	<ul> <li>Solution Units</li> </ul>	
Solution Control		
Structural Parameters	Solve-Time Units	Current Part
Solution Units	<ul> <li>Units Information</li> </ul>	
Ambient Conditions	• Onits mormation	
nitial Conditions	Mass	kg
Articulation Parameters	Length	mm
Results Options	Time	second
Restart Management	Power	microWatt
	Heat Flux	microW/mm^2
	Energy	microJoule
	Velocity	mm/s
	Pressure	mN/mm^2
	Viscosity	kg/mm-s
	Density	kg/mm^3
	Specific Heat	microJ/kg-C
	Force	mN
>		
/		



#### **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.
- 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.

Manage Materials Material List						U ?	1
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		1
AISI_Steel_Maraging	METAL	Isotropic	8e-06 kg/mm^3	25300 microW/mm-dC	476000000 micr		
<	APTAI	141-	2 704- OC 1 /	T-L. I D-A	••••••••••••••••••••••••••••••••••••••	*	

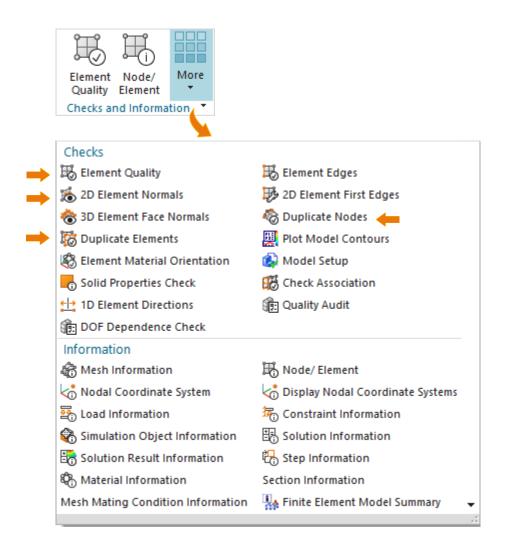
#### Verifying mesh density

- 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 with  $\beta$ <15°.
- Ensure the quality and consistency of your mesh.
- Validate that the model is complete and ready to solve.



SIEMENS

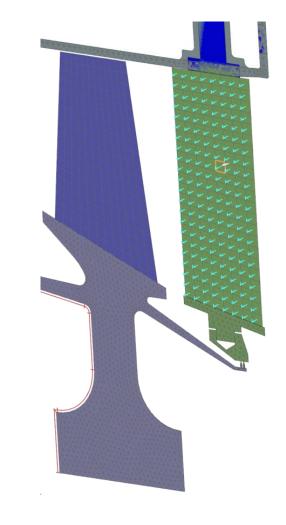
#### Checking and orienting the element normals

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:

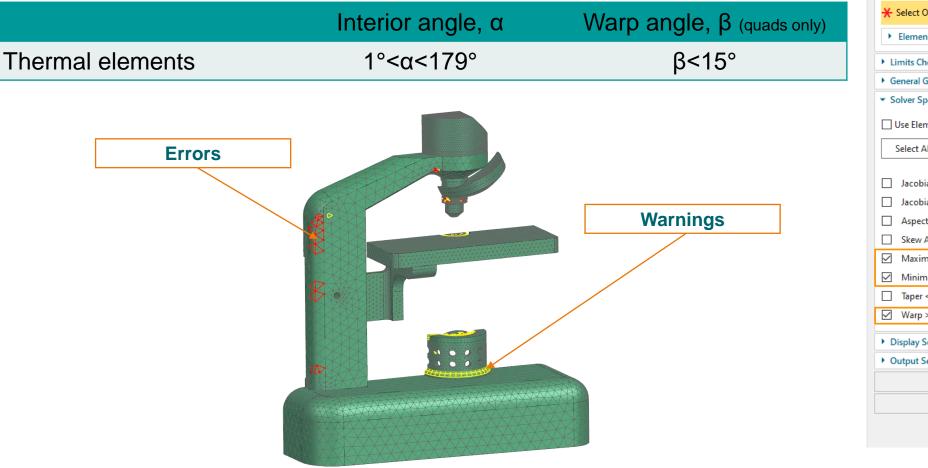
- Properly defining contact between surfaces.
- Properly defining top and bottom in the radiation request.





#### **Evaluating element quality**

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



Element Quality 🛛 ? 🗙							
▼ Elements to Check							
Selected		•					
X Select Object (0)		<u></u> Ф …					
Element Labels							
Limits Check							
General Geometry Checks							
▼ Solver Specific Geometry Checks							
Use Element Type Specific Values							
Select All Deselect All							
	Error Limit						
Jacobian Ratio >	30	*					
Jacobian Zero <=	0	Ŧ					
Aspect Ratio >	20	Ŧ					
Skew Angle >	30	° • •					
Maximum Interior Angle >	179	• • •					
Minimum Interior Angle <	1	° • •					
Taper <	0.5	-					
☑ Warp >	15	° • •					
Display Settings							
Output Settings							
Check Eleme	ents						
Correct Failed El	ements						
		Close					



#### **Identifying coincident nodes**

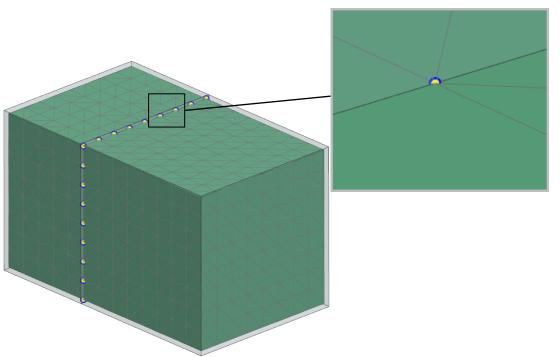
Check for coincident nodes which are duplicate nodes lying on top of each other.

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.

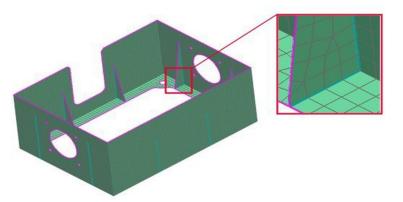
#### **Visual representation of coincident nodes**





#### **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 are many problem edges or if the part fails to mesh, you may need to repair the underlying master part geometry in the Modeling application.
  - 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.



🌣 Model Display	<b>৩ ?</b> ×
<ul> <li>Parameters</li> </ul>	
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>Thickness</li> <li>0.35 mm ▼</li> <li>Line Style</li> <li>✓</li> </ul>
	OK Apply Cancel

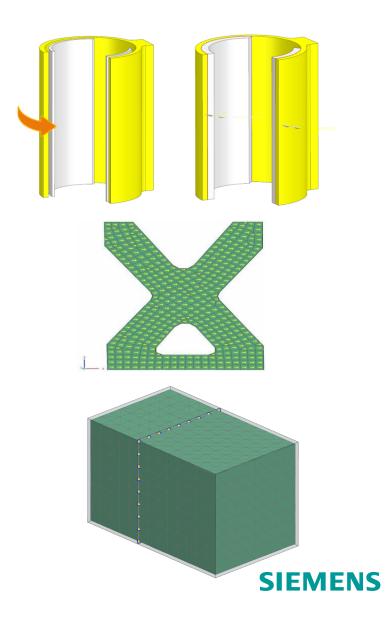
SIEMENS

#### **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		Minimum 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	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

Information	ø	?	-		×
					*
Solid Properties reported for following object(s)					
Solid Properties reported for following object(s) 2d_mesh(5)					
Volume : 8.037122E+03 mm <sup>3</sup> Surface Area : 2.679041E+03 mm <sup>2</sup>					
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∕	К			
Solid properties were calculated for temperature = 0					
Solid properties were calculated for temperature = 0					
4				+	*



#### 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 ٠ that shows the general statistical distribution of the thickness values across your 2D mesh.

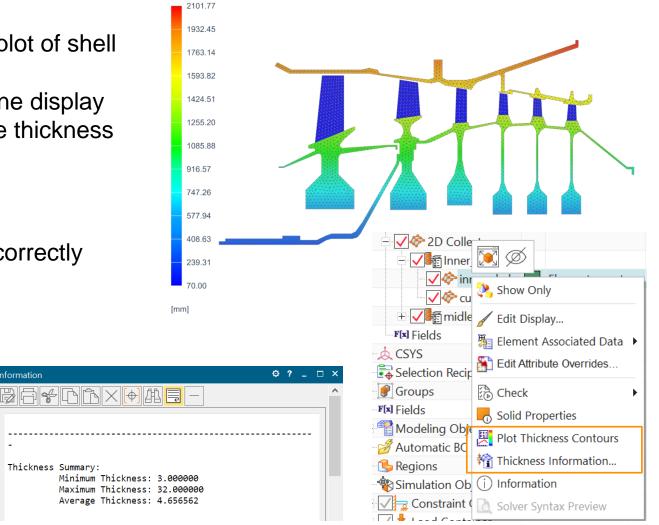
You can use the thickness display to quickly identify:

Any sudden changes in color that may indicate incorrectly ٠ assigned thickness values.

oformation

Thickness Summarv

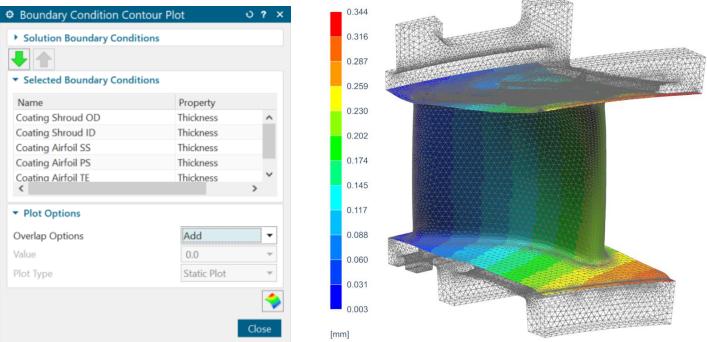
Elements that do not have an assigned thickness. •



#### Generating plot contours of boundary conditions

You can use the **Boundary Condition Contour Plot** command to generate a contour plot of most types of loads, constraints, and solver-specific simulation objects that contain a value. You can use these contour plots to verify your loading conditions, to generate high-quality visualizations for reports or presentations, and to interrogate and extract loading data.

In this example, the thickness field varies spatially. The coating is thicker at the leading edge and gets thinner towards the trailing edge.





#### **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  $\rightarrow$  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 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.

#### **Checking radiation enclosures**

- Check external and internal enclosures.
- Run a radiation only test to verify the set up.
- 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.
- Increase radiation calculation accuracy by using higher element subdivision, hemicube rendering, or more rays, and check if it impacts the temperatures.



#### **Configuring solution settings – Initial Conditions**

- Check the global ambient and initial conditions in the **Solution** dialog box. ٠
- Set the local initial conditions in the **Initial Conditions** constraint. ٠ Local conditions override global conditions.

Solution		ა ? ×	Initial Conditions	<b>৩ ?</b> া
▼ Solution			• Name	
Name	Solution		<ul> <li>Destination Folder</li> </ul>	
Solver	Simcenter 3D Space Systems Thermal	Ψ		
Analysis Type	Thermal	~	▼ Region	
Solution Type	Space Systems Thermal	v	Group Reference	
Reference Set	Entire Part	Ψ.	Body Focus	
<ul> <li>Space Systems Thermal</li> </ul>			✓ Select Object (0)	<b>•</b> •••
Solution Details	Initial Temperature	Perform Steady-State Solution	Excluded	
<ul> <li>Ambient Conditions</li> <li>Initial Conditions</li> <li>Restart</li> </ul>	<ul> <li>Transient Thermal Loads</li> <li>Transient Thermal Loads to Use for Steady State Initial Conditions</li> </ul>	Automatic Uniform From Results in Other Directory	Region Override	
Thermal	Initial Conditions Use Loads at Time	From File (TEMPF format)	<ul> <li>Initial Temperature</li> </ul>	
- Transient Setup		Perform Steady-State Solution	Temperature	C • =
Results Options				
			Card Name Initial Temperature	
			OK	Apply Cancel
		OK Apply Cancel		



? ×

#### **Configuring solution settings – Transient Setup**

Verify the following transient solution options:

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

To speed up slow transient runs:

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

<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	▼ Solution Time Interval	
- Solution Units	Start Time	0 sec · ·
- Ambient Conditions		
- 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 Cha
	Number of Time Steps	100
	Results Sampling	
	Articulation Parameters	
	<	>



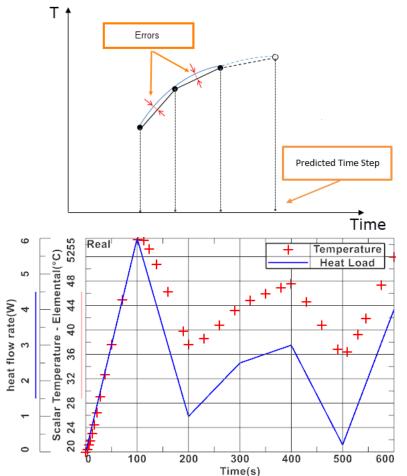
#### Using automatic time step

One way to speed up a solve is to use 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.

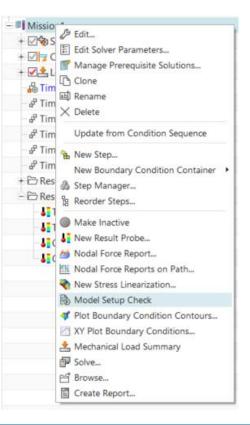


#### Running a model setup check

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

#### Model Setup Check outputs model checks to the Information window on:

- Assembly fem label conflicts.
- Simulation label conflicts.
- Mesh, materials, and physical properties.
- Groups.
- Loads, constraints, boundary conditions, such as invalid selections and values.
- Solutions.



Solve		ა	?	×
▼ Solvir	ng Options			
Submit	Solve			•
🗌 Mode	Setup Check			

#### SIEMENS

#### **Resolving ID conflicts**

#### **AFM label conflicts**

### Right-click the active assembly FEM file $\rightarrow$ Assembly Checks $\rightarrow$ Assembly Label Manager

Simulation Navigator				
Name		C	Status	Filter
GT1_sim.sim			Displayed	
+ 🗹 🖥 GT1_afm.afm	ै, Update		Work	
+ 🗹 🗁 CSYS				(Filter : Off)(Sort : Off)
• 🖉 🛱 Selection Recipe	G Freeze Assembly Updates	_	1 recipes	(Filter : Off)(Sort : Off)
+ 💽 Groups	Open in Window		285 groups	(Filter : Off)(Sort : Off)
- VIN Fields	FEM Representation Display	•		(Filter : Off)(Sort : Name)
HPC_Heat Trans	De Edit	_	No Interpolator	
HPC Pressure		-		
HPC_Table Fie	* New Connection	,	No Interpolator	
HPT_Heat Trans	K New Universal Connection	•	No Interpolator	
- I HPT_Pressure	8 New Simulation			
- HPT_swirl rati	🍓 Add Existing Component			
- HPT_Swirl Vel	S Reset Node and Element IDs		No Interpolator	
HPT_Table Fie	Assembly Checks	•	Assembly Labe	Manager
- 7 Tip_DX(1)	🍓 Update Options		Component Su	b-Assembly Label Conflicts
+ 1 Modeling Objects	🎭 Element Associated Data	•	Gomponent Up	date Pending
+ 🗁 Regions	Model Summary	1		(Filter : Off)(Sort : Off)
+ 🗹 🎕 Simulation Obje	to Manage Component Positioning CSYS			(Filter : Off)(Sort : Off)
+ 🗹 📴 Constraint Conta	🍪 Hide			(Filter : Off)(Sort : Off)
+ 🗹 📥 Load Container	🚯 Show Only			(Filter : Off)(Sort : Off)
+ 🗹 🗁 Solver Sets	Report Association			(Filter : Off)(Sort : Off)
Mission1	🔀 New Post-Processing Scenario			

Automatic La	bel Res	olutio	n		
<ul> <li>Entities</li> </ul>					
✓ Nodes		$\checkmark$	Elemer	nts	
Physical Prop	erties		Groups		
Modeling Ob					
Offset to Nearest			1		
dditional Top Of	fset			[	
	Auton	natically	Resolv	e	
	c	lear Off	sets		
		-			
Deep Label C	onflict	Check			
Labels					
Physical Properti	es Gro	oups	Plies	Mode	ling Object
Nodes	Eleme	ents	Cod	ordinate	Systems
Component	Range	Offset	Offset	Range	Range Cor
🖥 GT1_afm	0-0	0	0-0		<ul> <li>Image: A set of the set of the</li></ul>
<ul> <li>bolts.fem</li> </ul>	1-1842	5326	53261	6-534	<ul> <li>Image: A set of the set of the</li></ul>
- HPT_fem.fem				1-803	
	2087	7988	10076	68-14	~
- HPC_fem.fem					
- HPC_fem.fem					>
- HPC_fem.fem					1

#### **Simulation label conflicts**

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

)(So )(So

)(So )(So )(So

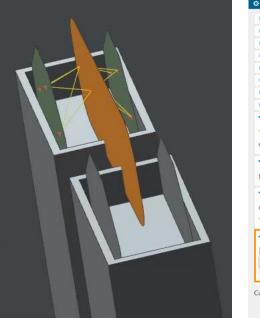
GT1_sim.sim	Displayed & Wo
+ ☑ = GT1_afm.afm + ☑ = CSYS	Bedit Edit Edit Model Display
Compared Selection Recipes      Oroups       Oroups      Orou	New Solution      New Solution From Condition Sequence      New Solution Process
HPC_Heat Transfer Coefficient     ImpC_Pressure(1)     ImpC_Pressure(1)     ImpC_Table Field2     HPT_Heat Transfer Coefficient     ImpT_ressure(1)     ImpT_swirl ratio     ImpT_Swirl Velocity     ImpT_Table Field2     ImpT_Table Field2	Replace FEM  Replace FEM.

	Manager			<u>ر ہ</u>
Same Offsets in Simul	ation			-
<ul> <li>Automatic Label</li> </ul>	Resolution			
Offset to Nearest			1	•
Automatically Resolve				5
<ul> <li>Labels</li> </ul>				
Coordinate Systems	Physical Properties	Groups	Modeling Object	ts
Coordinate Systems Component	Physical Properties Range	Groups		
		Offset		Status
Component	Range	Offset	t Offset Range	Status
Component	Range 798921-799	Offset	t Offset Range 798921-799	Status
Component	Range 798921-799	Offset	t Offset Range 798921-799	Status
Component	Range 798921-799	Offset	t Offset Range 798921-799	Status



#### **Verifying thermal couplings**

- Verify that the thermal coupling is set up 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 the 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	ن <b>؟</b>
• 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	
G	
Ancillary Display Options	
ard Name Thermal Coupling	
	OK Cano



#### **Verifying thermal couplings**

- Check warning messages in the log file.
- Use the Report command to investigate heat flow between components in the assembly and from each component to the environment. This helps you identify areas with significant heat, allowing you to determine where thermal tapes or thermal standoffs would be most beneficial 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.
- If the model has perfect contact thermal couplings and is experiencing convergence issues, use the Thermal Coupling with a high heat transfer coefficient instead to define a coupling that represents a perfect/contact resistance interface, where the mesh does not match.

#### **Using the Thermal Coupling Report tool**

- Use an Excel file to generate a table of thermal coupling data for a model.
- Right-click all thermal couplings in the model and select **Information**. Save the information window to a text file, and then import this text file into the Excel sheet.

A	В	С	D	E	F	G	н	I	
Name	Description	Туре	Value					Thermal Coupling List File:	D:\users\Test.txt
TC-Hotbox-2	3.5mm bolts, small stiff surface 2.37	Total Conductance	1.68776371308017 W/°C						
	C/W x2								
Regulator-PCB	Perfect contact	Total Conductance	1000 W/°C			Boo	d File		
Regulator-PCB-2	Perfect contact	Total Conductance	1000 W/°C			Red	urile		
Regulator-PCB-3	Perfect contact	Total Conductance	1000 W/°C						
Regulator-PCB-4	Perfect contact	Total Conductance	1000 W/°C						
GapPad	Sil-pad	Conductive Gap	0.9 W/(m*K)						
PCDU-PCB-HeatSink	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
PCDU-PCB-HeatSink(2)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
0 PCDU-PCB-HeatSink(3)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
1 PCDU-PCB-HeatSink(4)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
2 PCDU-PCB-HeatSink(5)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
B PCDU-PCB-HeatSink(6)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
4 PCDU-PCB-HeatSink(7)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						
5 PCDU-PCB-HeatSink(8)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C						

Refer to the **ThermalCoupling.xlsm** file linked to this knowledge base article.

Note: This is not commercial grade tool. It is provided as is and not supported by us.



#### Inspecting coupling areas per element in scratch files

You can inspect elemental coupling areas in the solver, by converting the MODLCF file to ASCII format either by:

- Using the executive menu command.
- Specifying a FILES MODLCF, VUFF, MODLF IN ASCII advanced parameter.

From the MODLCF file below, surface element 66974 is connected to elements 67375 and 67376.

Advanced Controls	C:\Windows\System32\cmd.exe
a Advanced Parameter	Microsoft Windows [version 10.0.19044.1826] (c) Microsoft Corporation. Tous droits réservés.
▶ Name	D:\Projects\P W\thermal connection test\D:\NoRackun\NY LOCAL\NY2206\NYCAE EXTRAS\tmg\com\tmgny cmd AS
Destination Folder	Launching TMG executive menu in text mode with UGII_TMG_DIR=D:\NoBackup\NX_LOCAL\NX2007\nxcae_extras\tmg Binary file VUFF has been converted to ASCII format
▼ Region	Binary tile MODLF has been converted to ASCII format
<ul> <li>Body Focus</li> </ul>	Binary file MODICE has been converted to ASCII format
✓ Select Object (0) 👘 …	Binary file TEMPF has been converted to ASCII format Binary file GTEMPF has been converted to ASCII format
✓ Parameter Definition	D:\Projects\P_W\thermal_connection_test>_
Catalog	O kletný 2 X
Name FILES MODLCF, VUFF, MODLF IN ASCII	Demet Reults Pick from Model -
Type None	Adva Section Much Review of the section of the sect
Card Name Advanced Parameter           OK         Apply         Cancel	Element 67376 50 °C Coupled Area
	Search "66974" (3 hits in 1 file of 1 searched)
	D:\Projects\P W\duct to wall coupling cest\TEST01\MODLCF (5 hits) Line 44175: CAP 66974 0.0000E+00 1.0000E+00
	Line 82240: CNF 66974 67375 0.0456E+02 1 1000E+01 3999. 6.6998E+01 1.1000E+01 5.0380E+01 0.0000E+00 1882 Line 82241: CNF 66974 67376 .9942E+02 1 1000E+01 3999. 6.6998E+01 1.1000E+01 1.6618E+01 0.0000E+00 1882

**SIEMENS** 

#### **Verifying thermal connections**

Investigate if the primary and secondary elements are correctly connected thermally using the **Thermal Connection** result sets. This enables you to contour thermal connections in their model to verify element connections.

Thermal Output Requ	Jests		0 ? X
<ul> <li>Modeling Object</li> </ul>			
Name HPT_Therm	nal Output Requests1(1)		
Label 798933			
<ul> <li>Properties</li> </ul>			
Description			6
- Control			
			All On/Off
Create Groups for Boun	ndary Conditions		All On/Oll
Thermal Radiation			
- Radiative Source Fluxes	Entity	All	•
1D Flow	☑ Temperatures		
	Temperature Results Location	Nodes	•
	Min/Max Temperatures and Occurrence Times		
	Total Temperatures		
	Total Temperature Results Location	Nodes	•
	Local Temperature Error Estimate		
	Conductive Heat Fluxes		
	Convective Heat Flux Results Location	Elements and Nodes	-
	Temperature Gradients		
	Transverse Temperature Gradients		
	Total Loads and Fluxes		
	Residuals		
	Free and Forced Convection Coefficients		
	Free and Forced Convection Coefficient Results Location Area Corrected Free and Forced Convection Coefficients	Nodes	•
	Area Corrected Free and Forced Convection Coefficients     Wall Velocities		
	Joule Data		
	Phase Change Quality		
	RC Products		
	✓ Displacements		
	Thermal Connections		
	Coupled Area Ratio		

OK Cancel

#### Using the plot bc summary

To monitor important areas of your model at run-time, you create **Advanced Controls** with the PLOT BC SUMMARY advanced parameter in the solution. The thermal solver generates the *<simulation name>-<solution name>data.html* file where you can inspect various result quantities associated to boundary conditions, thermal couplings, and named points.

The graph below shows two stream inlet and outlet temperatures during a transient analysis. **Convective Area** can also be inspected in this report.

Advanced Controls	ა? ×	Thermal Convecting Zone		
▶ Туре		Thermal Stream		
Name		Thermal Void	800	7
<ul> <li>Destination Folder</li> </ul>		Named Points	/	
▼ Region		Thermal Coupling	600	1 I I I I I I I I I I I I I I I I I I I
<ul> <li>Body Focus</li> </ul>			1	
✓ Select Object (0)	۰۰۰ 🎓		400	
<ul> <li>Parameter Definition</li> </ul>				
Catalog	Ô		0 2k 4k	6k 8k 10k Time Seconds
Name PLOT	BC SUMMARY		Temperature 😽	_
Type Non	e		HPT_Stream 3	
Card Name Advanced Parameter			HPT_Stream 4	
	OK Cancel			



#### Using bc summary tables

When you include the DISPLAY BC SUMMARY TABLES advanced parameter in the solution, the thermal solver generates the *<simulation name>-<solution name>.bcdata* file where you can inspect various quantities related to thermal couplings, voids, streams, and more. A common use case is to validate the <u>convective area</u>.

•
🍞 ···
Ô
PLAY BC SUMMARY TABLES
e
OK Apply Cancel

-										
-						VOID BC DATA				
	ID	Reg.#	TIME	Tmax	Pmax	THR	Trel-max	CHR	AREAC	SVmax
-	1.	1.	8200.	393.1	0.2464	-0.2997E-01	393.1	-0.2997E-01	0.2752E+06	0.0000E+00
	2.	1.	8200.	393.1	0.2464	-0.1892E-01	393.1	-0.1892E-01	0.6561E+05	<b>0000E+00</b>
	3.	1.	8200.	393.1	0.2464	0.1519E-01	393.1	0.1519E-01	0.7295E+05	0.0000E+00
	4.	1.	8200.	393.1	0.2464	-0.2271E-01	393.1	-0.2271E-01	0.5865E+05	0.00001.00
	5.	1.	8200.	393.1	0.2464	-0.1959E+07	393.1	-0.1959E+07	0.2742E+07	0.0000E+00
	6.	1.	8200.	304.8	0.1111	0.3397E-12	304.8	0.3397E-12	0.3402E+06	0.9231E+05
		2.	8200.	304.8	0.1111	0.8531E-13	304.8	0.8531E-13	0.5081E+05	0.0000E+00
	ALL RE	GIONS	8200.	304.8	0.1111	0.4250E-12	304.8	0.4250E-12	0.3910E+06	0.9231E+05
	7.	1.	8200.	472.7	0.1111	0.4194E-13	472.5	0.4194E-13	0.1286E+06	0.9195E+05
		2.	8200.	472.7	0.1111	15.83	472.7	15.83	0.7849E+05	0.0000E+00
	ALL RE	GIONS	8200.	472.7	0.1111	15.83	472.7	15.83	0.2071E+06	0.9195E+05
	8.	1.	8200.	307.7	0.1111	0.2674E-12	307.5	0.2674E-12	0.1338E+06	0.8242E+05
	9.	1.	8200.	361.9	0.1923	0.7710E+07	361.9	0.7710E+07	0.1294E+07	0.0000E+00
	10.	1.	8200.	363.6	0.1346	0.2438E+07	363.6	0.2438E+07	0.2312E+06	0.0000E+00
	11.	1.	8200.	421.5	0.1571	0.3677E+07	421.5	0.3677E+07	0.2342E+06	0.0000E+00
	12.	1.	8200.	314.0	0.1111	0.1684E+06	314.0	0.1684E+06	0.1344E+06	0.1231E+06
		2.	8200.	314.0	0.1111	0.1677E+06	314.0	0.1677E+06	0.1672E+06	0.1231E+06
	ALL RE	GIONS	8200.	314.0	0.1111	0.3361E+06	314.0	0.3361E+06	0.3016E+06	0.1231E+06
	13.	1.	8200.	303.2	0.1111	0.4573E-13	301.9	0.4573E-13	0.1175E+06	0.1190E+06
		2.	8200.	303.2	0.1111	0.2944E-13	296.1	0.2944E-13	0.4842E+05	0.1221E+06
		з.	8200.	303.2	0.1111	0.7991E-13	301.9	0.7991E-13	0.1308E+06	0.1221E+06
	ALL RE	GIONS	8200.	303.2	0.1111	0.1551E-12	301.9	0.1551E-12	0.2967E+06	0.1221E+06
	14.	1.	8200.	302.4	0.1111	0.1847E-13	302.4	0.1847E-13	0.9839E+05	0.1076E+06
		2.	8200.	302.4	0.1111	0.4280E-13	302.4	0.4280E-13	0.1561E+06	0.1164E+06
	ALL RE	GIONS	8200.	302.4	0.1111	0.6126E-13	302.4	0.6126E-13	0.2545E+06	0.1164E+06
	15.	1.	8200.	304.3	0.1111	0.2246E-12	304.0	0.2246E-12	0.2973E+06	0.1171E+06
	16.	1.	8200.	303.7	0.1111	0.8619E-13	302.5	0.8619E-13	0.1227E+06	0.1187E+06
		2.	8200.	303.7	0.1111	0.1032E-12	302.5	0.1032E-12	0.1622E+06	0.1199E+06
	ALL RE	GIONS	8200.	303.7	0.1111	0.1894E-12	302.5	0.1894E-12	0.2849E+06	0.1199E+06
	17.	1.	8200.	293.2	0.1259	0.1031E-12	292.8	0.1031E-12	0.5358E+06	0.1048E+06
	18.	1.	8200.	302.1	0.1111	0.2548E-13	300.6	0.2548E-13	0.1289E+06	0.9389E+05
	19.	1.	8200.	386.8	0.2369	-0.1033E+05	386.8	-0.1033E+05	0.8465E+05	0.0000E+00
	20.	1.	8200.	344.6	0.1788	-0.3840	344.6	-0.3840	0.1695E+06	0.0000E+00

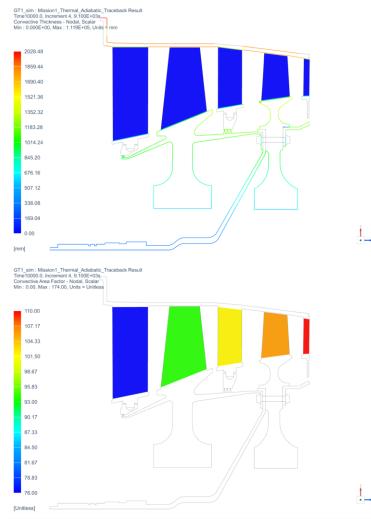
Confirm convective area of boundary conditions



#### **Displaying convective thickness and area factors**

In the **Post Processing Navigator** you can display:

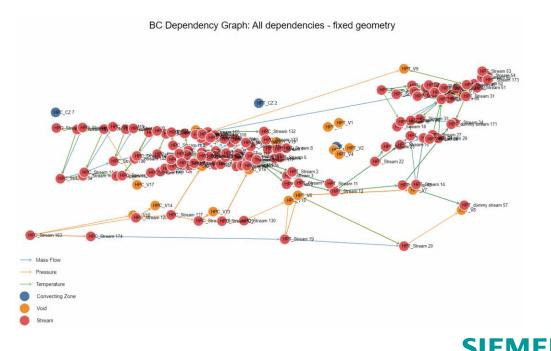
- **Convective Area Factor** to visualize applied area factors on convective BCs.
- **Convective Thickness** to visualize convective area of 2D element edges. The thermal solver computes the thickness in a hybrid 2D-3D axisymmetric model depending on the 2D element type as follows:
  - For a 2D axisymmetric element, the element thickness is equal to  $2\pi$  times the radius.
  - For a 2D plane stress or strain element, the element thickness is equal to the specified thickness times the number of instances.
  - For a 2D chocking element, the element thickness is equal to  $2\pi$  times the radius minus the specified thickness times the number of instances.



#### Using the bc dependency graph

When you include the BC DEPENDENCY GRAPH advanced parameter in the solution, the thermal solver generates the *BCInterdependencyGraph.html* file, that contains a graph illustrating the dependencies, such as temperature or mass flow, between the thermal streams, voids, and convecting zones boundary conditions in the solution. You can also display the dependencies, such as pressure, swirl velocity, heat load, area correction, and heat transfer coefficient, or choose only to display the temperature or the mass flow rate dependencies, separately.

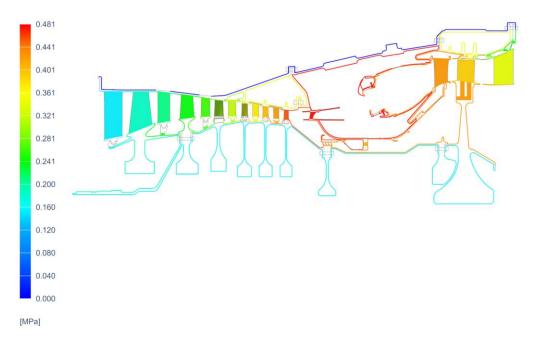
Advanced Controls	ა? X
▶ Туре	
Name	
Destination Folder	
▼ Region	
Body Focus	
✓ Select Object (0)	👔 😶
<ul> <li>Parameter Definition</li> </ul>	
Catalog	Ô
Name	BC DEPENDENCY GRAPH
Туре	Integer
Value	4
Card Name Advanced Parameter	
	OK Cancel



#### **Checking fluid pressures on walls from the thermal solve**

Errors in applying pressure are easy to overlook and can lead to significant deflection errors in thermal-structural runs.

Perform spot checks on the pressure results at different time points to verify their accuracy. GT1\_sim : Baseload\_Hold Result Time10000.0, Increment 1, 1.000E+03s Fluid Pressure on Walls - Nodal, Scalar Min : 0.000, Max : 0.481, Units = MPa

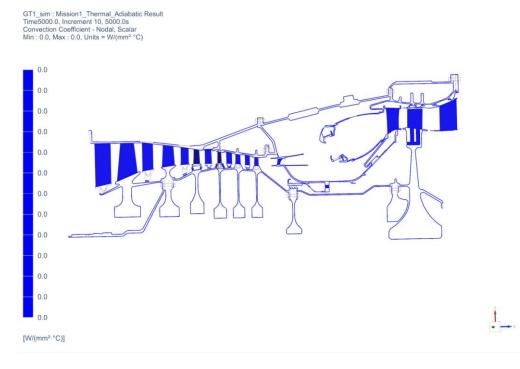


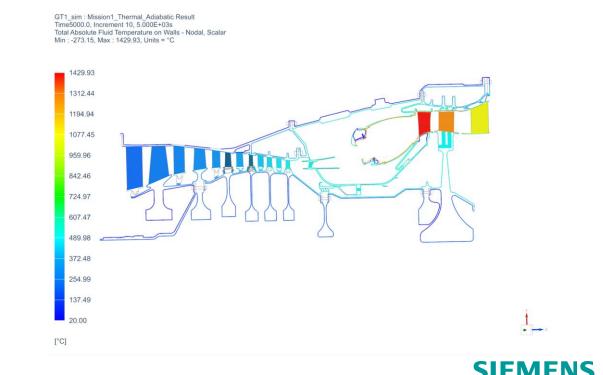


#### **Running the model adiabatically**

- Run the model adiabatically (HTC=0) and compare it to secondary air results to confirm that the adiabatic heat pickup in the fluid network due to windage is reasonable.
- Verify this by examining the **1D Fluid Temperature**, **Total Absolute**, or **Relative Fluid Temperature**.

Comparisons with secondary air results are valid only for adiabatic secondary air models.

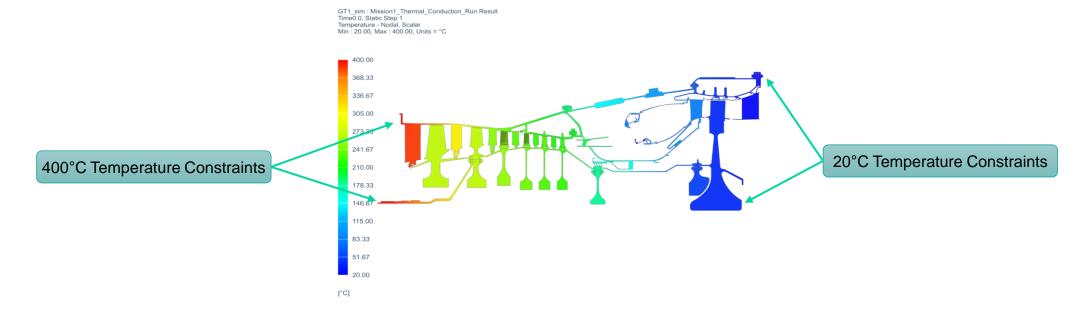




#### **Running a conduction only simulation**

- Remove all convective and radiative boundary conditions and loads, but retain thermal contacts and joints in the model.
- Apply constraints at both ends.

This allows you to confirm that thermal contacts are correctly modeled and serves as a check for specific heat in transient runs.

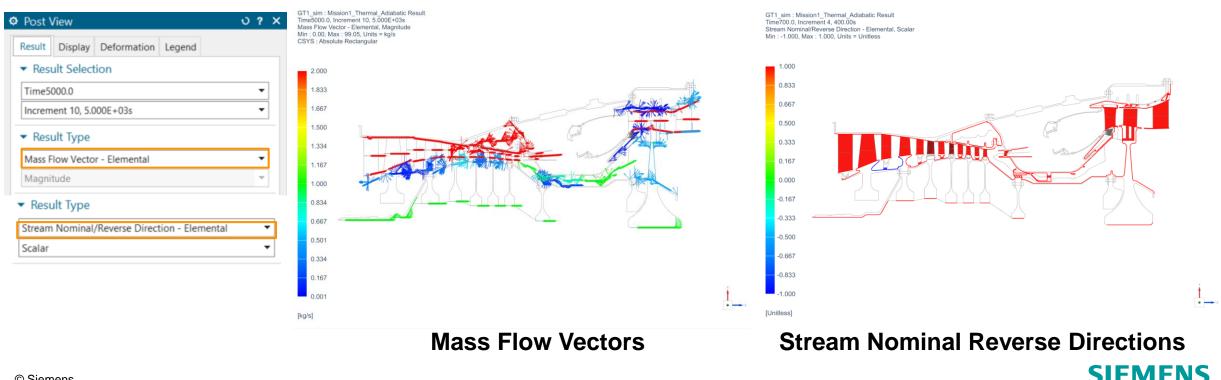


SIEMENS

#### **Checking mass flow and stream nominal directions**

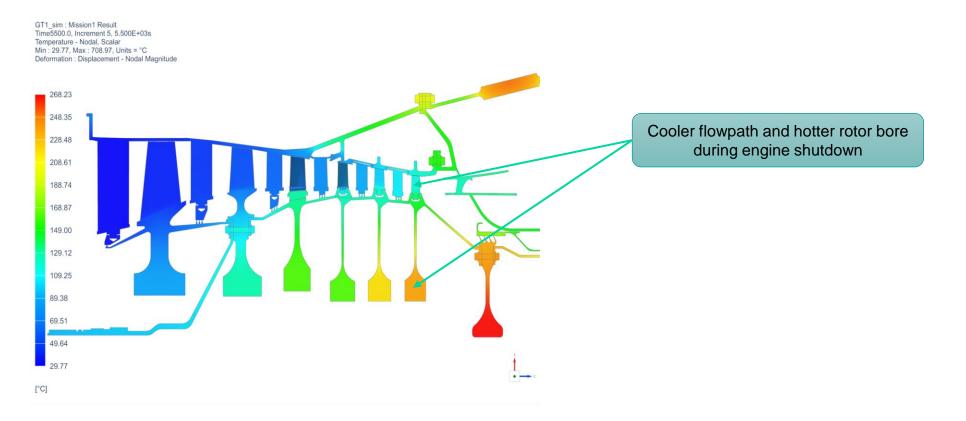
You can display:

- **Mass Flow Vector** to confirm mass flow directions within the fluid network. Use the **Arrows** command to view • the direction of the mass flow in ducts and streams.
- Stream Nominal/Reverse Direction to contour and identify flow reversals: nominal (1) or reverse (-1). •



# **Checking temperature gradients at shutdown conditions**

Temperature scaling issues are common in transient analysis. Inspect temperatures upon shutdown to ensure appropriate transient behavior.





# **Thermal solver troubleshooting**

The following resources help troubleshoot the model:

- Review the following files:
  - Log
  - Verbose
  - Report
- Inspect the partial .bun file.
- Apply the traceback patch.
- Simplify the model by removing some features.



# **Inspecting a log file**

<simulation/model name>-<analysis name>.log 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 the 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> 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 Si	nk 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	۱ +
Solution elapsed time: 05 min 06 sec	
Solve completed at:	
Time: Tue Aug 29 16:30:46 2023	
-	

### **Inspecting a verbose file**

Inspect the *<simulation/model name>-<analysis name>\_verbose.log* file to review crashes, memory or time usage, and post-processing issues. This log file may contain specific details about the cause of the crash and the process being executed at the time.

Advanced Controls	Time= 50.0000000000 Integration timestep= 50.0000 Cpu time in ANALYZER module= 24.11
data Advanced Parameter	
▶ Name	Minimum temperature         =         288.150 at element         1296635           Maximum temperature         =         569.913 at element         1136454           Average temperature         =         296.028
Destination Folder	Average temperature = 296.020
▼ Region	Heat Flow+Load Summary Into Different Sink Entities: Sink Entity Temperature Heat Energy absorbed
Body Focus	Flow+Load since start
✓ Select Object (0)	HPT_Duct_Inlet_Temp_1         3.933E+02         -3.327E-04         -1.664E-02           HPT_Duct_Inlet_Temp_2         3.931E+02         0.000E+00         0.000E+00
✓ Parameter Definition	nri_Dotinieiempz 5.5316+02 0.0000+00 0.0000+00 Sink elements with no entity names: 3.6738+02 -7.6208+09 -3.8028+09
	ILU iteration 1 Residual= 7.78E-07 ILU iteration 1 DTmax= 1.22E-02 at 0
Name LEVEL OF VERBOSENESS OF MESSAGING OUTPUT	Ibo footofool i Dimma- fibbo did
Type Integer	Iter Tmax At Tmin At TDmax At T(TDmax) Time
Value S	1 569.91 1136432 293.13 1304180 8.06E-03 1109770 293.14 1.0000000E+02
Card Name Advanced Parameter	
OK Apply Cancel	<ul> <li>Printing out thermal solve results for time= 0.100000E+03</li> <li>Memory snapshot: Before results printout:</li> <li>Memory snapshot: Peak Virtual Memory (MB), Peak Physical Memory (MB), Allocated C arrays (MB), Top 20 C arrays (MB)</li> <li>Memory snapshot: Rank 0 0.000E+00 0.000E+00 0.124E+04 755.</li> <li>Top C array labels:</li> </ul>
	* Memory snapshot: Rank 0 901 903 1767 1768 1772
Now owner in iteration accurate this TNAC Flows and	<ul> <li>Memory snapshot: Rank 0 905 1769 2607 176 907</li> <li>Memory snapshot: Rank 0 422 1654 2609 1658 835</li> </ul>
Max error in iteration occurs at this TMG Element.	* Memory snapshot: Rank 0 572 1666 516 517 1905
See the Report File for TMG Element Associations	<ul> <li>Top C array corresponding sizes (MB):</li> <li>Memory snapshot: Rank 0 201. 101. 53.5 53.5 53.3</li> <li>Memory snapshot: Rank 0 38.0 34.1 33.3 26.2 19.0</li> </ul>
	* Memory snapshot: Rank 0 17.2 16.9 16.6 16.6 16.5
	<ul> <li>* Memory snapshot: Rank 0 13.6 12.4 11.0 10.8 9.98</li> <li>* Updating BCs</li> </ul>
r Tmax At Tmin At TDmax At T(TDmax) Time	
1 569.91 1136432 293.02 1128721 4.61E-03 1404070 293.03 1.0000000E+04	

Iter

### **Inspecting a report file**

This *<simulation/model name>-<analysis name>\_report.log* file contains calculation details, model parameters, stream details, thermal solver created elements, results summary of groups.

Inspect the file to troubleshoot stream junction interdependencies and to review elements created by the thermal solver.

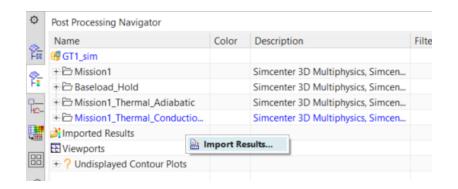
GT1_sim-Mission1_Thermal_Adiabatic	Tue Aug 29 TMG2306.4 8/29/2023
Options for run:	
Module selection parameter=	231
File translation control parameter=	512
Subdivision parameter=	3
Radiative Coupling Threshold=	0.00000E+00
First conductance # for SINDA output=	0
Residual view factor control value=	0
Stefan-Boltzmann constant=	5.66900E-11
Solution is transient with fixed Alpha	
Absolute temperature offset=	0.000
Results output interval=	0.000E+00
Integration time step=	5.000E-01
Start Time=	0.000000E+00
Final Time=	1.000000E+04
Transient Damping Parameter=	1.000E+00
Conductive conductances and capacitance	es will be calculated with CG method
Card 9 PARAMS Cards	
Parameter Card: PARAM UNITS 10 1.0000	
Parameter Card: PARAM ILU 60 1.000000	E-05 100 MAX
Parameter Card: PARAM COND NEW	
Parameter Card: PARAM FEM	
Parameter Card: PARAM CNVGTRA 1.0 Parameter Card: PARAM CSOLVE 2	
Parameter Card: PARAM NLOOP 10000	01
Parameter Card: PARAM TDIFS 1.000000E	
Parameter Card: PARAM PDMAX -1.000000E Parameter Card: PARAM HYDLOOP 100	-02
Parameter Card: PARAM HYDLOOP 100 Parameter Card: PARAM HYDDAMP 1.00000	25.00
Parameter Card: PARAM HYDDAMP 1.00000 Parameter Card: PARAM MAXNODETD 144249	
	L
Parameter Card: PARAM OPPENHEIM 1	
Parameter Card: PARAM QOPPCAL	
Parameter Card: PARAM SPECTRA 0 14387. Parameter Card: PARAM TIMETABLE TRUNCA	
Farameter Card: PARAM LIMETABLE TRUNCA	

Stream: 34 - "HPT_Stream 34"				- 1
- Cluster:	0			
- Inlet Flow Element:	1131564			
- Outlet Flow Element:	1131607			
<ul> <li>Inlet Mass Flow Junction (Calculation):</li> </ul>	0	(A)	0	(B)
- Outlet Mass Flow Junction (Calculation):	0	(A)	0	(B)
- Auto mass flow:	NO			
- Auto reverse mass flow:	NO			
- Inlet Wall Node:	678767	(A)	0	(B)
- Outlet Wall Node:	678768	(A)	0	(B)

			Tempera	ature sur	mmary for	groups		
	Maximum Temp		Minimum Temp		Average Temp	Total Heat in	Total Capacitance	Total Mass
Group:	ASSY glu	ue 2 - Pi	cimary Re	egion				
					683.59	0.00E+00	0.00E+00	0.00E+00
Group:	ASSY_glu	1e_2 - Se	econdary	Region				
	697.07	1127345	674.15	1127340	684.36	0.00E+00	0.00E+00	0.00E+00
Group:	ASSY_glu	1e_3 - P1	cimary Re	egion				
	694.10	1119918	673.61	1119909	681.15	0.00E+00	0.00E+00	0.00E+00
Group:	ASSY_glu	1e_3 - Se	econdary	Region				
	691.47	1127346	673.76	1127349	680.86	0.00E+00	0.00E+00	0.00E+00

# **Inspecting partial .bun file**

During a crash, a partial .bun file may be available. Check the simulation directory for the available .bun file. If not automatically connected to the solution, import the .bun file into post-processing.

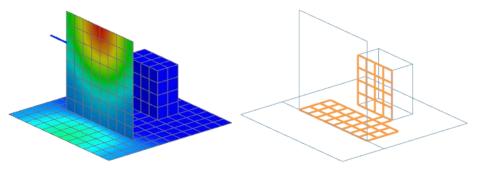




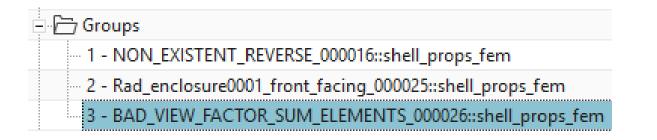
### **Resolving a warning**

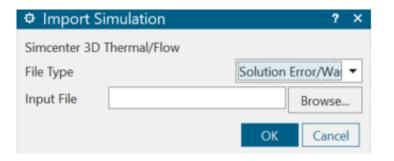
Review the messages in the **Solution Monitor** or the log file for the thermal and flow solvers:

**	WARNING 4316	**						
**	The view fact	or sum err	or of the	following	41	element(s)		
**	exceeds 20%.	The total	number el	ements with	non-zero			
**	view factor s	ums is	267. I	ncomplete e	nclosures	may		
**	exist in the	model. A c	omplete	element lis	t can be			
**	found in file	groups.un	v under t	he group na	me:			
**	BAD_VIEW_FACT	OR_SUM_ELE	MENTS_000	026				
	422	423	424	425	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.

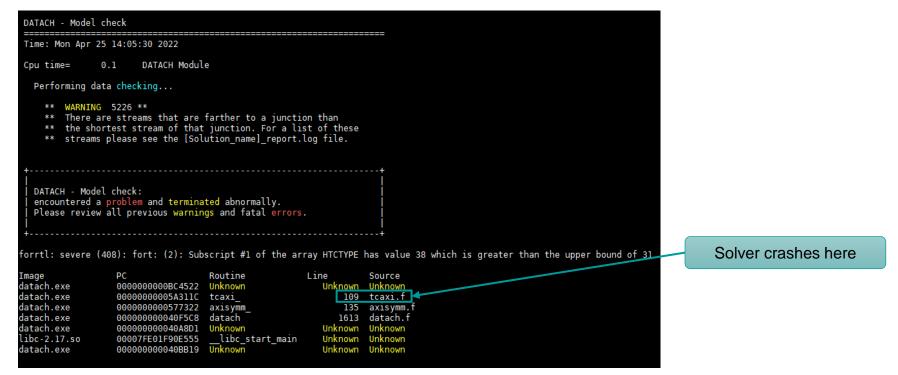






# Applying a traceback patch

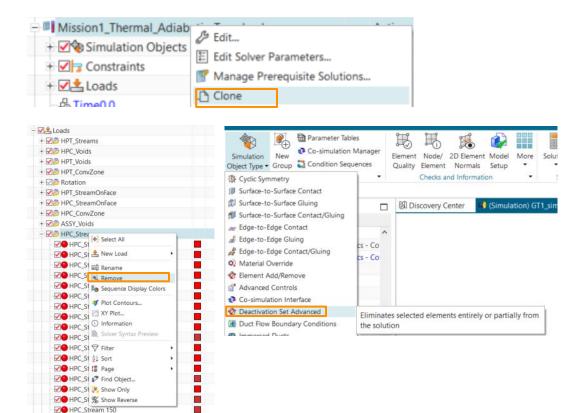
Applying a traceback patch requires referencing a new set of thermal solver files before solving. The log file provides detailed information after a fatal crash, including the code location and the line number where the crash occurred.

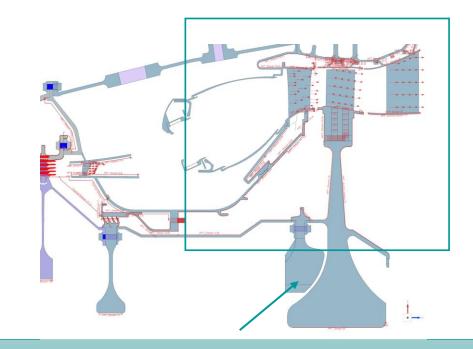




### Simplifying the model

Simplify the model to identify the issue if there is no clear cause for the crash. Clone the problematic solution and remove boundary conditions in sections. For large models, use **Deactivation Set Advanced** to deactivate meshes and reduce time steps to shorten solve time.





Remove downstream boundary conditions in chunks.

If the crash still persists, remove more boundary conditions until you find where the problem is.

HPC Stream 151



# **Thermal- structural solutions troubleshooting**

To troubleshoot thermal-structural analyses, consider the following steps:

- Run thermal and structural model independently before combining them.
- Review the .mplg file, which is a high level log file that provides the status of the thermal and structural solvers. However, it does not provide detailed crash information.
- Refer to the .log file and other files mentioned above for detailed information on the thermal crash.
- Inspect the .f06 file for detailed information on the structural crash.

# Solution Solution Name Solution 1 Solver Simcenter 3D Multiphysics Analysis Type Coupled Thermal-Structural 2D Solid Option None Solution Type Thermal-Structural (SOL 401 Multi-Step Nonlinear) Reference Set Entire Part Automatically Create Step or Subcase

#### Thermal-Structural (SOL 401 Multi-Step Nonlinear)

General	Description	
- Solution Control	Description	
<ul> <li>Structural Parameters</li> </ul>	▼ Solve Options	
- Solution Units		
- Ambient Conditions	Solve Structural	
<ul> <li>Initial Conditions</li> </ul>	✓ Solve Thermal	
- Articulation Parameters	Solve Co-simulation	
- Results Options	Run Job in Foreground	
Restart Management	Values Outside Time-dependent Tables	Constant 💌
	✓ User Defined Text	
	File Management User Defined Text	None 👻 🚰 👻
	Executive Control User Defined Text	None 👻 🚰 👻 💌
	Case Control User Defined Text	None 👻 🚰 👻
	Bulk Data User Defined Text	None 🔻 🚰 👻 🔻
	Thermal Include File	
	Fields File Directory	
	Boundary Condition Control Variables	
< >>	<	>
		OK Apply Cancel

### SIEMENS

# **Thermal mapping troubleshooting**

- It is recommended to use the **Simcenter3D Thermal/Flow** mapping solution.
- Common issues:
  - No mapping Association Zones appear when trying to set the Target Zones.
  - Mapping results show unexpected temperature gradients.
  - FATAL 15018 Target Zone <xxx> intersects target zone <yyy>.

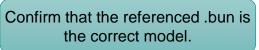


# **Thermal mapping troubleshooting**

Confirm that the source thermal model includes the Association Zones:

- Check for constraints in the source solution.
- Check the source .map or .xml file for constraints. A map file size of ~32 kB likely indicates that no zones were defined.
- Confirm the target zone type matches the assigned source **Association Zones**.

rmal Target Zone 🛛 👌 ? 🗙	Solution	0?
	✓ Solution	
me	Name Solution 1	
	Solver Simcenter 3D Thermal/Flow	
nal Target Zone(1)	Analysis Type Mapping	*
scription	2D Solid Option ZX Plane, Z Axis	*
inpuon	Solution Type Thermal-Flow	*
	Reference Set Entire Part	*
ation Folder	<ul> <li>Thermal-Flow</li> </ul>	
nation Elements	Mapping Details Optional Output	
	✓ Info	
Reference	Description	L.
Focus	Run Job in Foreground	
	✓ Data Source and Destination	
Object (0)	▼ Source Model Results File	_
ided		e
ed		
	Source Model Solution Units Source Model Analysis Type Thermal	
tion Nodes	Source Model Analysis Type Thermal Create Merged Results File	
oonding Source Model Elements	Transient Times	
Update Zones	Select Output Times Unit Select Output Times (0)	-
	Seet Oupst lines (0)	÷
ssociation Zone		Ď
		1 Ch
Thermal Target Zone		
OK Apply Cancel	✓ Settings	
	Temperature Extrapolation Type Limit to Elemen	nt Rang 👻
	Relaxation of Extrapolated Temperature Limits, %(Tmax-Tmin) 0	•
	Temperature Offset 0	∆°F • •



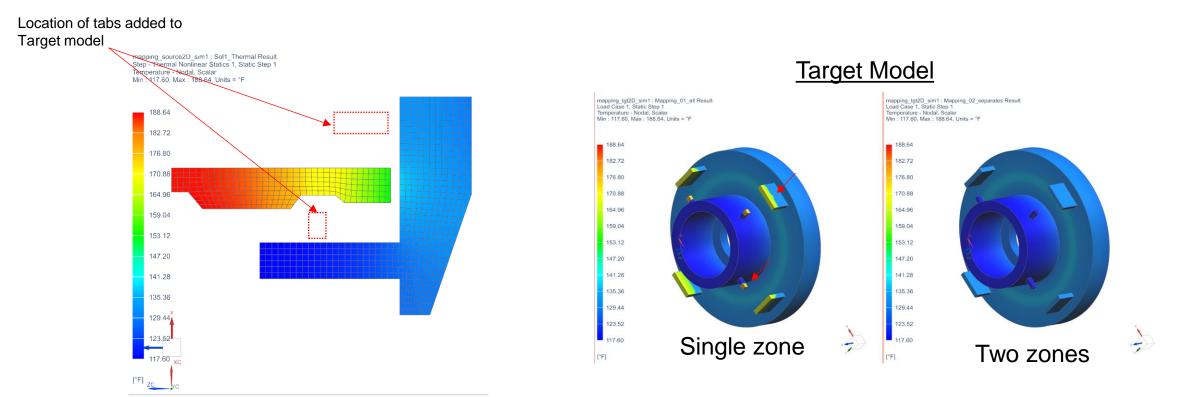
Note: If the only change to the source model is the addition of **Association Zones**, the source .xml and .map files can be regenerated without re-solving the solution.



### SIEMENS

### **Unexpected temperature gradients**

Use association and target zones to guide the mapping solver. If not specified, the solver maps using the nearest source temperature based on proximity. Ensure the source model's association zones cover the desired regions and verify alignment with the correct **2D Solid Option** defined, if applicable.





# **Resolving FATAL 15018 issue**

Mapping target zones cannot overlap.

The solver will issue the FATAL 15018 error if they overlap.

In the Target Zone, use **Destination Nodes** and exclude the face with shared nodes.

	Image: Constraint of the second sec	Axisymmetry Target Zone
	Name	▶ Name
	Destination Folder	Destination Folder
	<ul> <li>Destination Elements</li> </ul>	Destination Elements
		Group Reference
	<ul> <li>Body Focus</li> </ul>	Body Focus
	✓ Select Object (1)	✓ Select Object (0)
	Excluded	Excluded
	✓ Destination Nodes	
		Destination Nodes
	Group Reference	Group Reference
	Body Focus	Body Focus
	✓ Select Object (0)	✓ Select Object (1)
luating Target - Source Association	Excluded	▼ Excluded
	<ul> <li>Corresponding Source Model Elements</li> </ul>	
** FATAL 15018 **	Update Zones	✓ Select Object (1)
	Rotational Periodicity Association Zone RotPer_Upper	✓ Element Selection Filtering
** Target zone ATZ_nd	Rotational Periodicity Association Zone RotPer_Upper	Filter Type Elements
** intersects target zone RTZ_nd	Card Name Rotational Periodicity Target Zone	
** One example of a common node is 1	OK Cancel	Corresponding Source Model Elements
-		Update Zones
		Axisymmetry Association Zone Axi_lower
** Run aborted due to errors.		Axisymmetry Association Zone Axi_lower
** Run aborted due to errors.		Card Name Axisymmetry Target Zone



### **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.

