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
- ✓ Verify elements quality and mesh
- ✓ Verify mesh normals
- ✓ Check for duplicate nodes
- ✓ Do model mass check
- ✓ Verify the element thickness
- ✓ Generate a boundary condition contour plot to display defined quantities
- ✓ Display all thermal couplings
- ✓ View thermal results on model
- ✓ Check log file for errors and warnings
- ✓ 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
- ✓ Check convective thickness and area factors
- ✓ Inspect coupling areas in scratch files
- ✓ Check pressures on walls
- ✓ Perform adiabatic check
- ✓ Run conduction only solution to check thermal contacts
- ✓ Check mass flow and stream nominal directions
- ✓ Check temperature gradients at shutdown conditions
- ✓ Use the traceback patch

Most common issues

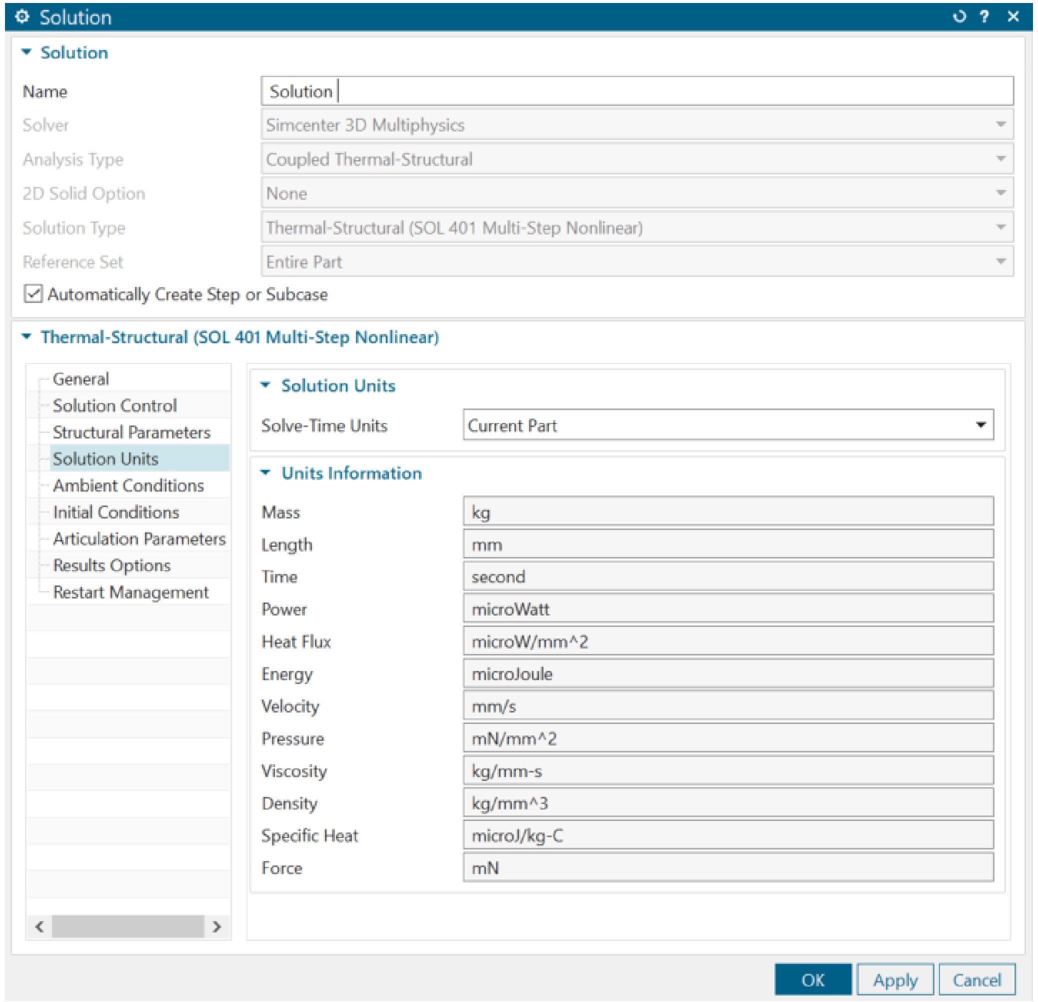
- Units
- Material property assignments
- Poor mesh quality, inadequate resolution near the blades, or insufficient refinement in boundary layers
- Mesh distortion, skewness, and element quality
- Improper boundary conditions
- Thermal coupling setup
- Radiation setup

Verifying units

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

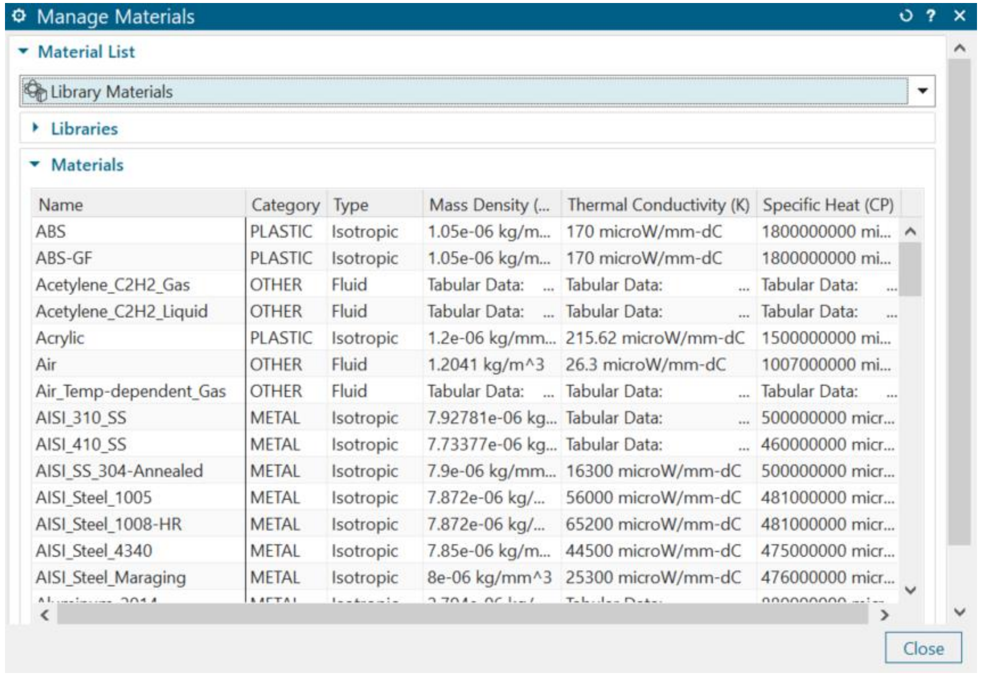
- Check expression for potential issues that could arise from changing units.
- Make units consistent. Select the **List Expression and Units Inconsistency Warnings** customer default to display the warning when inconsistent units are used.
- Do one of the following, since not all units are case insensitive.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

    For more information, see Case sensitivity in expressions.
- Use the **Plot Contours** command to view the values used in the boundary conditions. This applies to non-solver evaluated boundary condition definitions.



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.



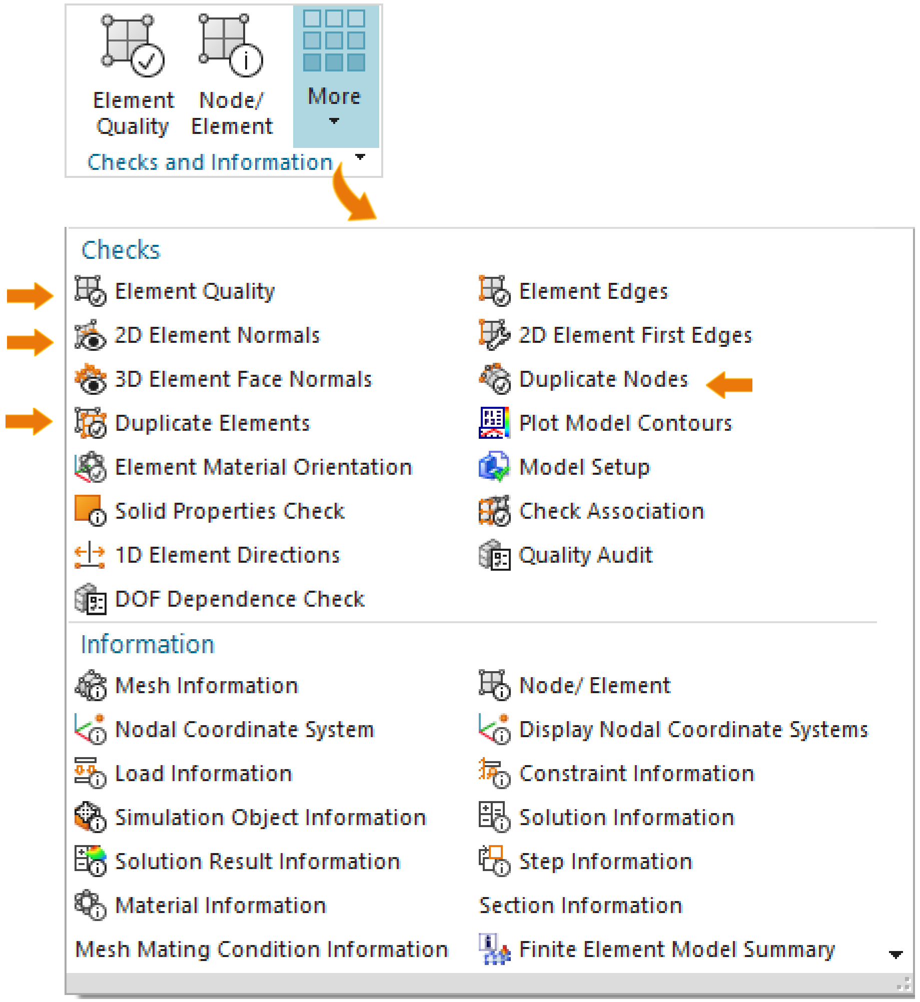
Verifying mesh density

- 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

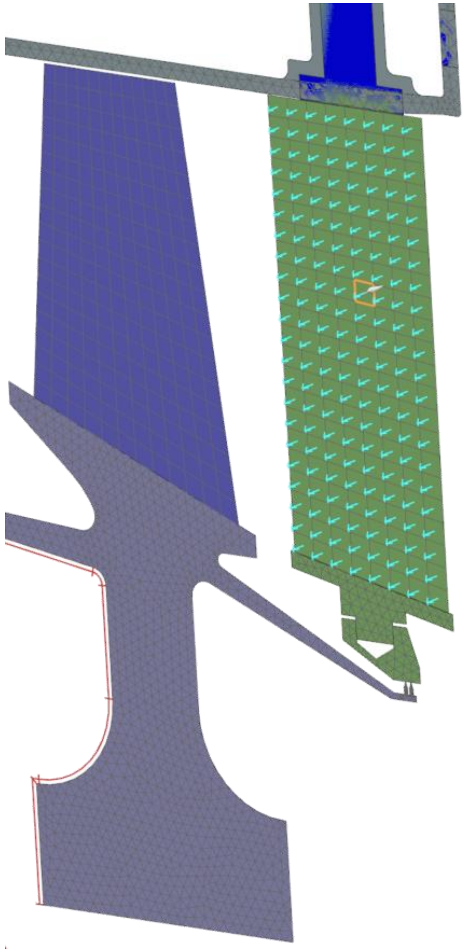
Use the **Element Normals** command to:

- 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:

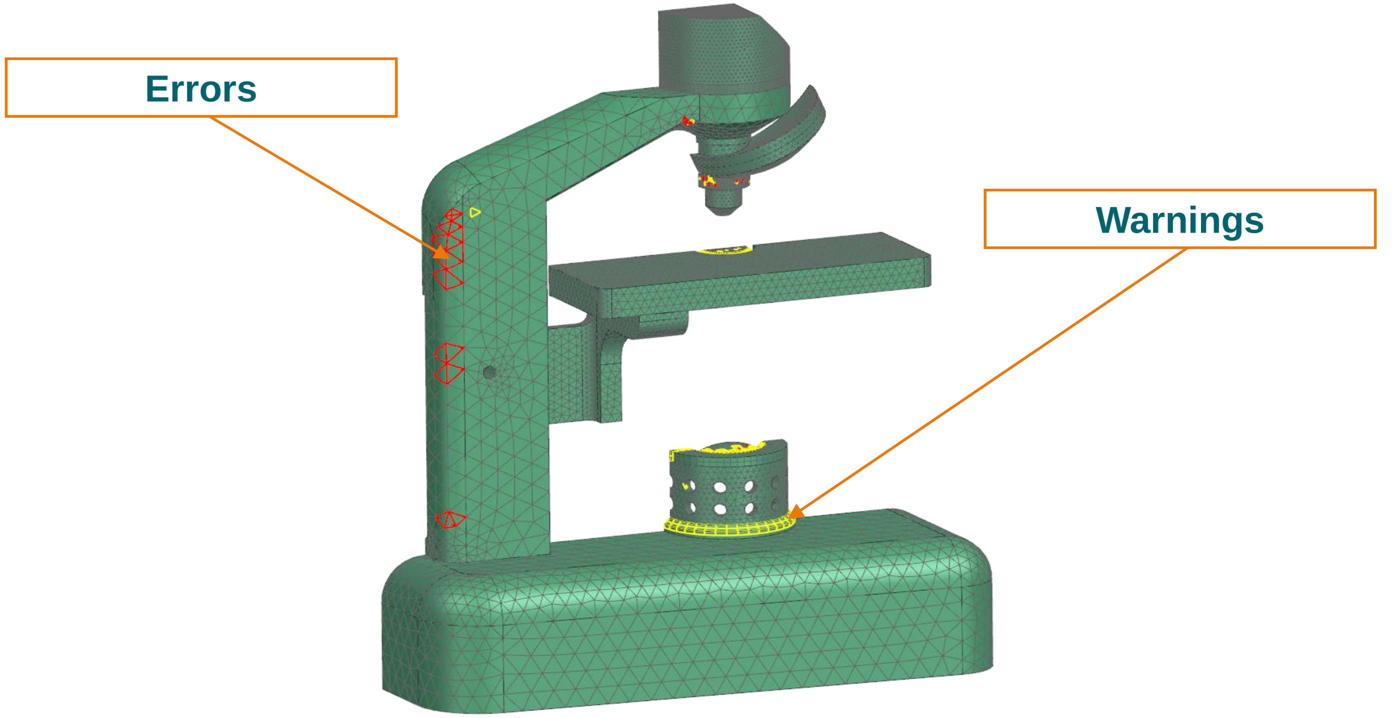
- Properly defining contact between surfaces.
- Properly defining top and bottom in the radiation request.

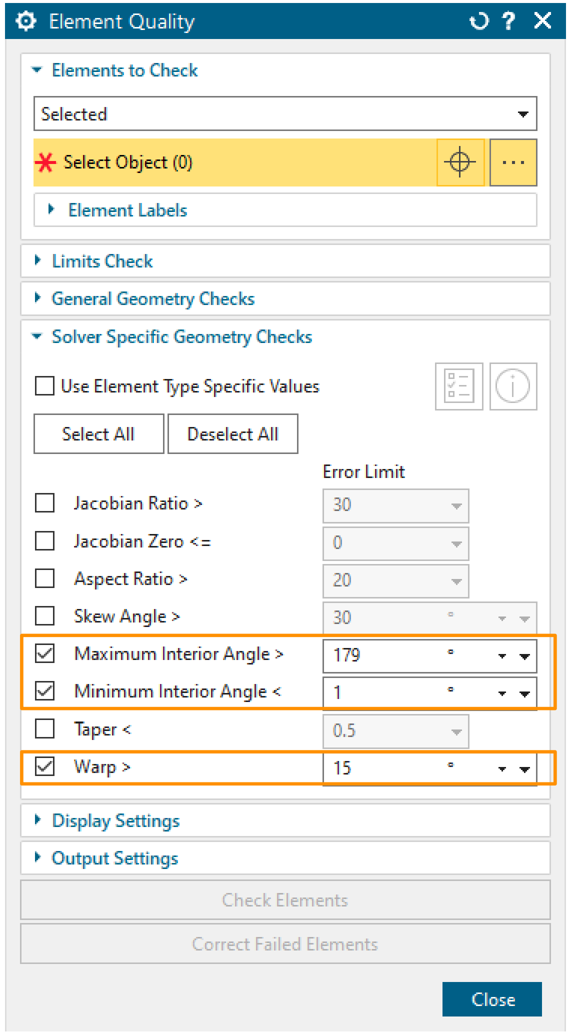


Evaluating element quality

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

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





Identifying coincident nodes

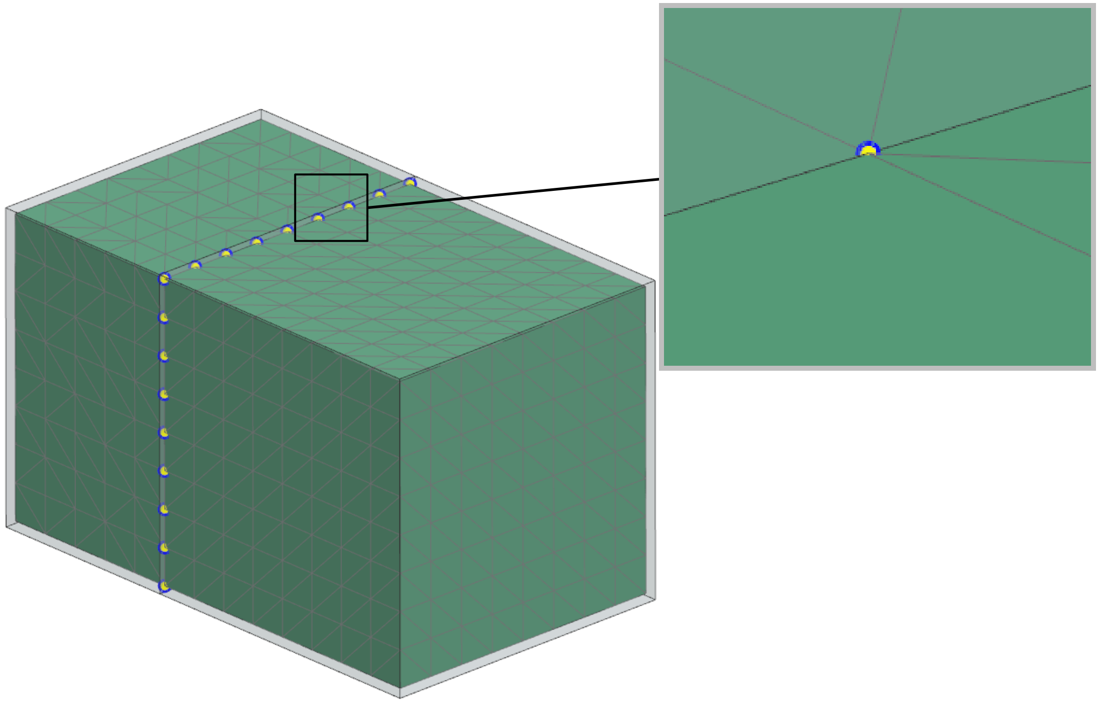
Check for coincident nodes which are duplicate nodes lying on top of each other.

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.

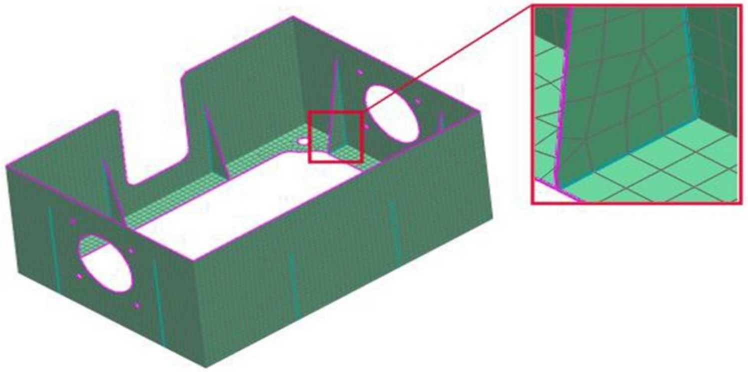
To avoid, check, or resolve duplicate node issues, use the **Mesh Mating Conditions** or **Duplicate Nodes** commands.

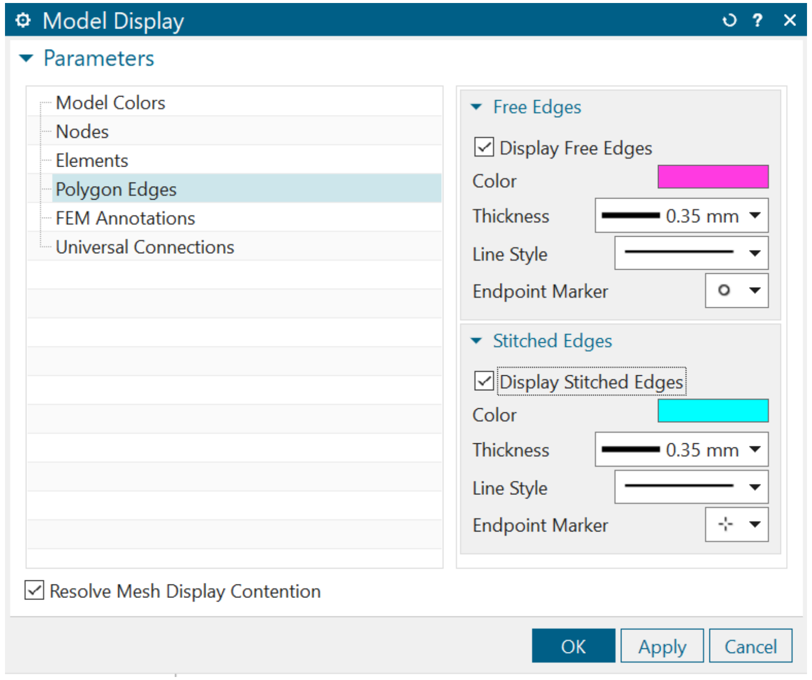
Visual representation of coincident nodes



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.

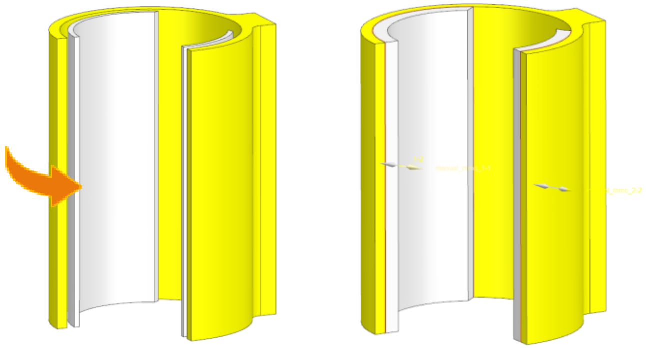


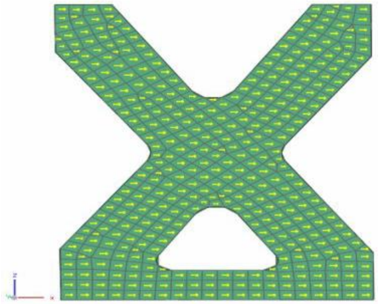


SIEMENS

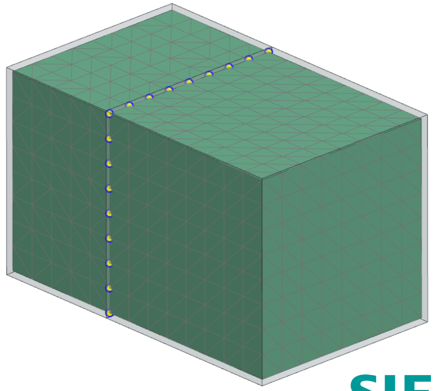
Conducting geometry checks

- 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.

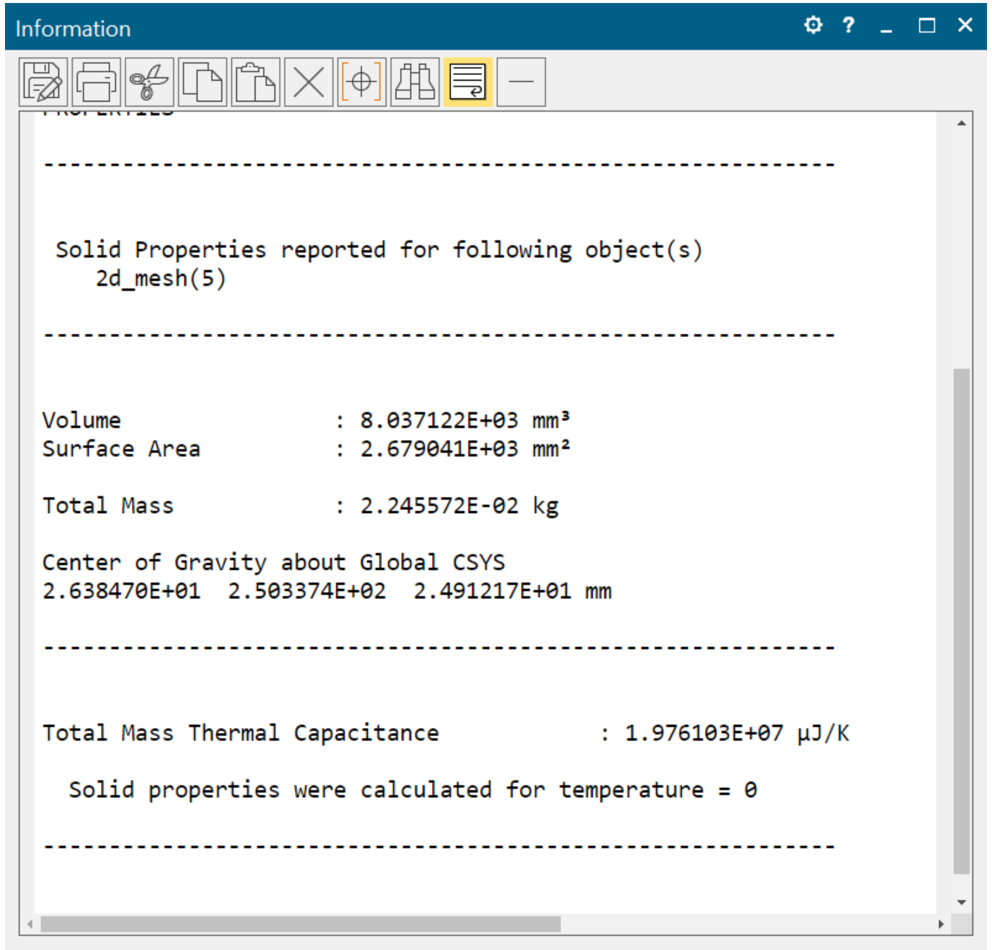
SIEMENS

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

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



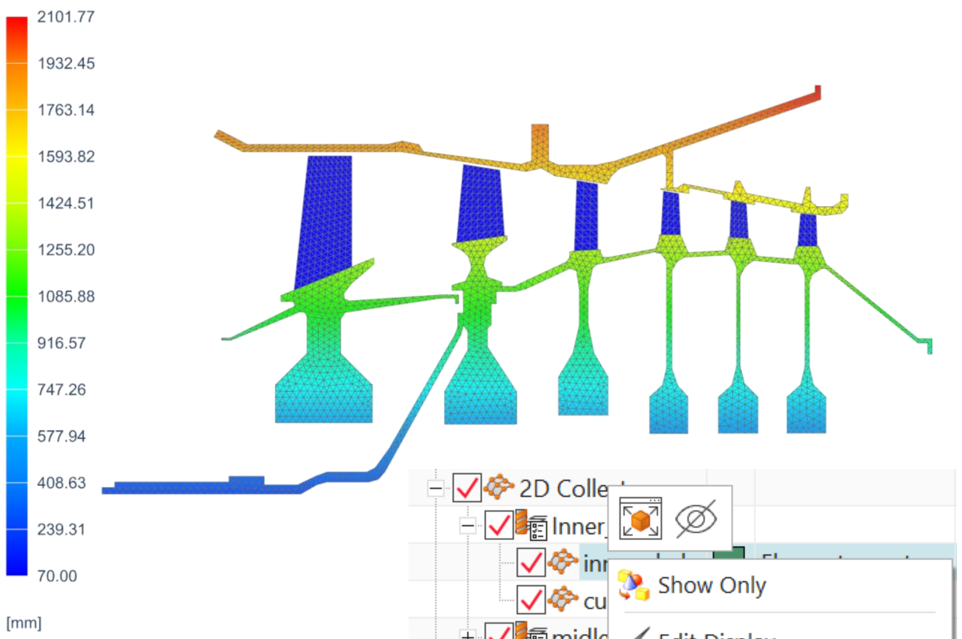
Verifying element thicknesses

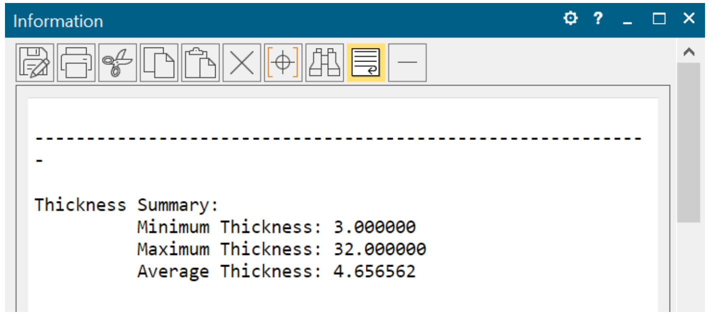
You can use:

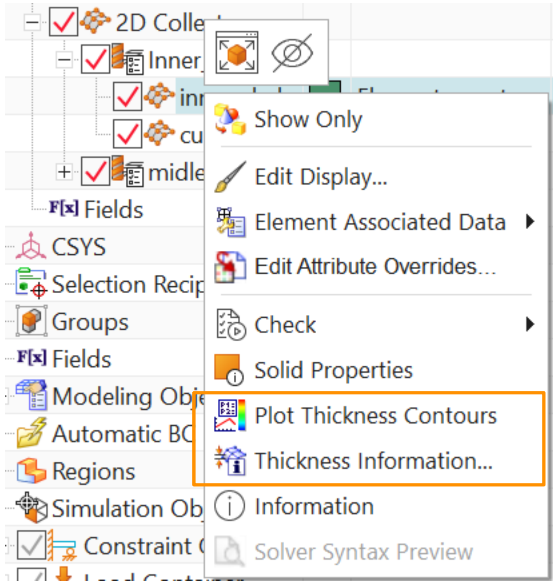
- **Plot Thickness Contours** to generate a contour plot of shell element thicknesses as a standard post view.
- **Thickness Information** to create a color-coded line display that shows the general statistical distribution of the thickness values across your 2D mesh.

You can use the thickness display to quickly identify:

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





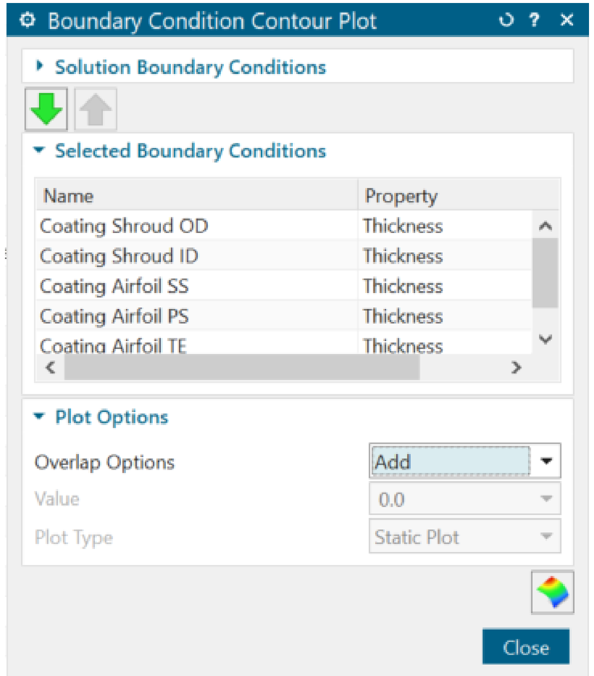


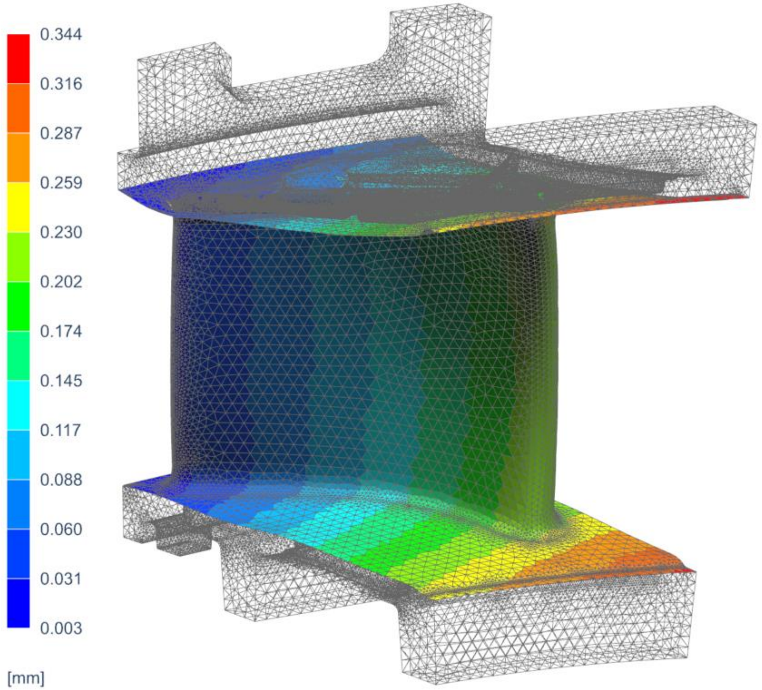
SIEMENS

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.





SIEMENS

Verifying expressions

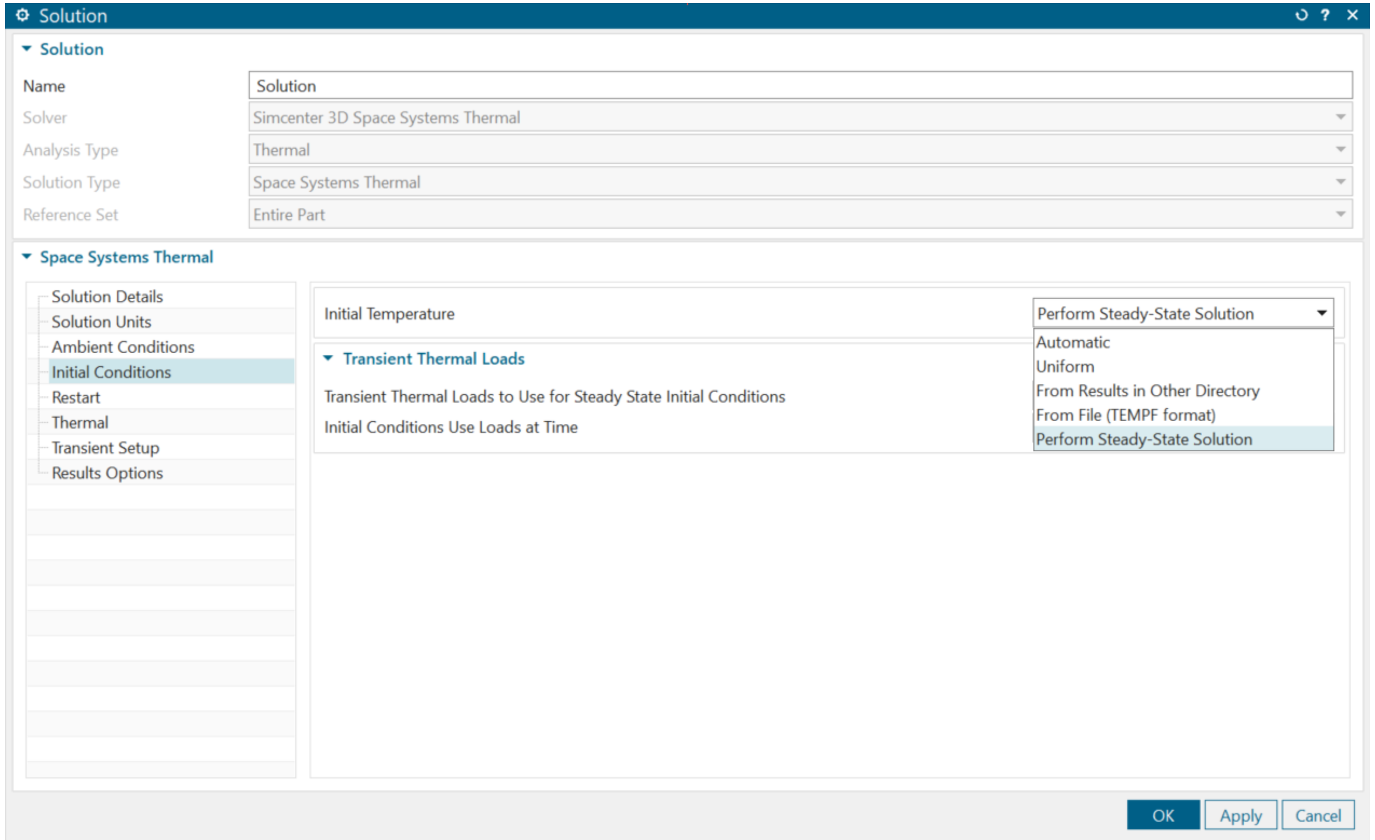
- Check expression logic and units.
- Specify a customer default so that the software issues a warning message about inconsistent units within mathematical functions.
  File →**Utilities** →**Customer Defaults** →**Pre/Post** →**Expressions** →**General** tab, select the **Warn about Inconsistent Units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

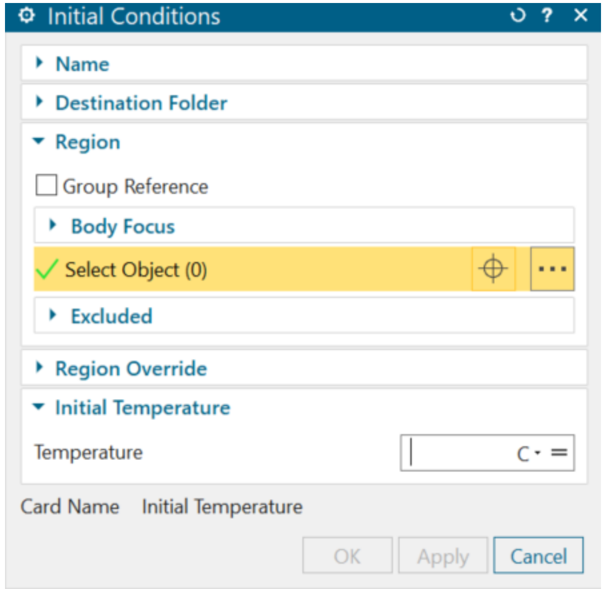
Checking radiation enclosures

- 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.
- Set the local initial conditions in the **Initial Conditions** constraint.
  Local conditions override global conditions.



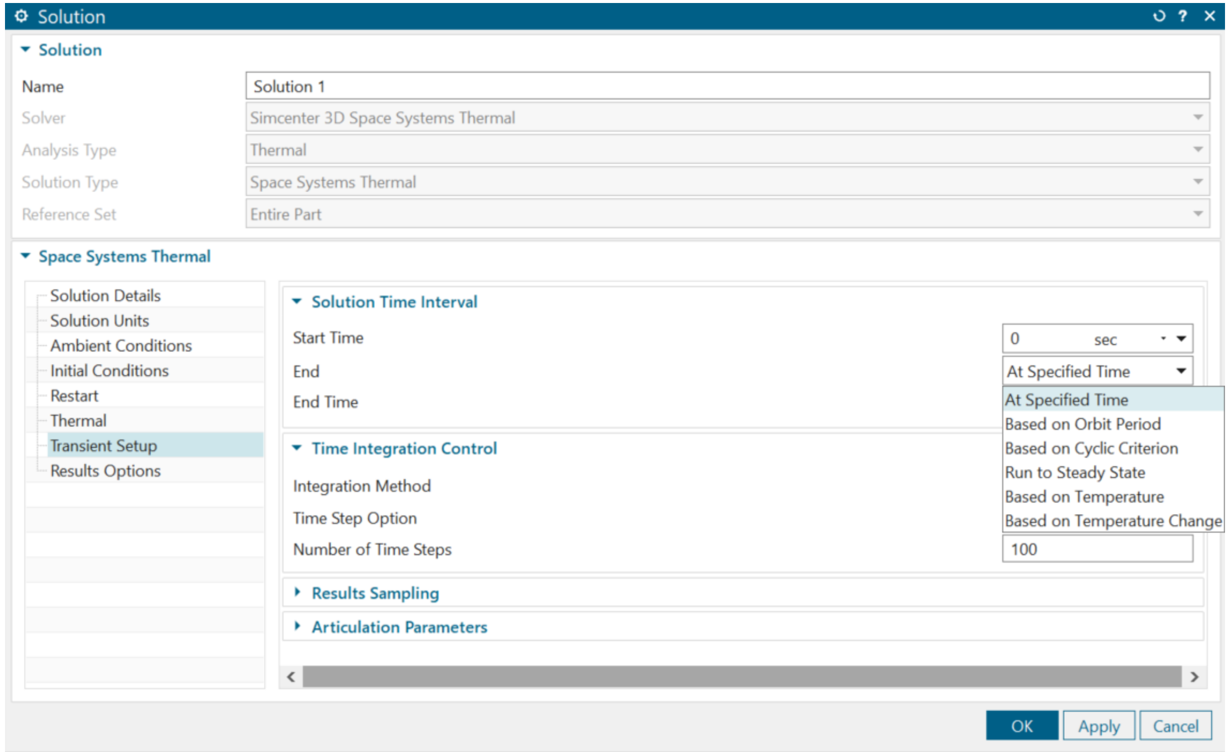


SIEMENS

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.

SIEMENS

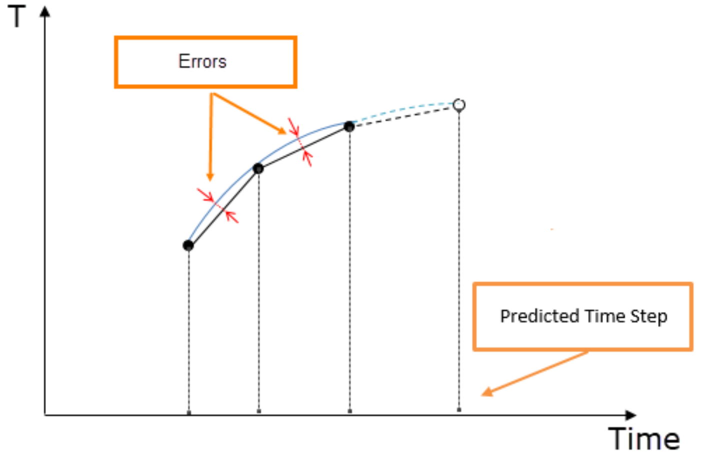
Using automatic time step

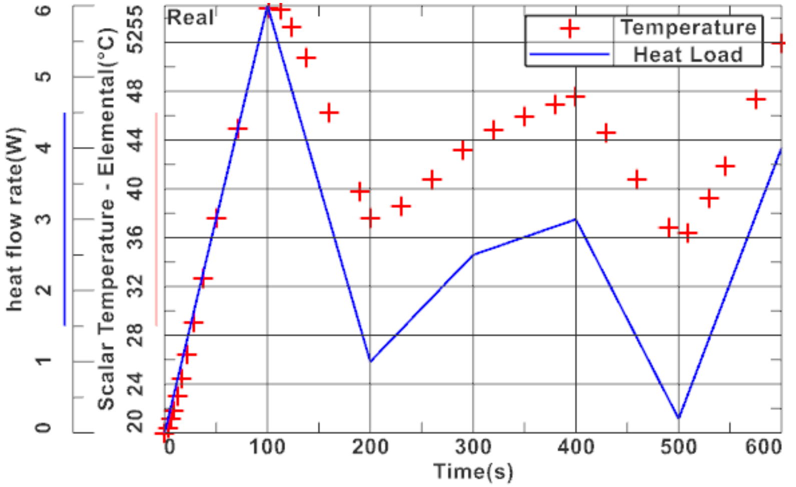
One way to speed up a solve is to use automatic time stepping.

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.



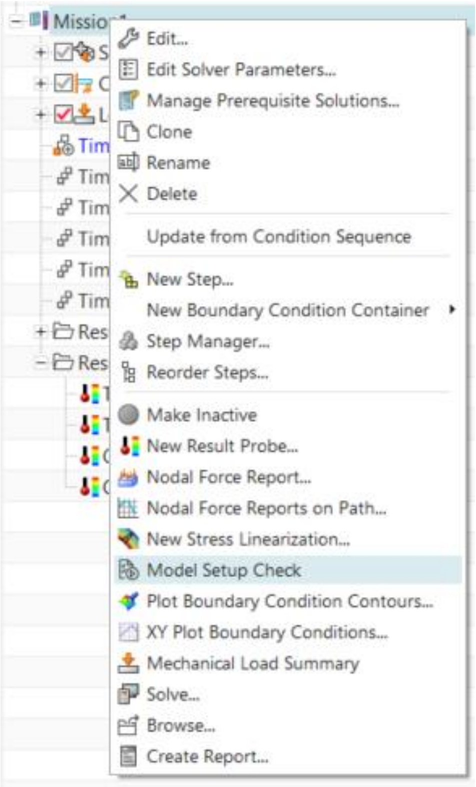


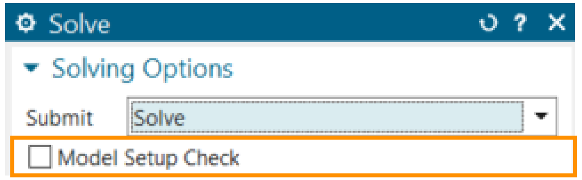
Running a model setup check

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.

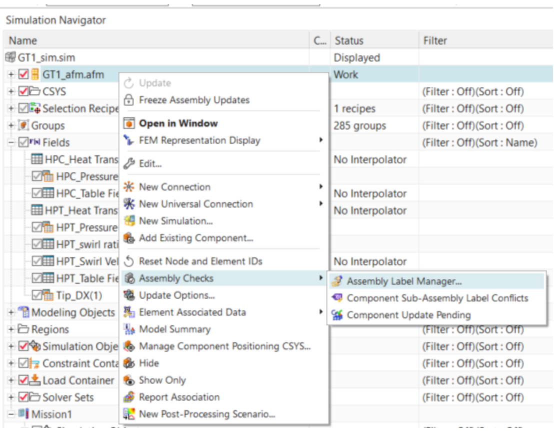


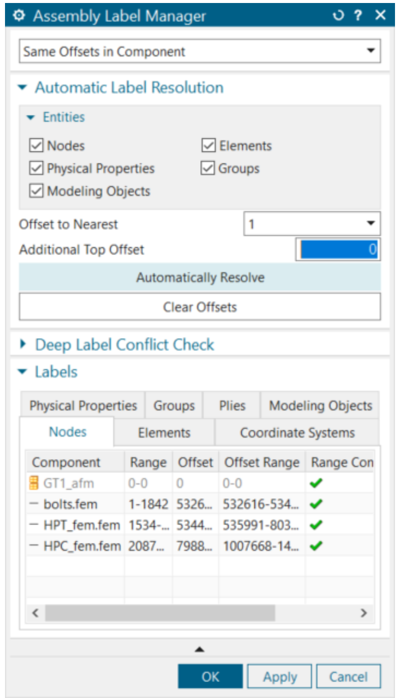


Resolving ID conflicts

AFM label conflicts

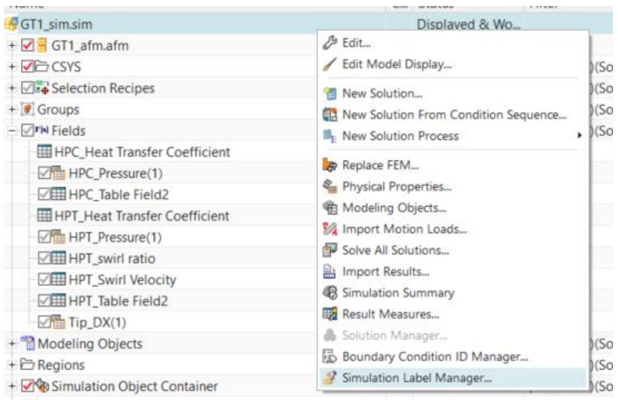
Right-click the active assembly FEM file → **Assembly Checks → Assembly Label Manager**

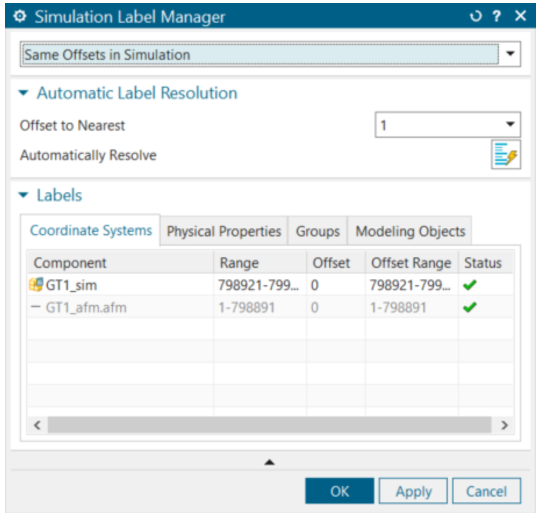




Simulation label conflicts

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

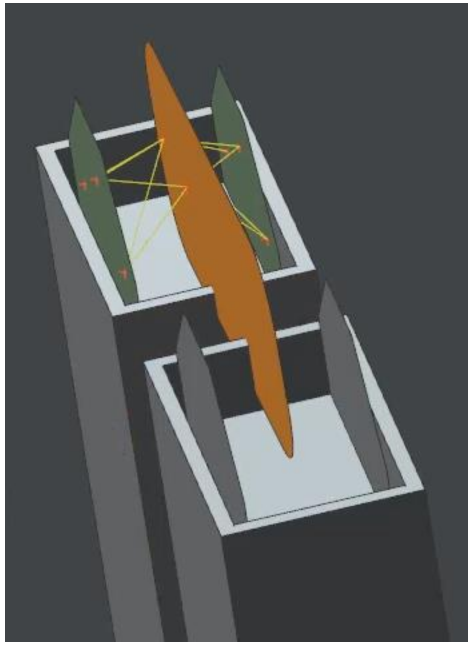


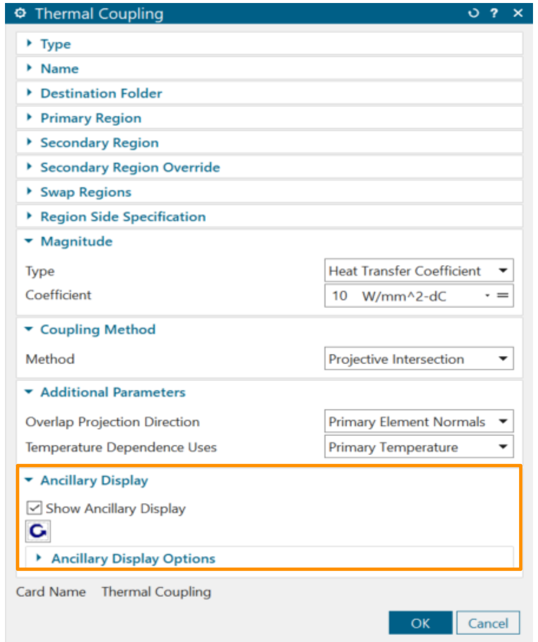


SIEMENS

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.





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.

G13

	A	B	C	D
1	Name	Description	Type	Value
2	TC-Hotbox-2	3.5mm bolts, small stiff surface 2.37 C/W x2	Total Conductance	1.68776371308017 W/°C
3	Regulator-PCB	Perfect contact	Total Conductance	1000 W/°C
4	Regulator-PCB-2	Perfect contact	Total Conductance	1000 W/°C
5	Regulator-PCB-3	Perfect contact	Total Conductance	1000 W/°C
6	Regulator-PCB-4	Perfect contact	Total Conductance	1000 W/°C
7	GapPad	Sil-pad	Conductive Gap	0.9 W/(m*K)
8	PCDU-PCB-HeatSink	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C
9	PCDU-PCB-HeatSink(2)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C
10	PCDU-PCB-HeatSink(3)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C
11	PCDU-PCB-HeatSink(4)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C
12	PCDU-PCB-HeatSink(5)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C
13	PCDU-PCB-HeatSink(6)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C
14	PCDU-PCB-HeatSink(7)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C
15	PCDU-PCB-HeatSink(8)	3.5mm bolts 2.37C/W	Total Conductance	1.687 W/°C

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

Read File

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.**

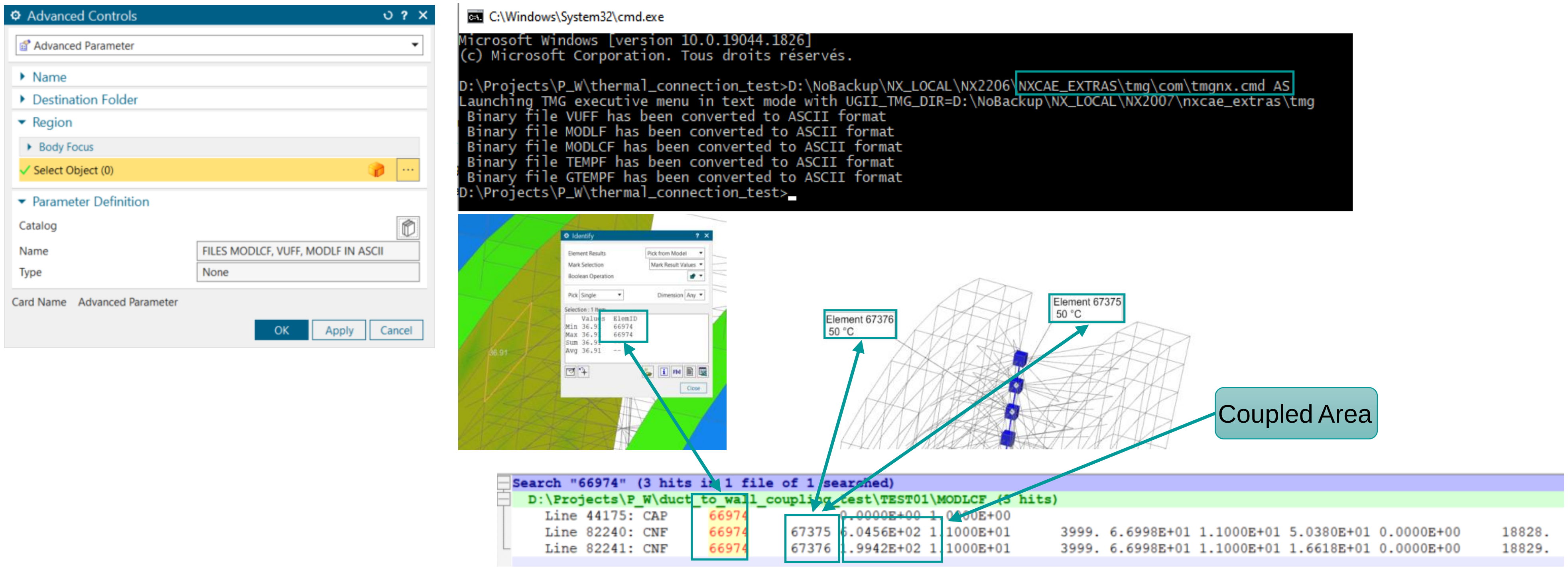
SIEMENS

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:

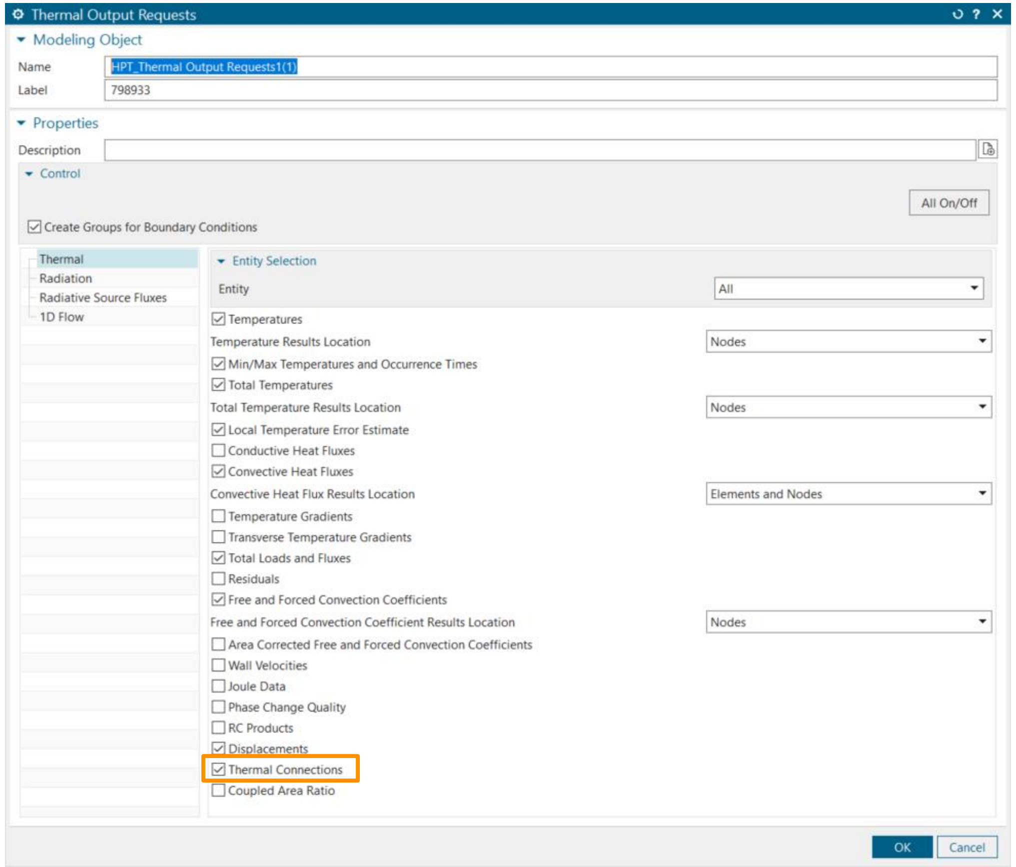
- Using the executive menu command.
- Specifying a FILES MODLCF, VUFF, MODLF IN ASCII advanced parameter.

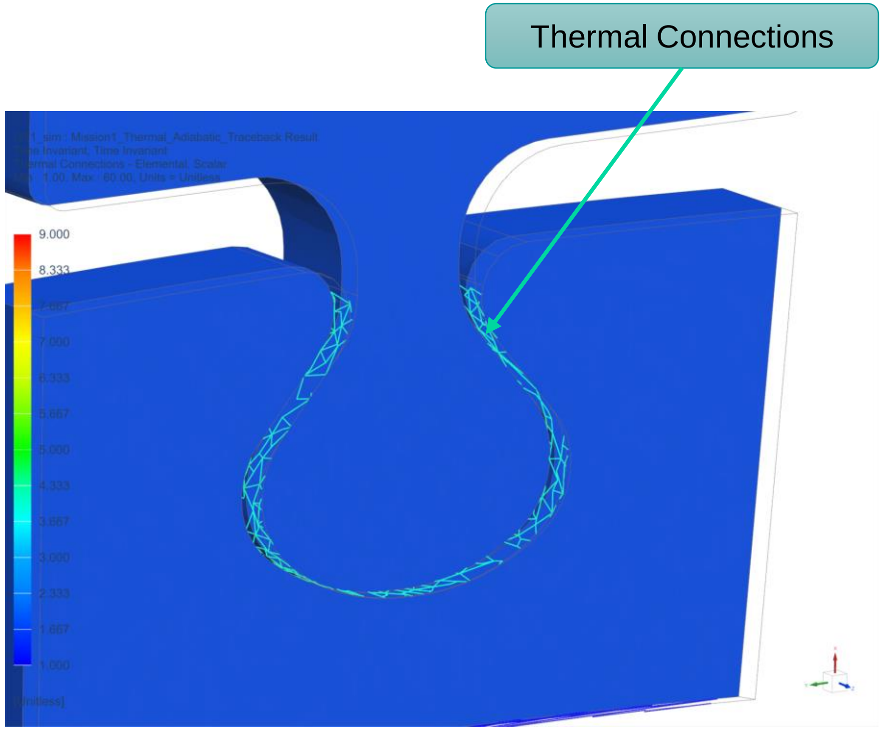
From the MODLCF file below, surface element 66974 is connected to elements 67375 and 67376.



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.



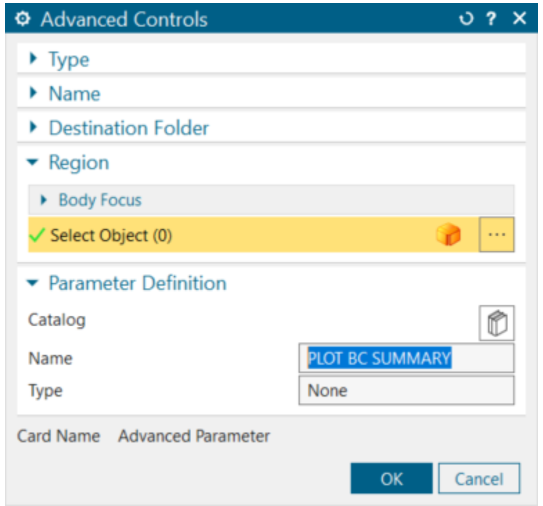


SIEMENS

Using the plot bc summary

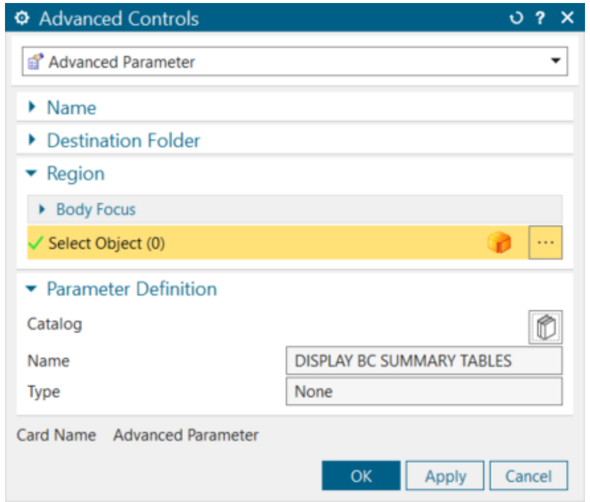
To monitor important areas of your model at run-time, you create **Advanced Controls** with the PLOT BC SUMMARY advanced parameter in the solution. The thermal solver generates the <*simulation name*>-<*solution name*>*data.html* file where you can inspect various result quantities associated to boundary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



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 convective area.



VOID BC DATA

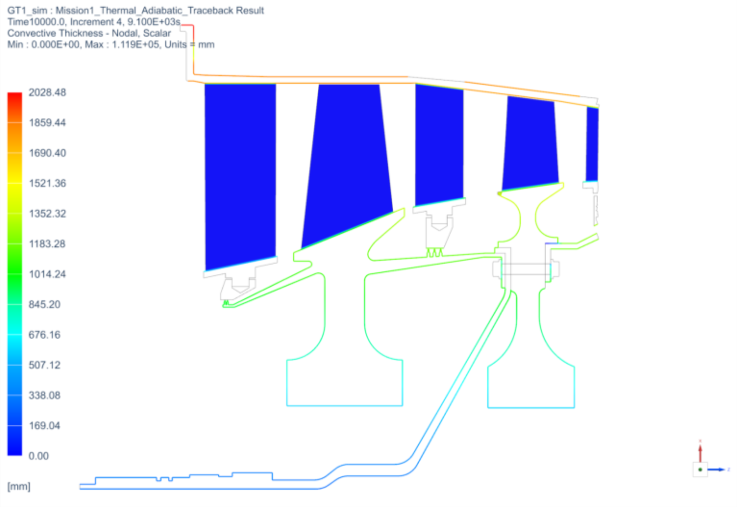
ID	Reg.#	TIME	Tmax	Pmax	THR	Trel-max	CHR	AREAC	SVmax
1.	1.	8200.	393.1	0.2464	-0.2997E-01	393.1	-0.2997E-01	0.2752E+06	0.0000E+00
2.	1.	8200.	393.1	0.2464	-0.1892E-01	393.1	-0.1892E-01	0.6561E+05	0.0000E+00
3.	1.	8200.	393.1	0.2464	0.1519E-01	393.1	0.1519E-01	0.7295E+05	0.0000E+00
4.	1.	8200.	393.1	0.2464	-0.2271E-01	393.1	-0.2271E-01	0.5865E+05	0.0000E+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 REGIONS		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 REGIONS		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 REGIONS		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
	3.	8200.	303.2	0.1111	0.7991E-13	301.9	0.7991E-13	0.1308E+06	0.1221E+06
ALL REGIONS		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 REGIONS		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 REGIONS		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

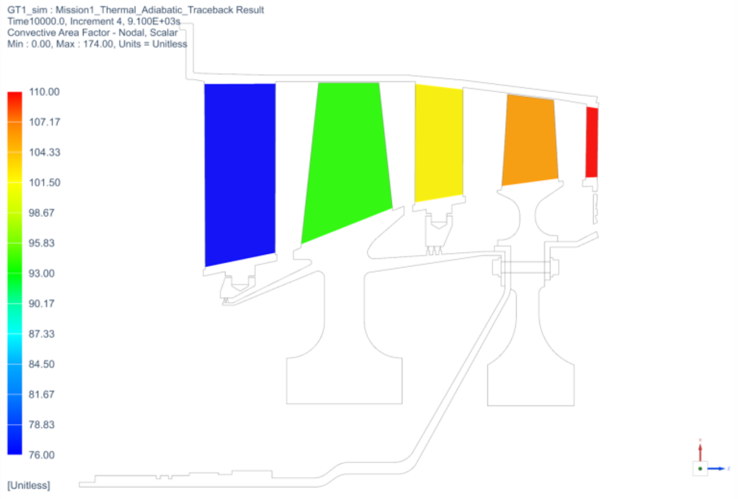
Confirm convective area of boundary conditions

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 2π 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 2π times the radius minus the specified thickness times the number of instances.

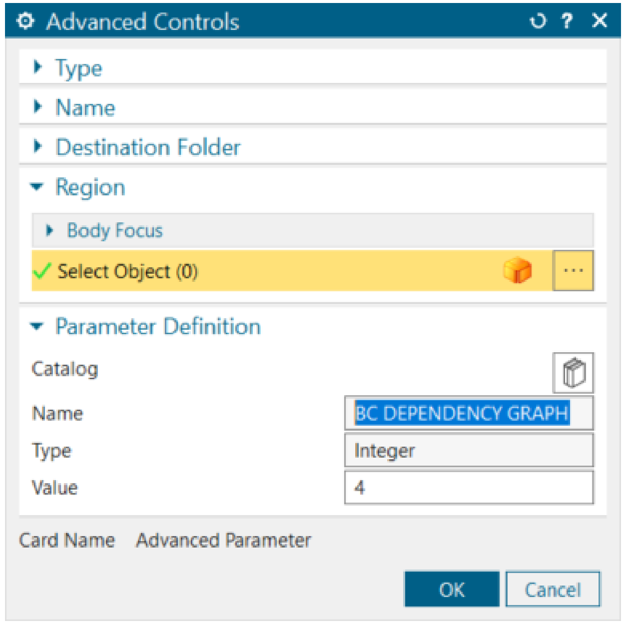


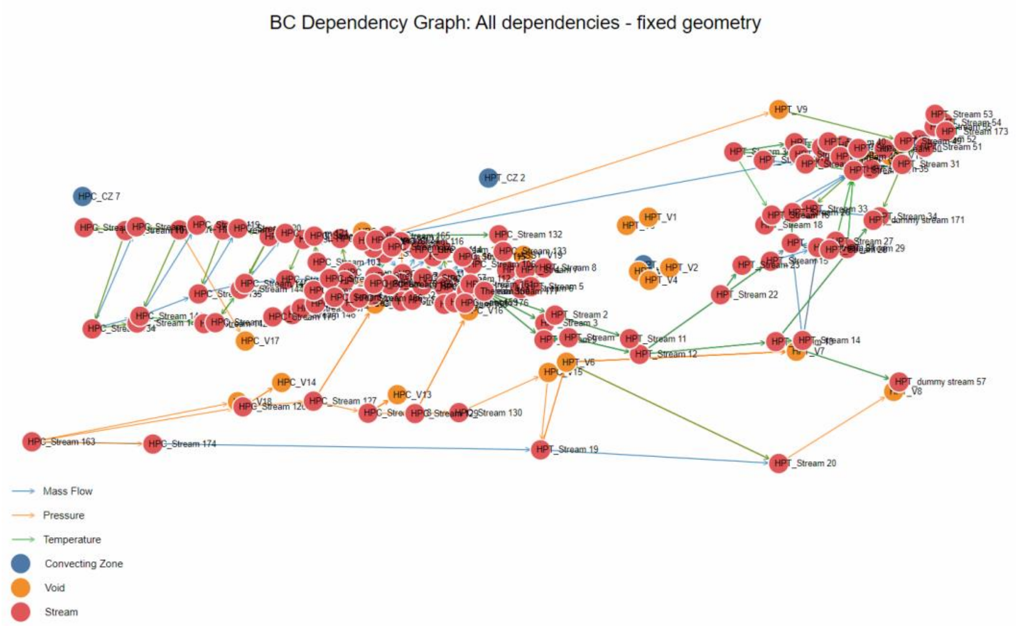


SIEMENS

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.



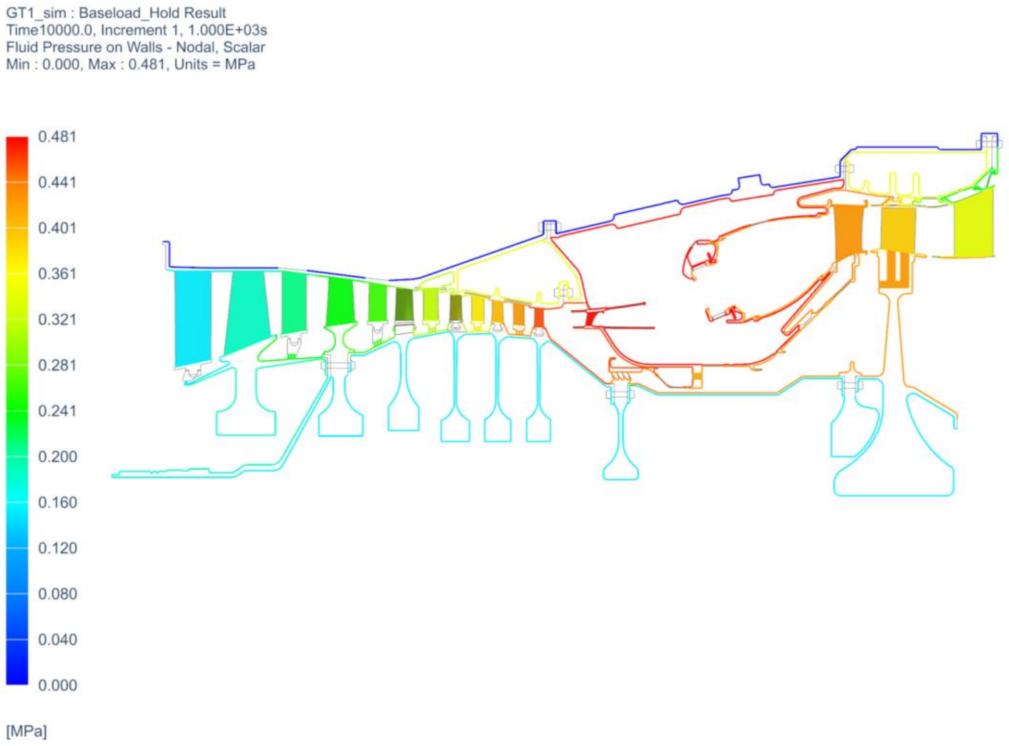


SIEMENS

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.

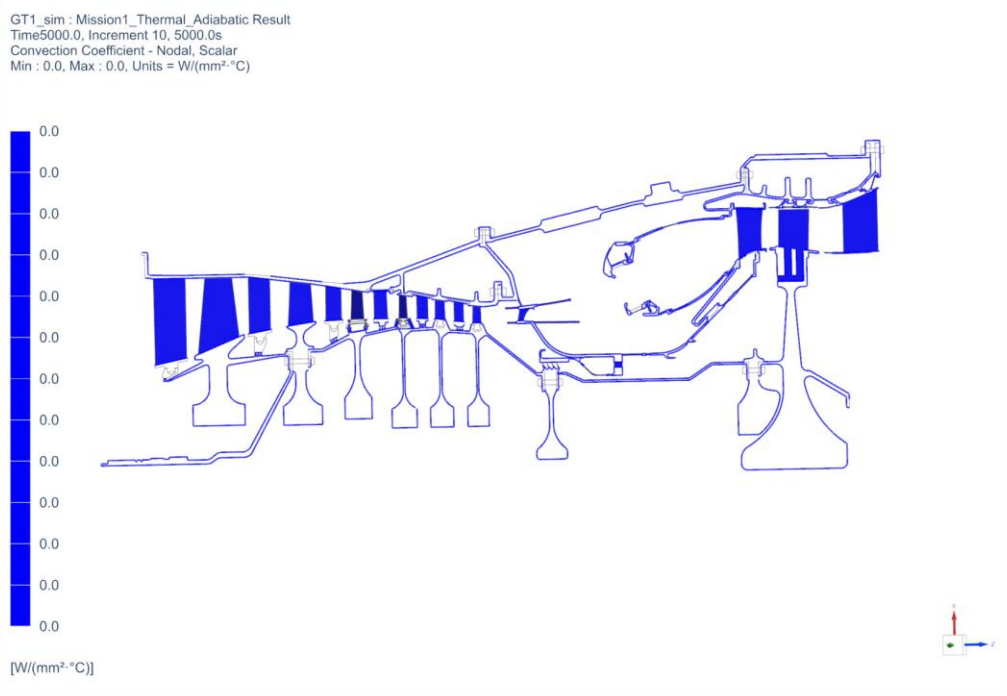


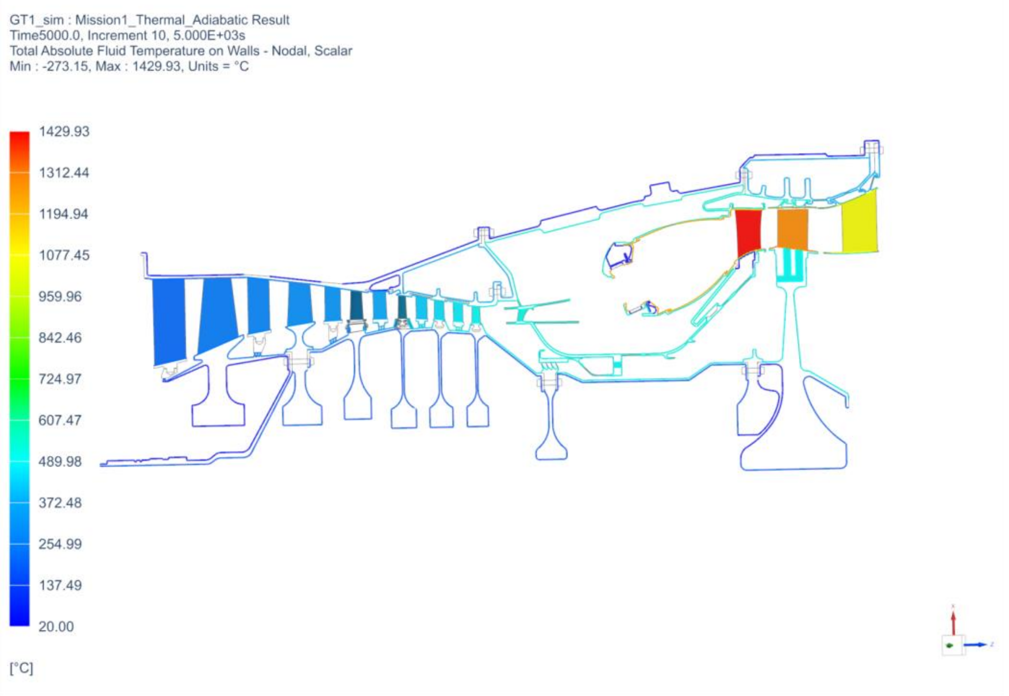
SIEMENS

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.

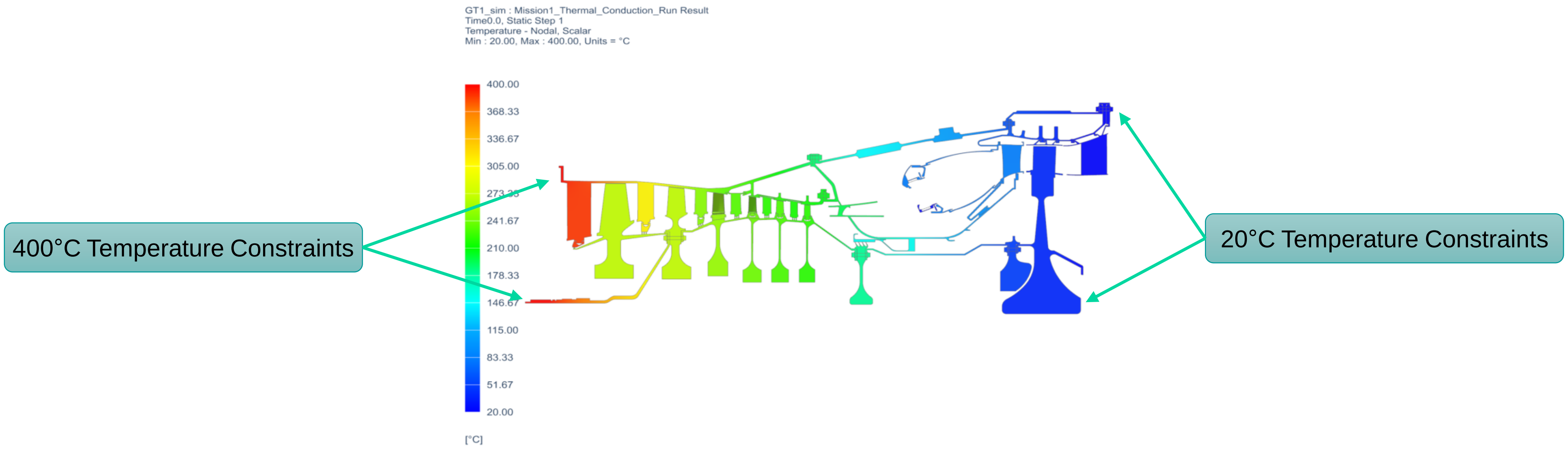




Running a conduction only simulation

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

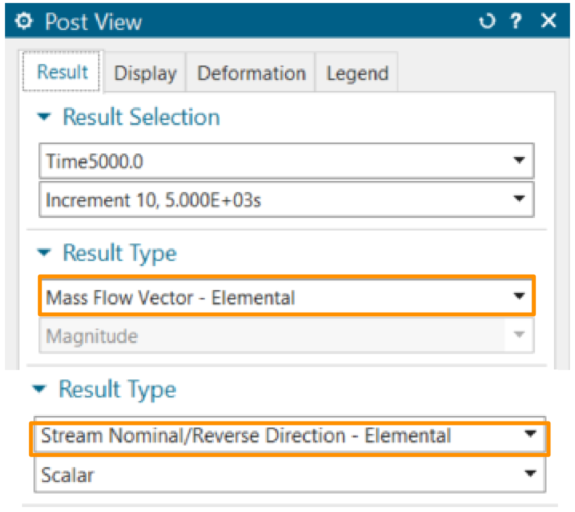
This allows you to confirm that thermal contacts are correctly modeled and serves as a check for specific heat in transient runs.

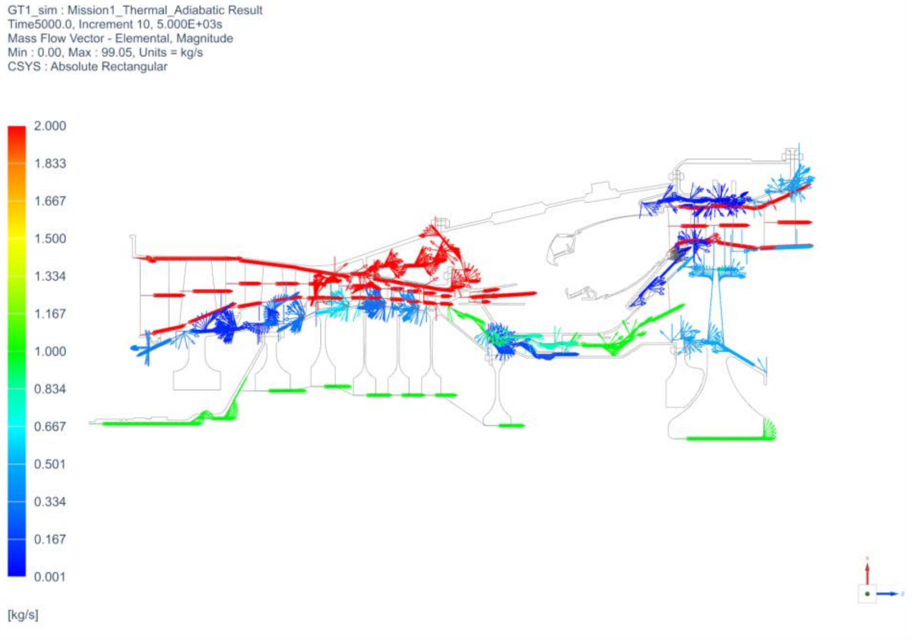


Checking mass flow and stream nominal directions

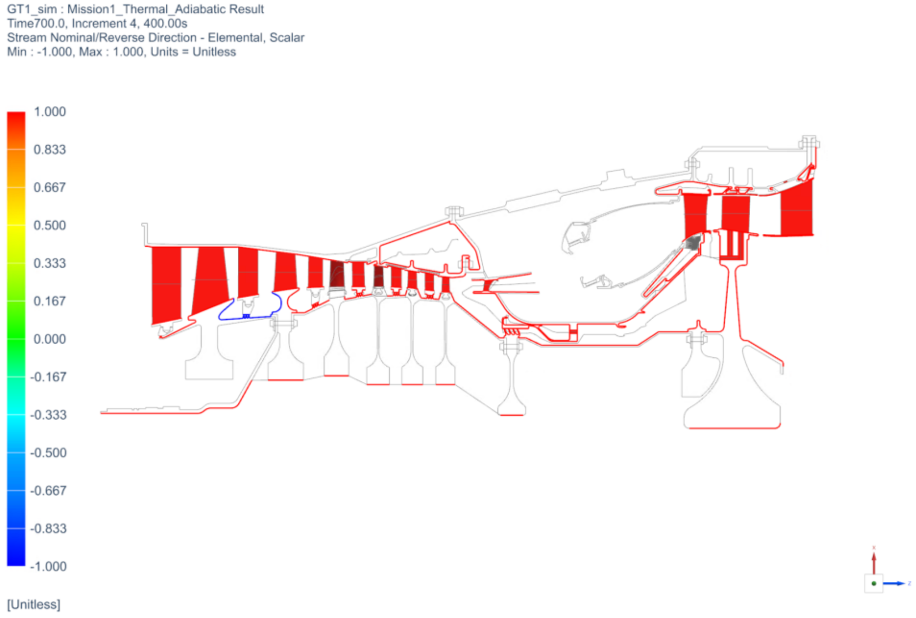
You can display:

- **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).





Mass Flow Vectors

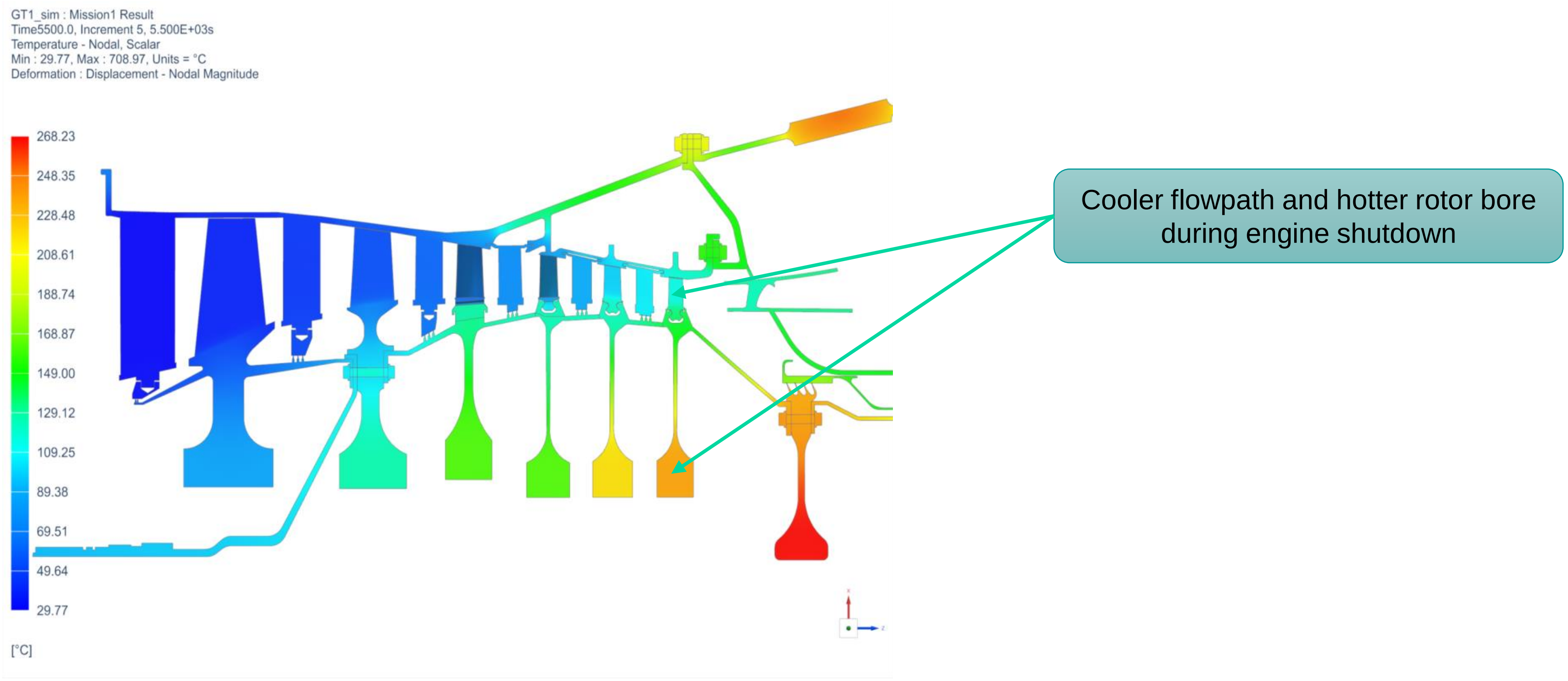


Stream Nominal Reverse Directions

SIEMENS

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.

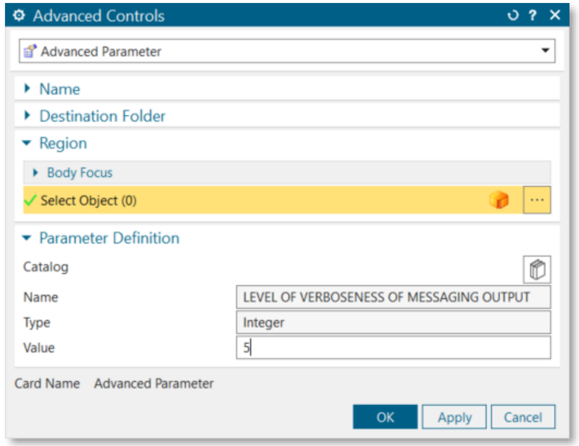
For more information about the files, see Overview of thermal solver files and how to use them (https://support.sw.siemens.com/en-US/product/289054037/knowledge-base/KB000128451_EN_US).

```
Time=  10000.0000000     Integration timestep=  900.000

Cpu time in ANALYZER module=  222.5


Minimum temperature                 =  288.150 at element   1296635

Maximum temperature                 =  569.913 at element   1136454

Average temperature                 =  295.980




Heat Flow+Load Summary Into Different Sink Entities:

       Sink Entity                              Temperature     Heat    Energy absorbed

                                                              Flow+Load   since start


HPT_Duct_Inlet_Temp_1                             3.933E+02  -4.694E-07  -3.995E+01

HPT_Duct_Inlet_Temp_2                             3.931E+02  -5.655E-10   3.941E-05

Sink elements with no entity names:               3.673E+02  -7.620E+07  -5.311E+12


  ...done.



+----------------------------------------------------------------------+
|                                 END                                  |
+----------------------------------------------------------------------+

Solution elapsed time: 05 min 06 sec

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


SIEMENS

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.




```
   Time=  50.0000000000     Integration timestep=  50.0000
   Cpu time in ANALYZER module=  24.11

   Minimum temperature                    =   288.150 at element   1296635
   Maximum temperature                    =   569.913 at element   1136454
   Average temperature                    =   296.028

 Heat Flow+Load Summary Into Different Sink Entities:
        Sink Entity                          Temperature     Heat   Energy absorbed
                                                          Flow+Load   since start

 HPT_Duct_Inlet_Temp_1                         3.933E+02  -3.327E-04  -1.664E-02
 HPT_Duct_Inlet_Temp_2                         3.931E+02   0.000E+00   0.000E+00
 Sink elements with no entity names:           3.673E+02  -7.620E+07  -3.802E+09
ILU iteration    1 Residual=  7.78E-07
ILU iteration    1 DTmax=  1.22E-02 at         0


Iter     Tmax      At     Tmin      At    TDmax       At  T(TDmax)     Time

   1   569.91 1136432  293.13 1304180 8.06E-03 1109770   293.14   1.0000000E+02

*   Printing out thermal solve results for time=   0.100000E+03
*     Memory snapshot: Before results printout:
*     Memory snapshot: Peak Virtual Memory (MB), Peak Physical Memory (MB), Allocated C arrays (MB), Top 20 C arrays (MB)
*     Memory snapshot: Rank   0  0.000E+00 0.000E+00 0.124E+04  755.
*     Top C array labels:
*     Memory snapshot: Rank   0       901       903      1767      1768      1772
*     Memory snapshot: Rank   0       905      1769      2607       176       907
*     Memory snapshot: Rank   0       422      1654      2609      1658       835
*     Memory snapshot: Rank   0       572      1666       516       517      1905
*     Top C array corresponding sizes (MB):
*     Memory snapshot: Rank   0   201.      101.      53.5      53.5      53.3
*     Memory snapshot: Rank   0   38.0      34.1      33.3      26.2      19.0
*     Memory snapshot: Rank   0   17.2      16.9      16.6      16.6      16.5
*     Memory snapshot: Rank   0   13.6      12.4      11.0      10.8      9.98
*     Updating BCs
```

Max error in iteration occurs at this TMG Element.
See the Report File for TMG Element Associations

```
Iter     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
```


SIEMENS

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=                              0.0000000E+00
Final Time=                              1.0000000E+04
Transient Damping Parameter=                 1.000E+00
Conductive conductances and capacitances will be calculated with CG method
                   Card 9 PARAMS Cards

Parameter Card: PARAM UNITS 10  1.000000E+03  1.000000E+00  1.000000E+00  0.
Parameter Card: PARAM ILU 60  1.000000E-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
Parameter Card: PARAM TDIFS  1.000000E-01
Parameter Card: PARAM PDMAX -1.000000E-02
Parameter Card: PARAM HYDLOOP 100
Parameter Card: PARAM HYDDAMP  1.000000E+00
Parameter Card: PARAM MAXNODEID 1442491
Parameter Card: PARAM OPPENHEIM 1
Parameter Card: PARAM QOPPCAL
Parameter Card: PARAM SPECTRA 0 14387.69 0  3.000000E+00 0 0 0 0
Parameter Card: PARAM TIMETABLE TRUNCATE
```

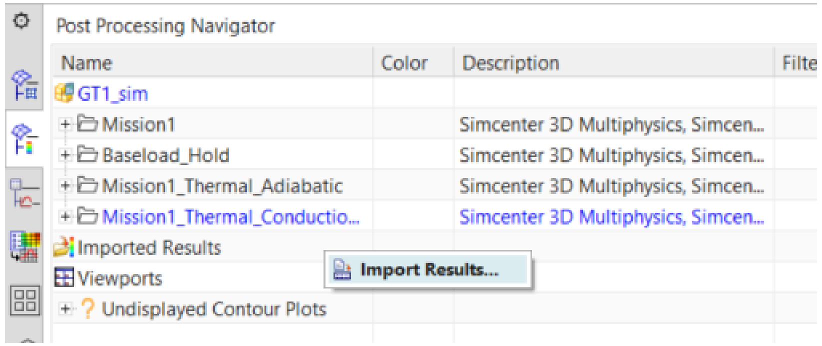
```
Stream: 34 - "HPT_Stream 34"
  - Cluster:                                           0
  - Inlet Flow Element:                          1131564
  - Outlet Flow Element:                         1131607
  - Inlet Mass Flow Junction (Calculation):            0 (A)        0 (B)
  - Outlet Mass Flow Junction (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)
```

```
                          Temperature summary for groups

          Maximum    at    Minimum    at    Average    Total      Total      Total
            Temp  element   Temp   element   Temp     Heat in  Capacitance   Mass

Group: ASSY_glue_2 - Primary Region
           696.41 1127321   674.19 1127329   683.59    0.00E+00    0.00E+00  0.00E+00
Group: ASSY_glue_2 - Secondary Region
           697.07 1127345   674.15 1127340   684.36    0.00E+00    0.00E+00  0.00E+00
Group: ASSY_glue_3 - Primary Region
           694.10 1119918   673.61 1119909   681.15    0.00E+00    0.00E+00  0.00E+00
Group: ASSY_glue_3 - Secondary Region
           691.47 1127346   673.76 1127349   680.86    0.00E+00    0.00E+00  0.00E+00
```


SIEMENS

Inspecting partial .bun file

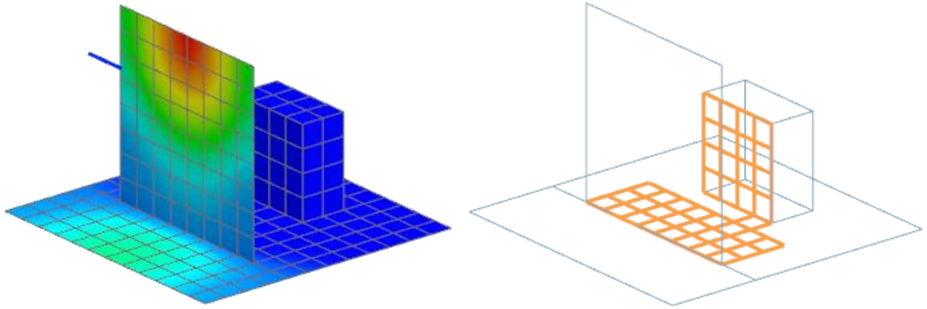
During a crash, a partial .bun file may be available. Check the simulation directory for the available .bun file. If not automatically connected to the solution, import the .bun file into post-processing.



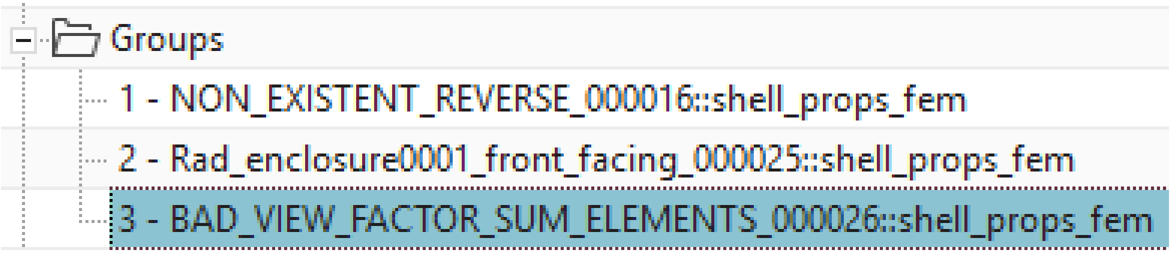
Resolving a warning

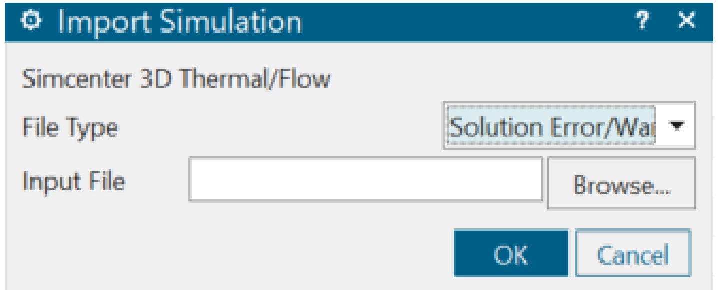
Review the messages in the **Solution Monitor** or the log file for the thermal and flow solvers:

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



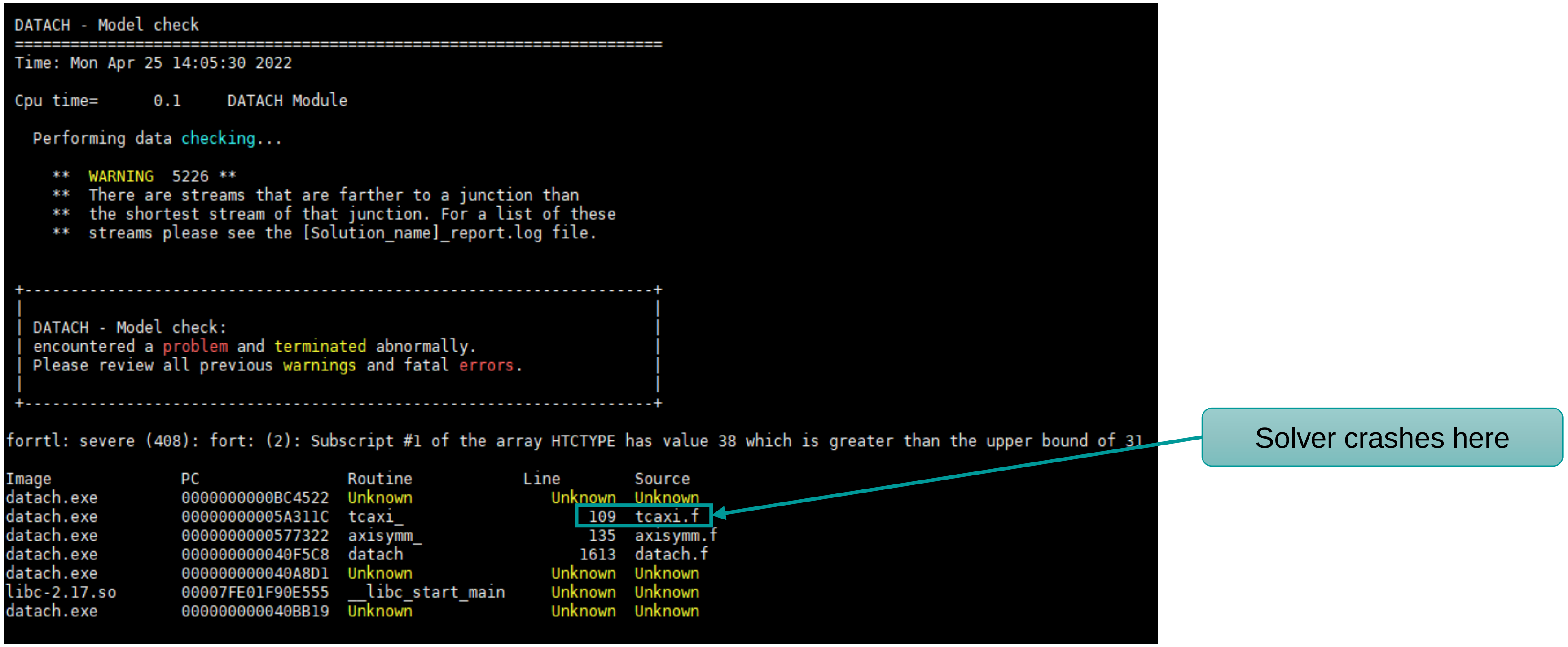
Import the solution warning groups and observe the failed elements. Choose **File** → **Import** → **Simulation**, select **Simcenter 3D Thermal/Flow**.





Applying a traceback patch

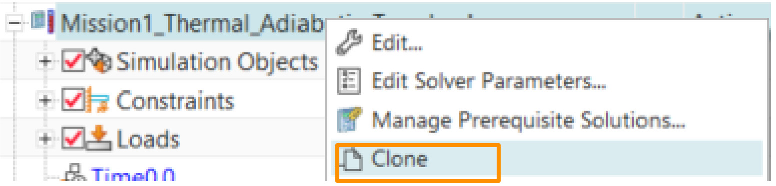
Applying a traceback patch requires referencing a new set of thermal solver files before solving. The log file provides detailed information after a fatal crash, including the code location and the line number where the crash occurred.

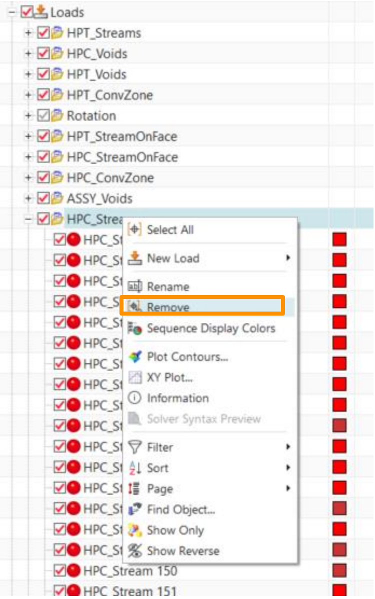


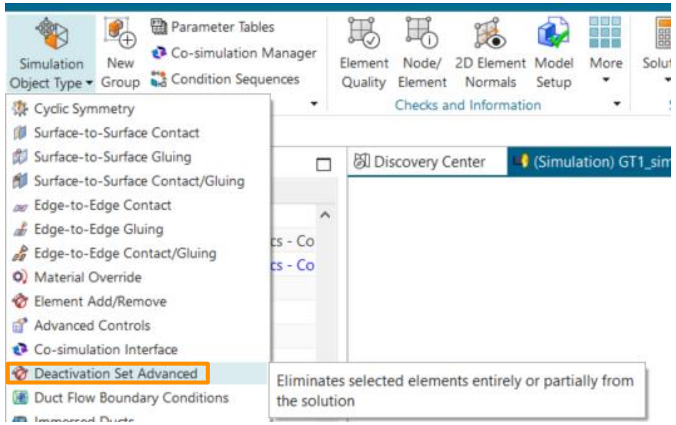
```
DATACH - Model check
======================================================================
Time: Mon Apr 25 14:05:30 2022

Cpu time=      0.1      DATACH Module

  Performing data checking...

    **  WARNING  5226 **
    **  There are streams that are farther to a junction than
    **  the shortest stream of that junction. For a list of these
    **  streams please see the [Solution_name]_report.log file.


+--------------------------------------------------------------------+
|                                                                    |
| DATACH - Model check:                                              |
| encountered a problem and terminated abnormally.                   |
| Please review all previous warnings and fatal errors.              |
|                                                                    |
+--------------------------------------------------------------------+

forrtl: severe (408): fort: (2): Subscript #1 of the array HTCTYPE has value 38 which is greater than the upper bound of 31

Image              PC                Routine            Line        Source
datach.exe         0000000000BC4522  Unknown               Unknown  Unknown
datach.exe         00000000005A311C  tcaxi_                    109  tcaxi.f
datach.exe         0000000000577322  axisymm_                  135  axisymm.f
datach.exe         000000000040F5C8  datach                   1613  datach.f
datach.exe         000000000040A8D1  Unknown               Unknown  Unknown
libc-2.17.so       00007FE01F90E555  __libc_start_main     Unknown  Unknown
datach.exe         000000000040BB19  Unknown               Unknown  Unknown
```

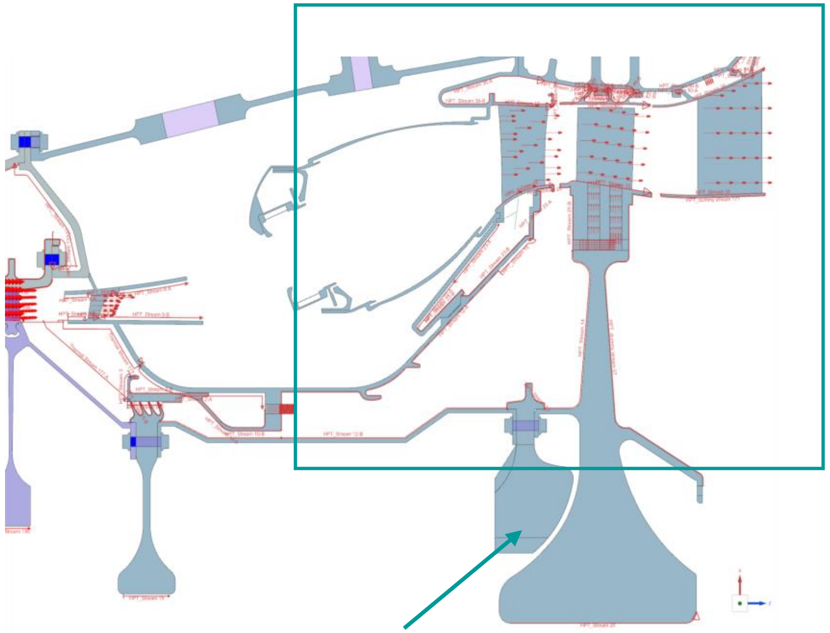
Simplifying the model

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.

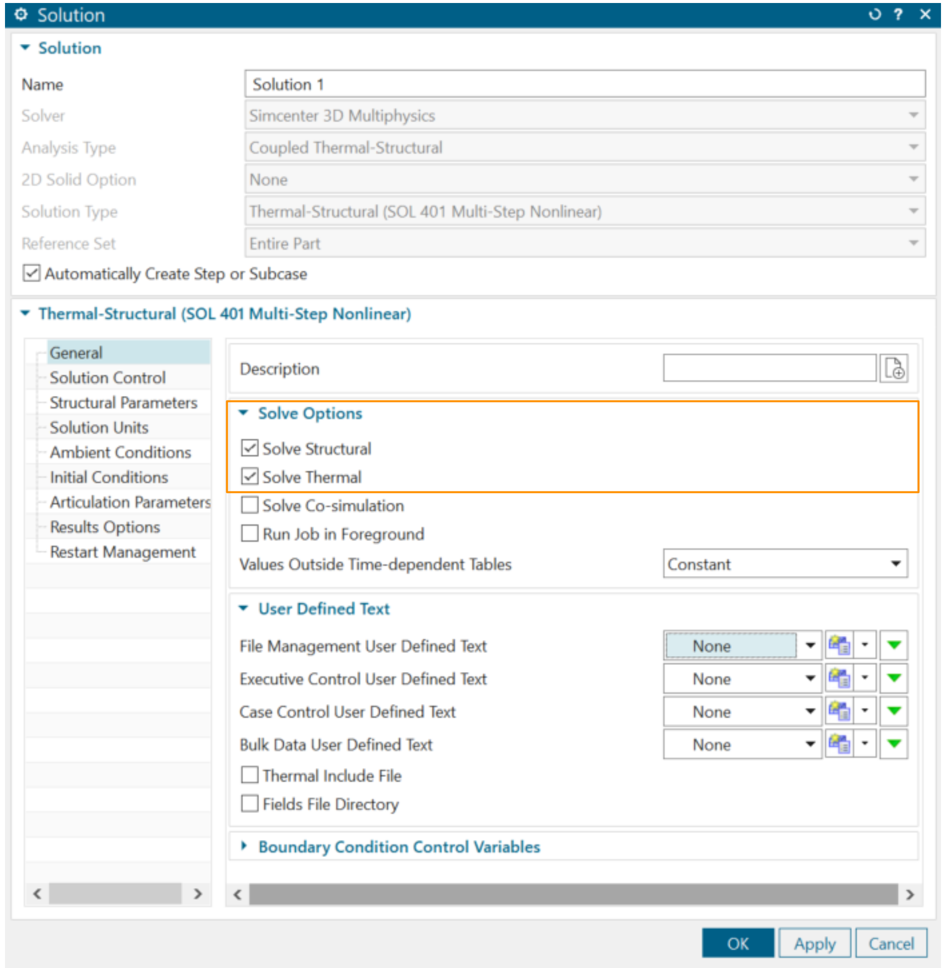
If the crash still persists, remove more boundary conditions until you find where the problem is.

SIEMENS

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.



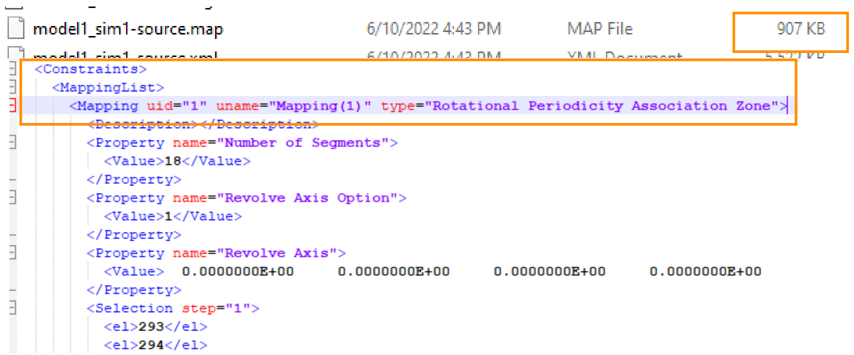
Thermal mapping troubleshooting

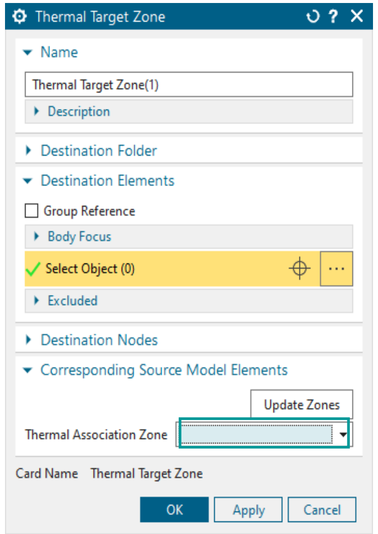
- It is recommended to use the **Simcenter3D Thermal/Flow** mapping solution.
- Common issues:
  - No mapping **Association Zones** appear when trying to set the **Target Zones**.
  - Mapping results show unexpected temperature gradients.
  - FATAL 15018 – Target Zone <xxx> intersects target zone <yyy>.

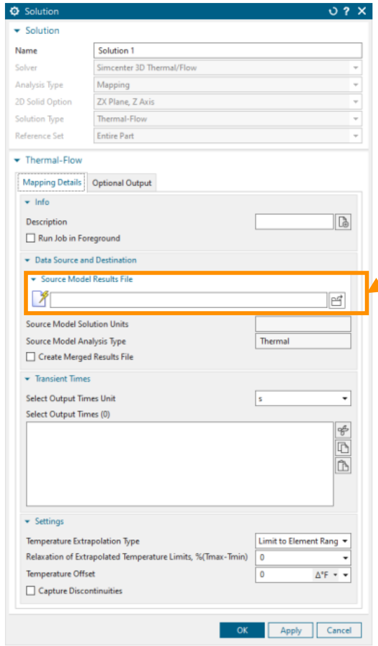
Thermal mapping troubleshooting

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**.







Confirm that the referenced .bun is the correct model.

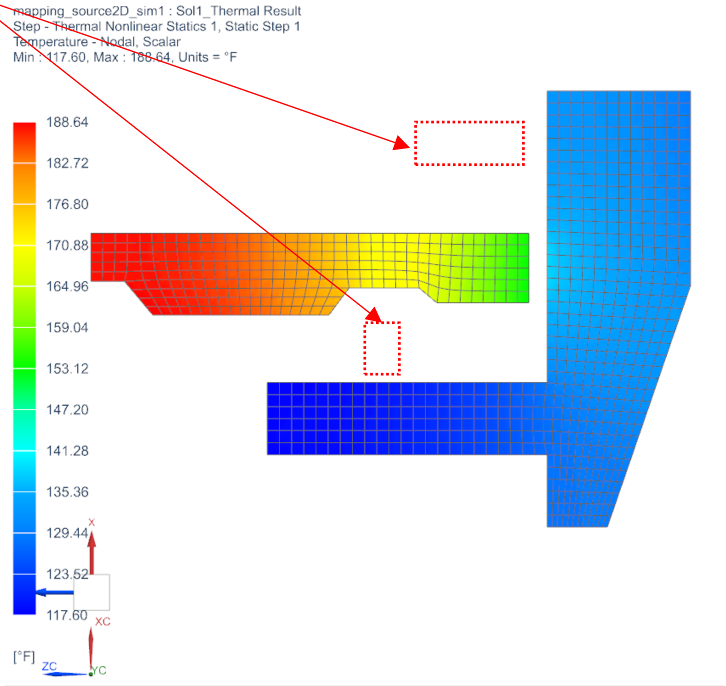
Note: If the only change to the source model is the addition of **Association Zones**, the source .xml and .map files can be regenerated without re-solving the solution.

SIEMENS

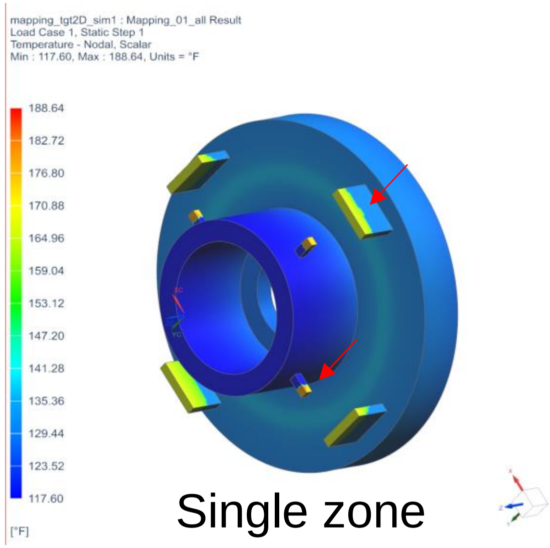
Unexpected temperature gradients

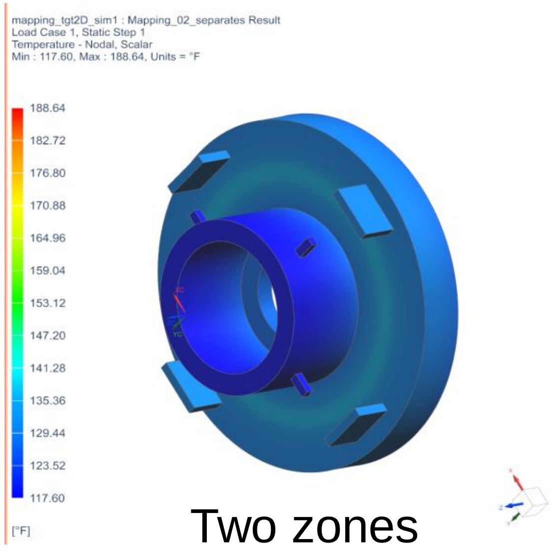
Use association and target zones to guide the mapping solver. If not specified, the solver maps using the nearest source temperature based on proximity. Ensure the source model's association zones cover the desired regions and verify alignment with the correct **2D Solid Option** defined, if applicable.

Location of tabs added to Target model



Target Model



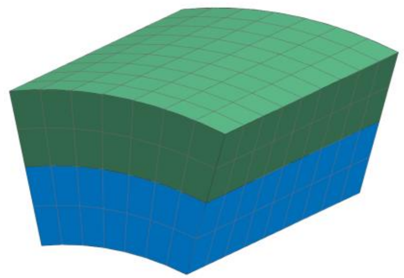


Resolving FATAL 15018 issue

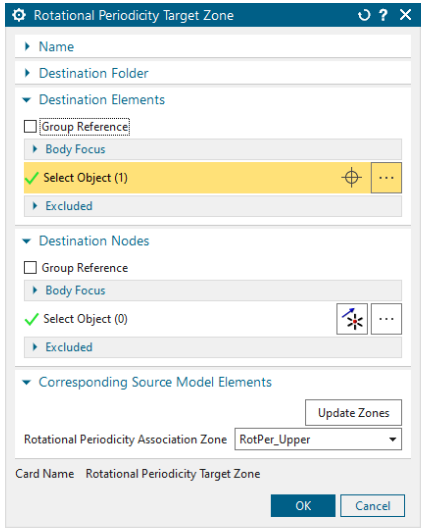
Mapping target zones cannot overlap.

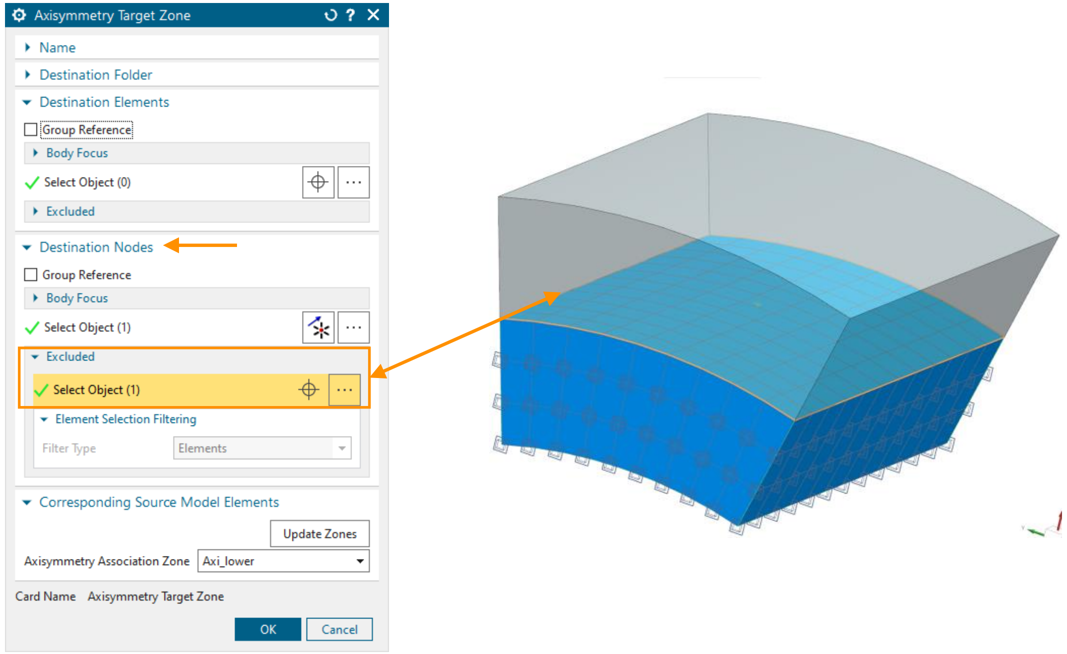
The solver will issue the FATAL 15018 error if they overlap.

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



```
Evaluating Target - Source Association...

  **  FATAL 15018 **
  **  Target zone ATZ_nd
  **  intersects target zone RTZ_nd
  **  One example of a common node is          1

  **  Run aborted due to errors.
```





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.