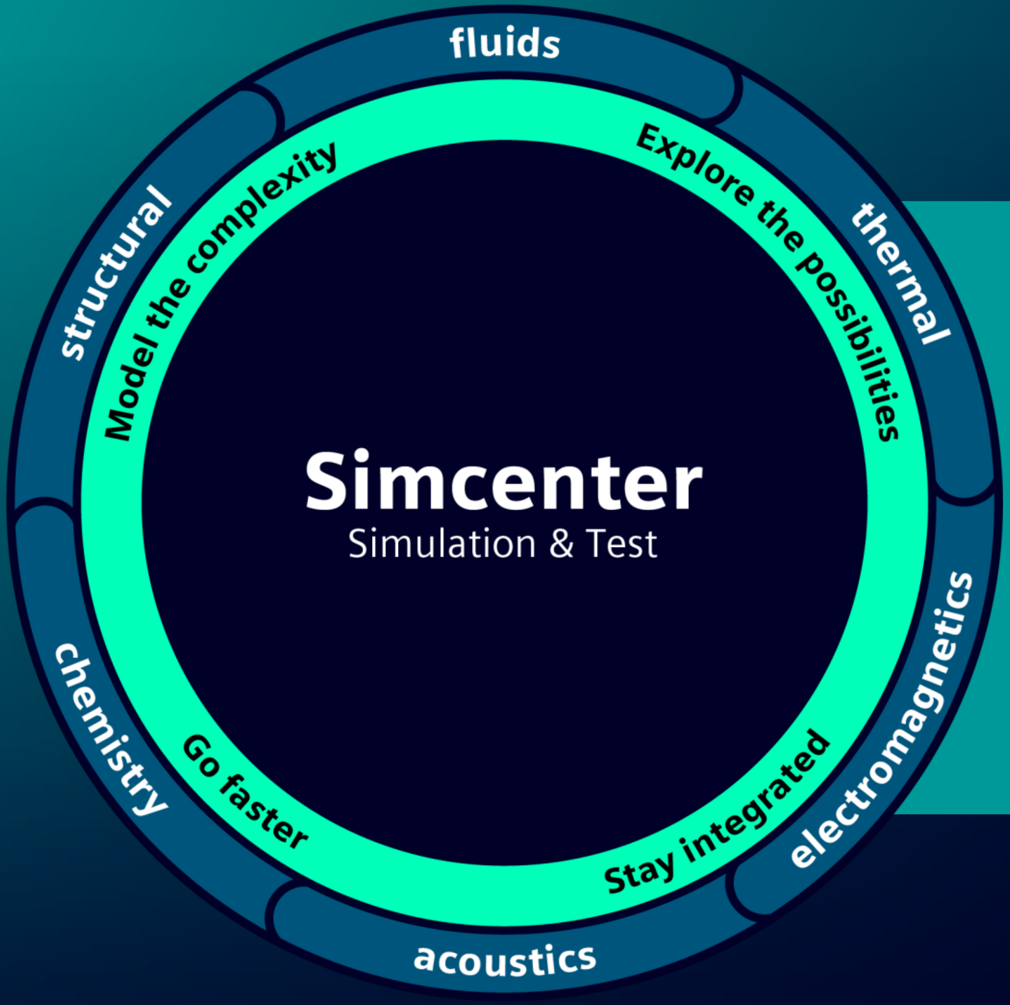



Thermal

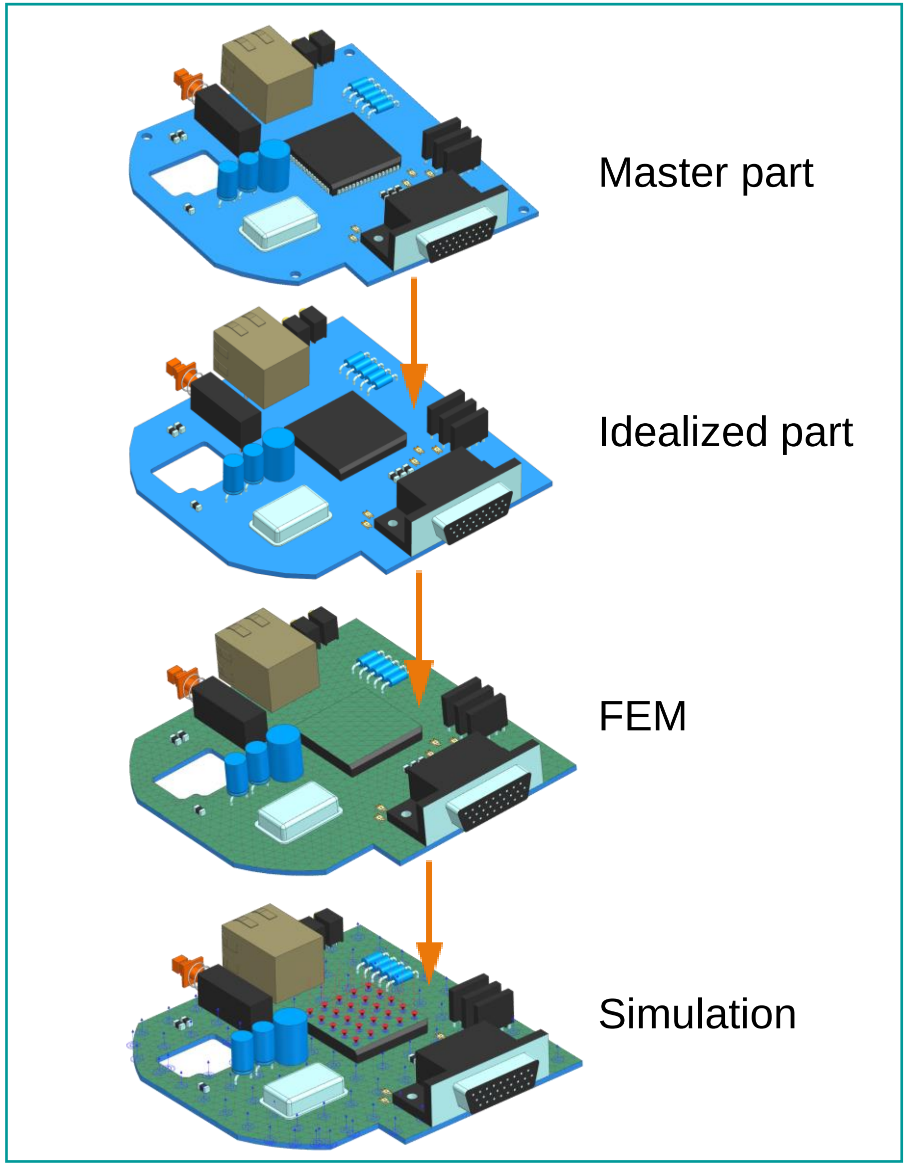
# Overview of Thermal Solver files and how to use them

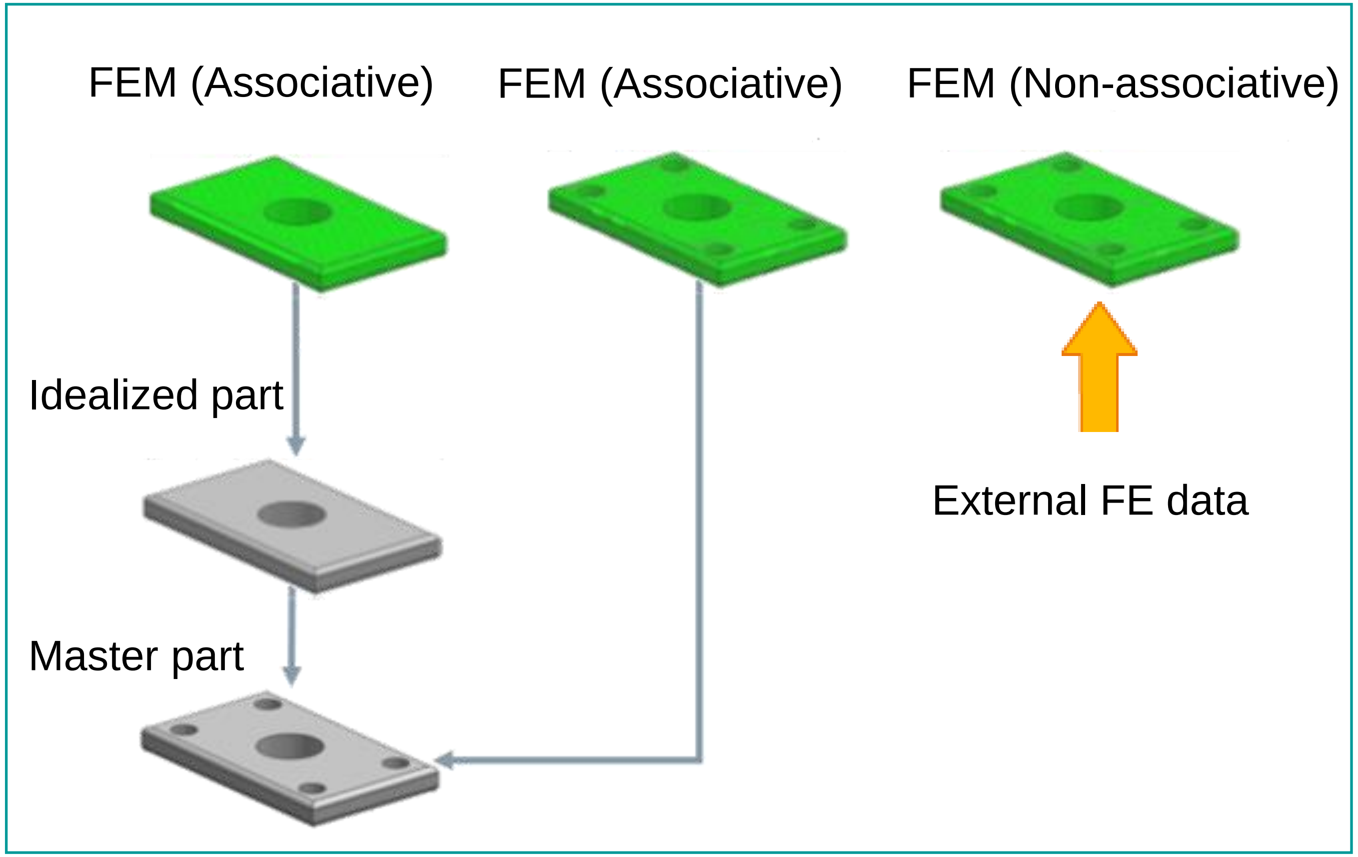
## Simcenter 3D Thermal Multiphysics




SIEMENS

# Simcenter 3D part files



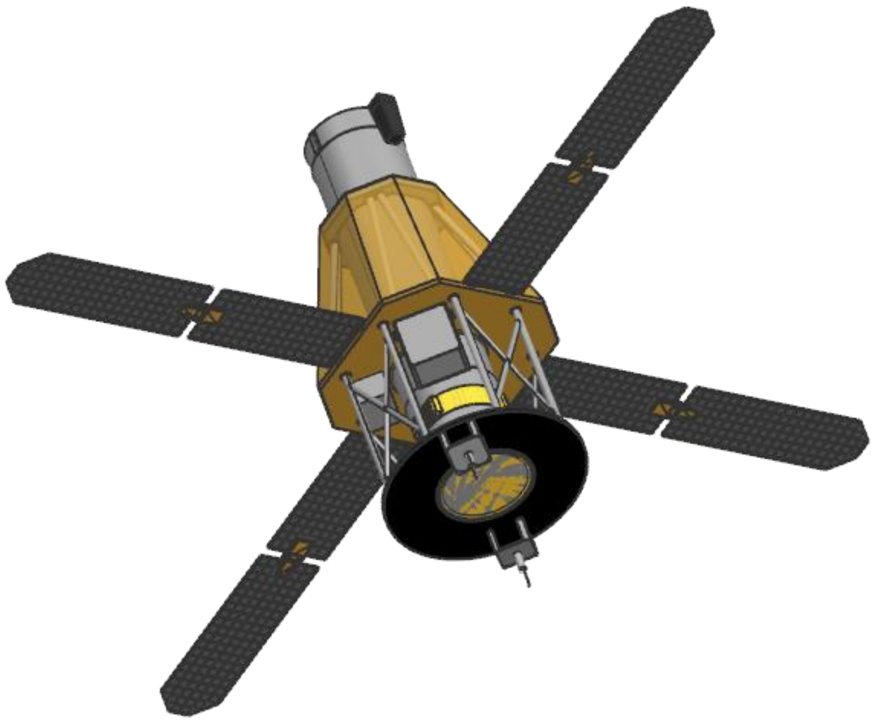

 

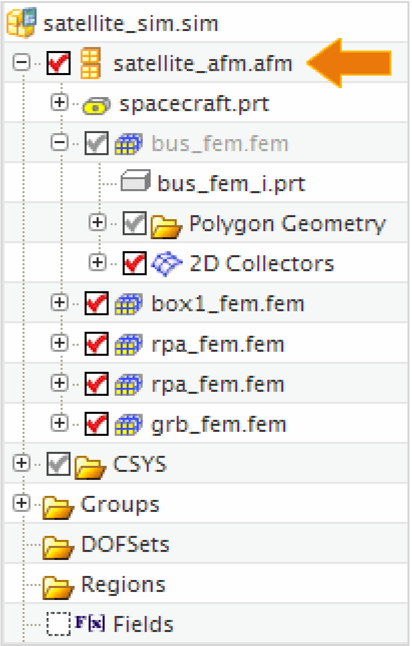
SIEMENS

# Assembly FEMs

The assembly FEM contains:

- Occurrences of the component FEMs.
- Properties overwritten on specific FEM occurrences.





SIEMENS

# Solution Monitor

The **Solver Monitor** provides the following information from the solver during the analysis:

- Solver version, run time, and run directory.
- Model summary.
- Status of the solution and current module being executed by the thermal solver.
- Convergence residuals at each iteration during the analysis.
- Warnings and errors.

# Solution Monitor

You can abort or stop a solution using the **Solution Monitor** window.

- **Abort** terminates the analysis. All temporary files are removed and the results are not post processed. You cannot restart or continue an aborted solution.
- **Stop** ends the analysis before the final time step or steady state. The software keeps the temporary files, performs post-processing, and creates results output files.

The solution monitor window is written to a *<simulation/model name>-<solution/analysis name>.log* file, located in the same directory as the SIM file.

# Solver Reference Manual

**All the following information and much more can be found in the thermal solver reference manual:**

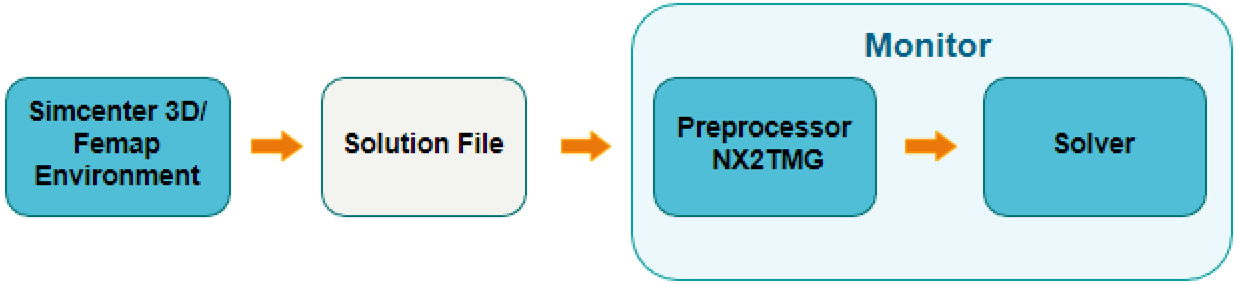
**2306:**

https://docs.sw.siemens.com/en-US/doc/289054037/PL20221116635232682.advanced/id629461?audience=external

# Solution file: XML

*<simulation/model name>-<solution/analysis name>.xml*

The file contains the model and solution definition. The model definition includes meshes, elements, and model properties. The solution definition includes the boundary conditions and solution settings.




The CAE software launches the Monitor, where the file data is processed and prepared in a format (INPF file) that the thermal solver require to perform the solve.

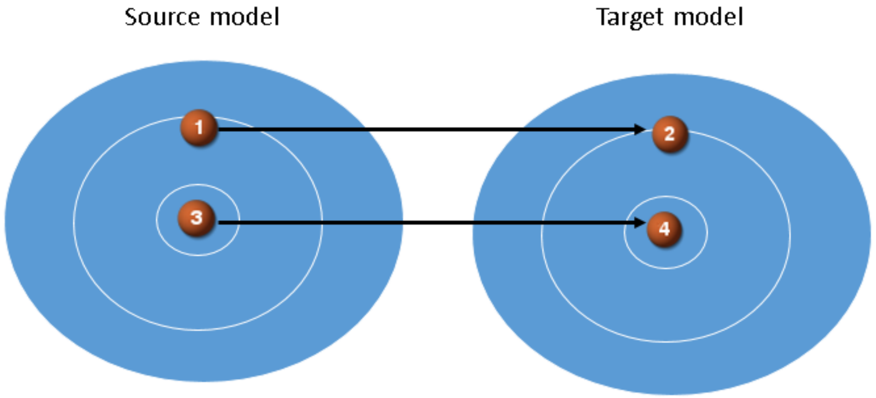


SIEMENS

# Mapping file: MAP

*<simulation name>-<solution name>.map*

A mapping file is used by the solution file to retrieve the regions where mapping is defined in the source and target models. It also includes information about the mapping zones defined for the source and target model, such as thermal, transverse gradient, and axisymmetry mapping zones.



# Solution Monitor/Log file: Module description

```
Thermal/Flow Analysis
=====================
MAYA Heat Transfer Technologies Ltd.

TMG Thermal/Flow 10.0.308  Oct 28 2016

Run directory specified for analysis:
.\Solution_thermal\

NX2TMG - Thermal/flow model file builder
=====================================================================
Time: Thu Nov 3 16:55:09 2016

+-------------------------------------------------------------------+
|                           Model Builder                           |
+-------------------------------------------------------------------+

Reading model file...
...model file read.
```

Model Summary

Run Directory

Monitor information is saved to:

*<simulation/model name>-<solution/analysis name>.log*

```
+-------------------------------------------------------------------+
|                           Model Summary                           |
+-------------------------------------------------------------------+

Finite Element Model
--------------------

Solver type:     NX ELECTRONIC SYSTEMS COOLING
Analysis type:   Coupled Thermal-Flow
Solution type:   Electronic Systems Cooling
Simulation File: D:\6_Projects\16_Siemens_Hydra\NX Assets\Problem_1\Problem
Solution:

Number of nodes in thermal model:                       5242
Number of elements in thermal model:                   24061

Number of nodes in flow model:                             0
Number of elements in flow model:                          0

Number of elements created for surface coating:        3675

Solution Options
----------------

Transient Analysis
Not Solving Flow
Solving Thermal

Ambient Temperature:                          3.700E+01 C
Ambient Pressure:                             1.014E+02 mN/mm^2
Gravitational Acceleration:                   9.810E+03 mm/s^2
Gravity Direction: X: 0.000E+00  Y: 0.000E+00  Z:-1.000E+00

Transient Options
-----------------

Thermal solver timestep:                      5.000E+00 s
Thermal results printout interval:            1.000E+01 s
Initial simulation time:                      0.000E+00 s
Final simulation time:                        1.000E+03 s

Performing mesh checks...

 - interior and warping angle
   checks on fluid elements...
   ...angle checks on fluid
      elements completed.

 - interior and warping angle
   checks on thermal elements...
   ...angle checks on thermal
      elements completed.
```

Model Summary

```
 - coincident node check
   on fluid model...
   ...node check on fluid
      model completed.

 - coincident node check
   on thermal model...
   ...node check on thermal
      model completed.

 - coincident element check
   on fluid model...

  ... coincident fluid elements
      check completed.

...mesh checks completed.

Thermal Boundary Conditions
---------------------------

Number of thermal boundary  conditions:                    2

Thermal Boundary Condition     Type         Value              FSF  TCP
-----------------------------------------------------------------------
  1 - Initial Conditions(1)    Initial Temp 1.800E+01 C         No   No
  1 - Convection to Environme  Convective   3.700E+01 C         No   No

Surface Property Definitions
-----------------------------

  1 - Default surface property
      Roughness:   0.000E+00 mm
      Heat Transfer Coefficient: Auto. Calc. Forced Convection

Writing thermal model files...
...done.

Executing Thermal Solver Only
```

# Solution Monitor/Log file: Module description

```
THERMAL - Thermal analysis
======================================================================
Time: Thu Nov 3 16:55:10 2016

+--------------------------------------------------------------------+
| Solving Thermal Model - Elapsed Time: 00 min 00 sec                |
+--------------------------------------------------------------------+
Cpu time=      0.0        MAIN Module
Current FE Model:   D:\6_Projects\16_Siemens_Hydra\NX Assets
\Problem_1\Problem_1\CUVETTE R
Current FE Study:

  Starting TMG Analysis...

  ...done.
Cpu time=      0.1      DATACH Module

  Performing data checking...

    Number of nodes                          5242
    Number of geometric elements            27735
    Number of 1-D/duct fluid elements           0

  ...done.
Cpu time=      1.9       ECHOS Module

  Calculating geometrical parameters...

  ...done.
Cpu time=      2.5        COND Module

  Calculating conductive conductances and capacitances...

  ...done.
Cpu time=      4.8       VUFAC Module

  Calculating thermal couplings and geometric radiative parameters...

  ...done.
Cpu time=      7.8       GRAYB Module

  Calculating radiative couplings and/or gray body matrices...

  ...done.
Cpu time=      7.8       POWER Module

  Calculating radiative heat loads...
    No radiative heat loads calculated.

  ...done.
Cpu time=      7.8       MEREL Module

  Performing element merging and elimination...

  ...done.
```

**MAIN Module:** Performs data checking, determines which modules to run.

**DATACH Module:** Performs data checking, and orbit creation.

**ECHOS Module:** ECHOS calculates each element's CG, element center, area or volume, hydraulic diameter, and surface normal along with the location of the nodes.

**COND Module:** Calculates capacitances, hydraulic resistances, and conductive conductances from geometry.

**VUFAC Module:** Calculates view factors, solar view factors, albedo factors, Earth view factors,heat flux view factors, and thermal couplings from geometry.

**GRAYB Module:** Calculates radiative conductances and gray body view factor matrices from view factors.

**POWER Module:** Calculates IR and solar spectrum heat loads from view factors and gray body view factor matrices.

**MEREL Module:** Performs model simplification, merging, substructuring, and combines parameters.

# Solution Monitor/Log file: Module description

```
THERMAL - Thermal analysis
========================================================================
Time: Mon Nov 7 15:13:19 2016
Cpu time=     16.4    ANALYZER Module

  Calculating temperatures...

    Model Summary:

    Number of elements                 =     74021
    Total number of conductances       =    148040
    Number of linear conductances      =    144366
    Number of convective conductances  =      3674
    Number of boundary elements        =     46285

    Note: During the analysis process a number of elements have been
    created whose labels you may not recognize. To find out more about
    these elements and the elements they are associated with, please
    look on file REPF for the phrase 'TMG element'


    **  WARNING **
    **         10 conductances out of a total of    148040 were modified to
    **  make the solution more robust. A list of modified elements was
    **  written on file REPF and file groups.unv with group name
    **  MODIFIED_CONDUCTANCES_000009


    Time= 0.000000000000E+00 Integration timestep= 5.000000E+00

    Minimum temperature                =   18.000 at element     77249 C2E_10
    Maximum temperature                =   37.000 at element     80896 FLUAMB
    Average temperature                =   18.001

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

Fluid ambient group                     3.700E+01  -4.966E+05   0.000E+00

    Time= 1.000000000000E+01 Integration timestep= 5.000000E+00

    Minimum temperature                =   19.197 at element     56897 IC_18
    Maximum temperature                =   37.000 at element     80896 FLUAMB
    Average temperature                =   20.135

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

Fluid ambient group                     3.700E+01  -4.344E+05  -4.481E+06

    Time= 2.000000000000E+01 Integration timestep= 5.000000E+00

    Minimum temperature                =   20.558 at element     57438 IC_18
    Maximum temperature                =   37.000 at element     80896 FLUAMB
    Average temperature                =   22.021

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

Fluid ambient group                     3.700E+01  -3.851E+05  -8.451E+06

    Time= 3.000000000000E+01 Integration timestep= 5.000000E+00

    Minimum temperature                =   21.849 at element     57438 IC_18
    Maximum temperature                =   37.000 at element     80896 FLUAMB
    Average temperature                =   23.692
```

**ANALYZER Module:** Calculates temperatures and total pressures.

```
Fluid ambient group                     3.700E+01  -1.006E+02  -4.006E+07

    Time= 7.500000000000E+02 Integration timestep= 5.000000E+00

    Minimum temperature                =   36.987 at element     56671 IC_18
    Maximum temperature                =   37.000 at element     80896 FLUAMB
    Average temperature                =   36.999
Heat Flow+Load Summary Into Different Sink Entities:
       Sink Entity                    Temperature      Heat   Energy absorbed
                                                   Flow+Load   since start

Fluid ambient group                     3.700E+01  -9.009E+01  -4.006E+07
  ...done.
Cpu time=     74.2    RSLTPOST Module

  Performing result postprocessing...


  ...done.
Cpu time=    160.2       End of run

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

Solution elapsed time: 02 min 57 sec
```

Heat Flow Summary

**RSLTPOST Module:** Transforms results into I-deas Universal file format.

SIEMENS

## Module description

| INPUT | EXECUTABLE | OUTPUT |
|---|---|---|
| INPF | MAIN<br>Data checking, determines modules to be run. | <simulation/model name>-<br><solution/analysis name>_verbose.log |
| INPF | DATACH<br>Data checking, orbit creation. | <simulation/model name>-<br><solution/analysis name>_verbose.log<br><simulation/model name>-<br><solution/analysis name>_report.log<br>tmggeom.dat |
| INPF<br>tmggeom.dat | ECHOS<br>Geometry parameter calculations. | VUFF<br>tmggeom.dat |
| INPF<br>tmggeom.dat | COND<br>Calculates capacitances, hydraulic resistances, and conductive conductances from geometry (optional). | MODLF<br>tmggeom.dat |
| INPF<br>tmggeom.dat | VUFAC<br>Calculates view factors, solar view factors, albedo factors, Earth view factors, heat flux view factors, thermal couplings from geometry (optional). | MODLF<br>VUFF<br><simulation/model name>-<br><solution/analysis name>_verbose.log<br><simulation/model name>-<br><solution/analysis name>_report.log |
| INPF<br>VUFF<br>tmggeom.dat | GRAYB<br>Calculates radiative conductances, gray body view factor matrices from view factors (optional). | VUFF<br>MODLF<br>tmggeom.dat |

# Module description

| INPUT | EXECUTABLE | OUTPUT |
|---|---|---|
| VUFF<br>INPF<br>tmggeom.dat | POWER<br>Calculates IR and solar spectrum heat loads from view factors and gray body view factor matrices (optional) | VUFF<br>MODLF |
| INPF<br>MODLF<br>tmggeom.dat | MEREL<br>Model simplification, merging, substructuring, combines parameters calculated from geometry and defined on Card 9. | <simulation/model name>-<solution/analysis name>_report.log<br>MODLCF<br>tmggeom.dat |
| USER1<br>MODLCF<br>tmggeom.dat | ANALYZER<br>Calculates temperatures and total pressures. | <simulation/model name>-<solution/analysis name>_report.log<br>TEMPF<br>PRESSF<br>tmggeom.dat |
| tmgrslt.dat<br>tmggeom.dat | RSLTPOST<br>Transforms results into I-deas Universal file format. | UNIVERSAL FILES |

SIEMENS

# <simulation/model name>-<solution/analysis name>.log

At the end of a steady state run, the log file includes a summary of the heat flow:

- Heat load is a directly applied heat load Q (through BCs: volumetric heat flows or surface heat fluxes).

- Heat flows are heat flows through conductances (because of temperature differences).

```
Maximum heat balance deviation occurs          at element     99088 INIT_BC
Heat flow into sinks              = 2.199E+03
Heat flow from non-fluid sinks    = 0.000E+00
Heat load into elements           = 2.199E+03
Heat load into sinks              = 0.000E+00
Heat flow from fluid sinks        = 0.000E+00
Deviation from heat balance       =-1.012E-01
```

- For a coupled thermal-flow(CFD) solution:

***Heat Imbalance = Total Heat Load on Elements + Heat Flow from Sinks – Heat Flow into Sinks***

- Using the terminology in the log file we have:

***Total Heat Imbalance = Total heat load on non-fluid elements + (Heat flow from temperature B.C.s + Heat convected from fluid) – (Heat flow into temperature B.C.s + Heat convected to fluid)***

- The steady state Heat Imbalance solution is when the Total Heat Imbalance goes to 0.
- The units on the log file are always the one defined in the **Solution Units** page. (Edit solution- Solution Units).

# <simulation/model name>-<solution/analysis name>.log

The log file also includes a summary of heat flow into various sinks (temperature boundary conditions):

```
Heat Flow+Load Summary Into Different Sink Entities:

       Sink Entity                          Temperature     Heat    Energy absorbed
                                                         Flow+Load    since start


Space Enclosure                             -1.531E+02   2.124E+03   0.000E+00
120K                                        -1.531E+02   7.558E+01   0.000E+00
```

The information about the heat needed to keep the sink entities at these temperatures can be very useful when selecting the size of the heater.

# INPF

INPF is the primary input file to the thermal solver. ASCII file that contains the nodes, elements, materials and boundary condition information for the thermal solver.

You can solve directly from INPF, similarly to XML. Sometimes can be useful to modify manually or with a script to run batch runs.

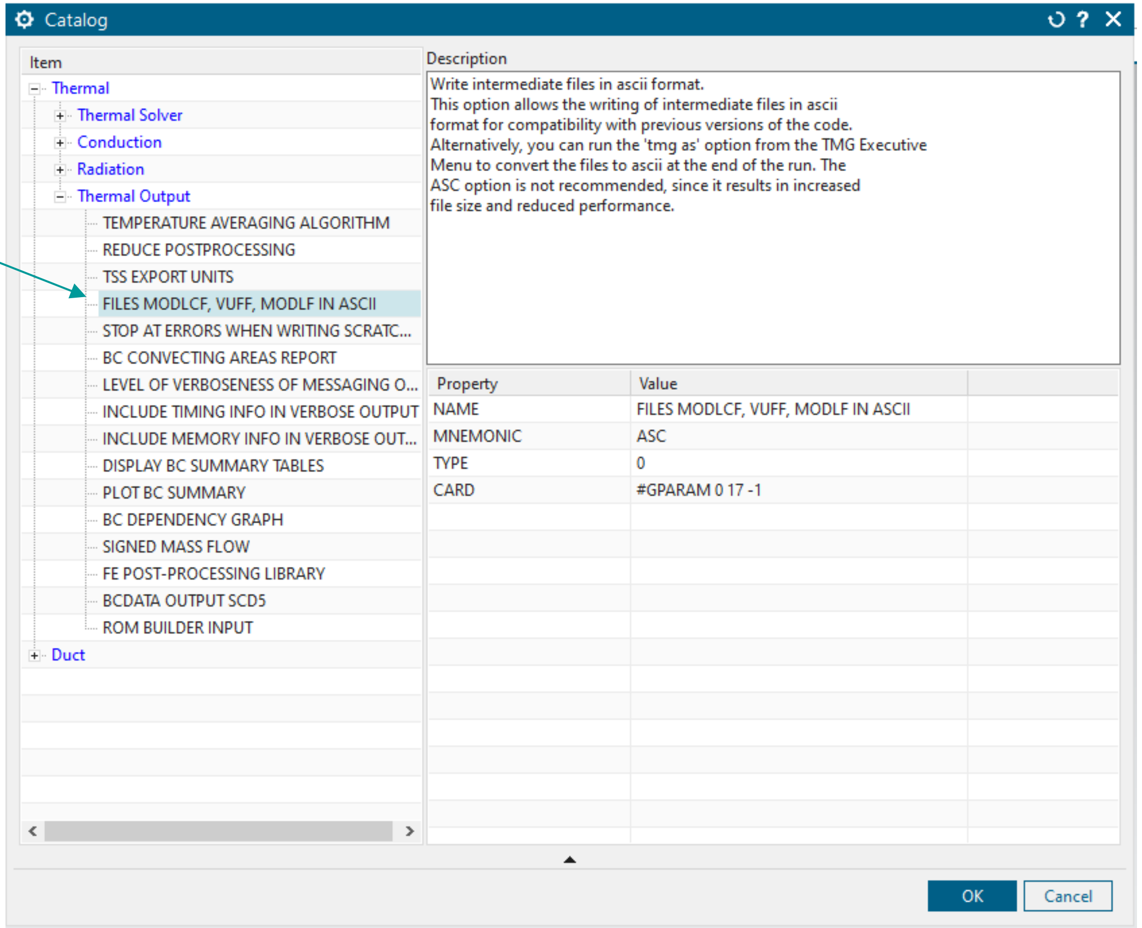
Solver include files are in the INPF format. Can be used to write additional loads on groups or include reduced models into a larger model.

# Binary files

The VUFF, MODLF, MODLCF, TEMPF are by default binary files and thus cannot be read in a text editor. However, it is possible to convert them to ASCII or human readable text files.

VUFF, MODLF and MODLCF can be converted to ASCII using the shown **Advanced Parameter**.




The TEMPF file can be converted after the solve using the TMG executive menu and the command AS.
The solver does not convert the TEMPF before the solve because of potential lose of performance and accuracy.

SC 2306:

https://docs.sw.siemens.com/en-US/doc/289054037/PL20221116635232682.advanced/id1008231

https://docs.sw.siemens.com/en-US/doc/289054037/PL20221116635232682.advanced/id1302047?audience=external

SIEMENS

# VUFF

The VUFF file contains:

- The model geometry elemental properties written by ECHOS.
- The view factors, heat flux view factors, solar view factors, albedo factors, and Earth view factors written by VUFAC.
- The IR and solar spectrum gray body view factor matrices written by GRAYB.

This is a good file to look for view factor or radiation conductances between elements. It can be more complicated to understand when using the Oppenheim method.

SIEMENS

# MODLF

MODLF contains all the thermal couplings, radiative, conductive, hydraulic, and convective conductances, element capacitances, and heat loads calculated from elemental geometry and material and physical properties by the COND, GRAYB, VUFAC, and POWER modules.

MODLCF is a condensed version of the MODLF file after addition of the Card 9 model parameters, element merging, substructuring, and combining of heat loads, capacitances, and conductances.

MODLF and MODLCF are by default binary files, however, they can be translated into ASCII format using the AS option in the TMG Executive Menu or with an **Advanced Controls** simulation object in Simcenter 3D. Both binary and ASCII formats are equally acceptable to TMG; however, the binary is preferred, because there is no loss in precision.

For debugging or manual inspection, MODLCF should be used over MODLF.

# REPF/<simulation/model name>-<solution/analysis name>_report.log

A log file that contains calculation details. This file is always appended to the solution. This file replaces the old REPF file.

The file contains information on:

- Groups
- Thermostats
- Multilayer shells
- Oppenheim elements
- Solver created elements

Very useful information for understanding the model, especially when looking at other solver files.

SIEMENS

# MSGF/<simulation/model name>-<solution/analysis name>_verbose.log

A log file that contains messages regarding thermal solver routines, including their timing and memory statistics and verbose messages. By default, the basic level of verboseness is activated. This file replaces the old MSFG file.

The thermal solver has the capability of displaying different levels of messaging in the log files:

- Level 1 only displays fatal errors.
- Level 2 displays fatal errors and warning messages.
- Level 3 displays fatal errors, warnings, and information messages. (This is the default.)

When you include the LEVEL OF VERBOSENESS OF MESSAGING OUTPUT advanced parameter into your solution, you can display:

- Level 4 displays fatal errors, warnings, information messages, and key information from different thermal solver routines including convergence traces. The level 4 information is written to the verbose log file. This is the default.
- Level 5 displays level 4 information and MPI secondary ranks. It also resolves messages by boundary conditions. All extra information from level 5 is written only to the verbose log file.

For levels 4 and 5, you can also request the timing and memory information in thermal solver modules that support it when you include the INCLUDE TIMING INFO IN VERBOSE OUTPUT and INCLUDE MEMORY INFO IN VERBOSE OUTPUT advanced parameters, respectively.

# TEMPF

The TEMPF file contains the calculated temperatures at the end of a run. The Analyzer writes data on TEMPF at the printout intervals

Format:

I, TEMP(I)

- I is the element number.
- TEMP(I) is the temperature of element I.
- If I is -99999, TEMP(I) is the printout time value for subsequent temperatures.

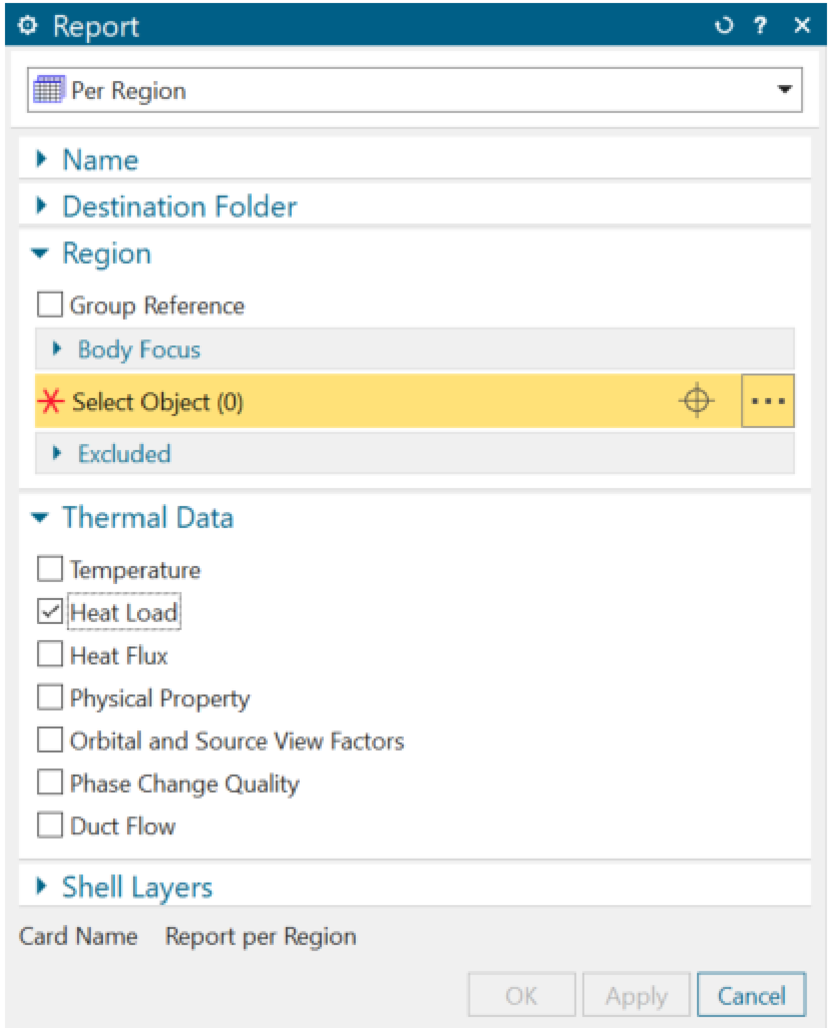
The file can be used for initial conditions or mapping.

# Post Processing: Reports

Reports are used to extract specific information from the model.

The types of report available is dependent on the solver, the analysis type, and the solution selected.



# Post Processing: Reports

| Report type | Description | Report format | File names |
|---|---|---|---|
| **Per Element** | Generates selected thermal solution or duct data for selected elements. | Text | *[simulation name]-[solution name]_report.log* |
| **Per Region** | Generates selected thermal solution data or 3D flow data for a specified region of your model. | HTML and text (.csv) | *[simulation name]-[solution name].GroupReport.htm*<br>*GroupReport.csv* |
| **Between Regions** | Generates selected thermal and flow solution data between specified regions of your model. | HTML and text (.csv) | *[simulation name]-[solution name].GroupReport.htm*<br>*GroupReport.csv* |
| **Track During Solve** | Generates a summary of thermal and flow conditions for the selected region. | Text (.csv) and graphs (.png) | *TrackReportThermal.csv*<br>*TrackReportFlow.csv*<br>*[report name] Fluid Pressure.png*<br>*[report name] Fluid Temperature.png*<br>*[report name] Fluid Velocity.png*<br>*[report name] Solid Temperature.png* |
| **Lift and Drag** | Reports lift and drag forces on flow surfaces. | HTML and text (.csv) | *[simulation name]-[solution name].GroupReport.htm*<br>*GroupReport.csv* |
| **Heat Maps** | Generates heat flow data between each pair of regions. | HTML and text (.csv) | *[simulation name]-[solution name].GroupReport.htm*<br>*GroupReport.csv* |

# BC summary table

When you include the DISPLAY BC SUMMARY TABLES advanced parameter in a solution, the thermal solver generates the <simulation name>-<solution name>.bcdata file that outputs in the table format the time and evaluated boundary condition data for the following loads and simulation objects:

- Thermal Stream loads
- Thermal Convecting Zone loads
- Thermal Void loads
- Immersed Duct simulation objects
- Heat Pipe type of Thermal Device simulation objects

The table also outputs the data for named points and groups of elements. (2306+)
This file is updated at each successful time step during the solve and stored in the run or scratch directory.
This table facilitates evaluation of the data related to the boundary condition.

VOID BC DATA

| ID | Reg.# | TIME | Tmax | Pmax | THR | Trel-max | CHR | AREAC | SVmax |
|---|---|---|---|---|---|---|---|---|---|
| 1. | 1. | 1.000 | 632.2 | 14.50 | -9764. | 632.2 | -9764. | 414.8 | 0.0000E+00 |

ZONE BC DATA

| ID | TIME | Tmax | Pmax | THR | Trel-max | CHR | AREAC | SVmax |
|---|---|---|---|---|---|---|---|---|
| 1. | 1.000 | 422.7 | 78.82 | 8280. | 422.7 | 8280. | 221.7 | 0.0000E+00 |

# PLOT BC Data

When you include the PLOT BC SUMMARY advanced parameter in the solution, the thermal solver generates the <simulation name>-<solution name>data.html file that displays graphs of thermal and fluid properties of the following loads and simulation objects included in the solution:

- Thermal Stream loads
- Thermal Convecting Zone loads
- Thermal Void loads
- Thermal Coupling simulation objects
- Immersed Duct simulation objects
- Heat Pipe type of Thermal Device simulation objects

You can also display graphs of thermal and fluid properties for named points and groups of elements.

This file is update during the run at each iteration.

SIEMENS

# BUN

The ***<simulation/model name>-<solution/analysis name>.bun*** is the results file in binary universal file format. This file can be read by NX/Simcenter 3D for post-processing.