

December 2017

ESATAN-TMS Workbench

Getting Started Guide

Prepared by:

ITP Engines UK Ltd.
Cambridge Road
Whetstone, Leicester, UK

All rights reserved, Copyright © 2017 ITP Engines UK Ltd.

This document may not be distributed, corrected, modified, translated or transmitted in whole or part without prior written authorisation of the above company.

ESATAN is a trademark of ITP Engines UK Ltd.

Contents

1: ESATAN-TMS Workbench Basics

1.1	Introduction	1-1
1.2	How do I use ESATAN-TMS Workbench?	1-1
1.3	ESATAN-TMS Workbench Modules	1-11
1.4	Conventions	1-12
1.5	Acronyms	1-13

2: Model 1 - Simple (Shell) Satellite Model

2.1	Introduction	2-1
2.2	Defining the Geometry	2-3
2.3	Running the Radiative Case	2-23
2.4	Preparing for Thermal Analysis Case Control	2-26
2.5	Solving with ESATAN-TMS Thermal	2-28
2.6	Post-processing the Thermal Analysis Results	2-32

3: Model 2 - Complex (Solid) Satellite Model

3.1	Introduction	3-1
3.2	The Orbit	3-3
3.3	Defining the Geometry	3-4
3.4	Running radiative cases	3-38
3.5	Solving with ESATAN-TMS Thermal	3-47
3.6	Post-Processing Thermal Analysis Results	3-55

1. ESATAN-TMS Workbench Basics

1.1 Introduction

ESATAN Thermal Modelling Suite (ESATAN-TMS) is a complete environment for thermal analysis. The Graphical User Interface (ESATAN-TMS Workbench) provides an intuitive and simple to use environment, providing extensive 3D model building and pre- and post-processing capabilities.

Workbench has three main modules which allow you to complete your analyses. These modules are for:

- Building the geometric model of your object
- Calculating the model's radiative characteristics
- Defining and creating the thermal model

Two further modules enable you to produce reports and run various utilities. The options for these modules can be selected from the menubar (see Subsection 1.2.1).

1.2 How do I use ESATAN-TMS Workbench?

The built-in graphical user interface makes ESATAN-TMS simple to use. Interaction is via menus, dialogs and buttons. You can easily complete the entire analysis process without requiring detailed knowledge of the language ESATAN-TMS is based on.

The main steps in the modelling process are as follows:

- Generate the geometry model
- Verify the model using visualisation and reporting
- Define and run the Radiative Case (orbit & radiative analysis)
- Validate the results using visualisation and reporting
- Define and run the Analysis Case (thermal model and thermal solution)
- Validate the thermal results using visualisation

1.2.1 ESATAN-TMS Workbench

When you first start ESATAN-TMS Workbench, the main Workbench window will appear on the screen - see Figure 1-1 below:

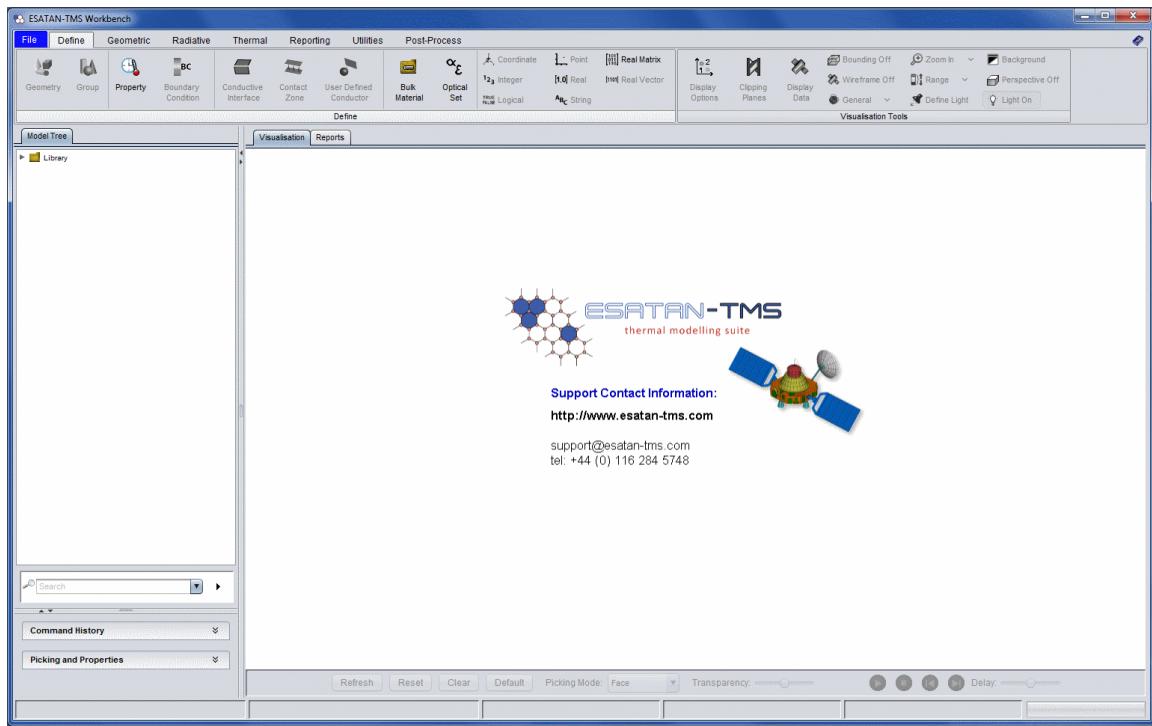


Figure 1-1 The ESATAN-TMS Workbench window

There are nine main areas to the Workbench window:

- Menubar
- Model tree
- Ribbon bar
- Search bar
- Toolbar
- Picking and Properties
- Video controls
- Visualisation controls
- Visualisation area
- Status bar

1.2.1.1 The Menubar

The menubar is used to access Workbench's various functions and operations via the ribbon bar.

- The Help menu under the File drop-down menu allows you to access Workbench manuals and guides. There are also Help buttons on all the main dialogs in Workbench and the manuals are also accessible from the Start menu (Windows only), outside of Workbench.



Figure 1-2 Menu bar

1.2.1.1.1 File drop-down menu

Selecting File from the menu bar will reveal the file drop-down menu in Figure 1-3 below where you can select options to enable you to manage files, access help manuals and set preferences.

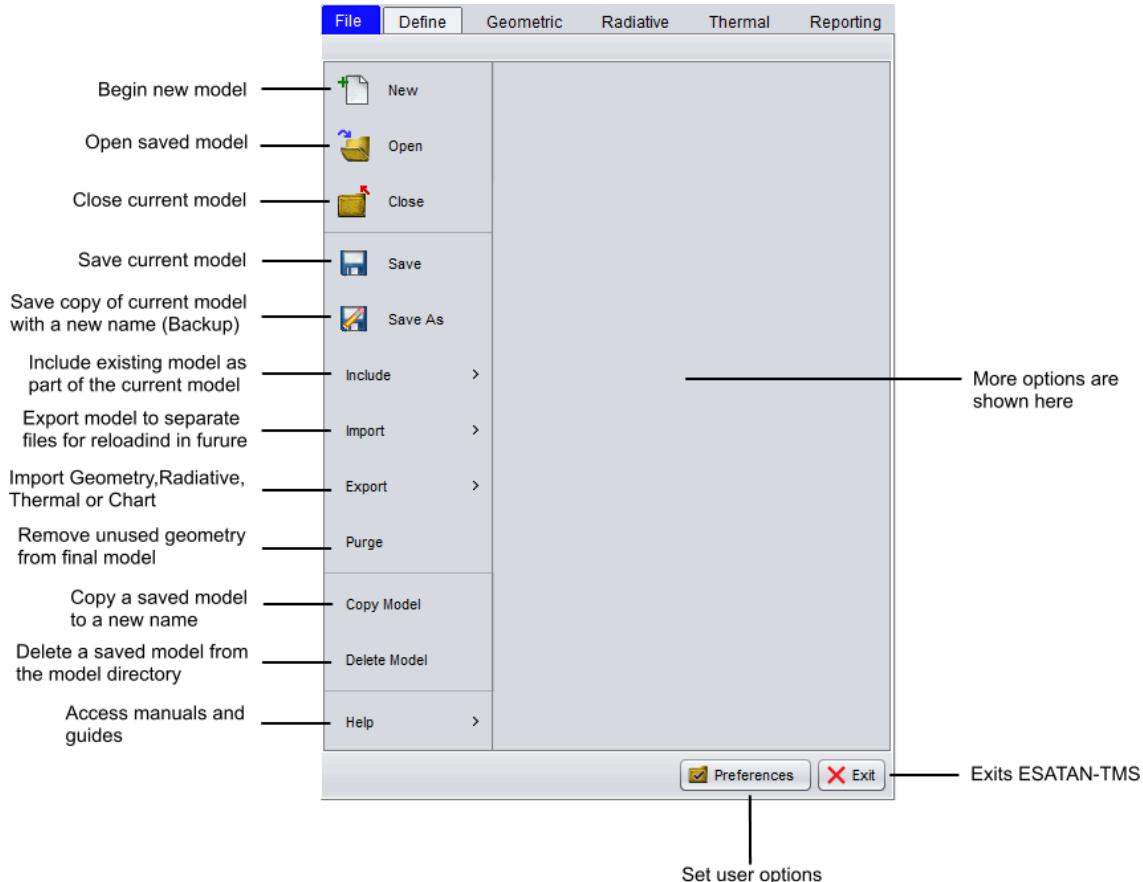


Figure 1-3 File menu

1.2.1.2 Ribbon bar

Each tab, except for File, in the menu bar will reveal the available options in its ribbon bar to provide quick access to dialogs and functions as shown in the subsections below.

1.2.1.2.1 The Define ribbon bar

The Define ribbon bar below is visible when you create a new model or open an existing model. Alternatively, you can select the Define tab in the menu bar to access the Define ribbon bar which enables you to define geometry, properties and variables.

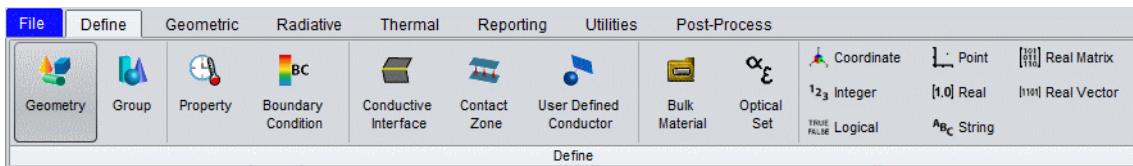


Figure 1-4 Define ribbon bar

1.2.1.2.2 The Geometric ribbon bar

The Geometric tab from the menu bar will show the below menu where you can transform the geometry, define the assembly, combine or cut the model geometry, or define recursive properties.

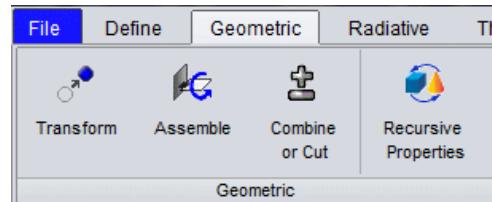


Figure 1-5 Geometric ribbon bar

1.2.1.2.3 The Radiative ribbon bar

The Radiative ribbon bar provides options to be able to open and close an existing Radiative Case, define a new case or cavity, and execute the Radiative Case.

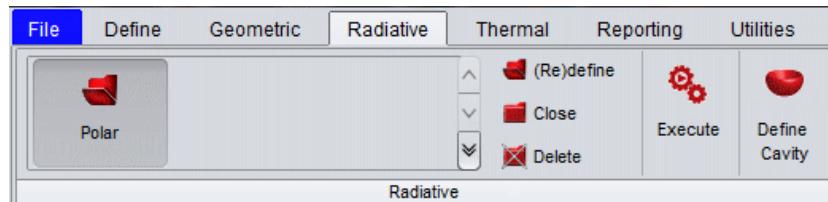


Figure 1-6 Radiative ribbon bar

1.2.1.2.4 The Thermal ribbon bar

The Thermal ribbon bar consists of two sections, Analysis Case and Thermal.

The items in the Analysis Case section allow you to select an existing Analysis Case which will open in the model tree. You can also open the Define Analysis Case dialog, and run the Analysis Case or parametric solution.

The Thermal section provides options for post-processing, generating conductive interface, calculating conductors and exporting FE temperatures.

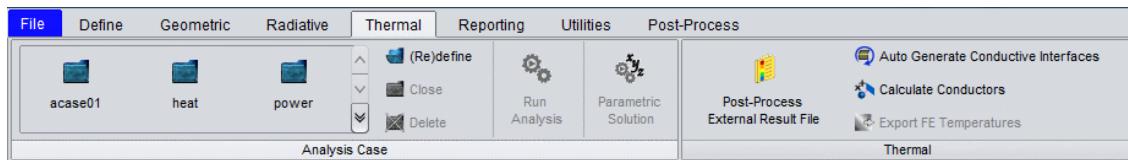


Figure 1-7 Thermal ribbon bar

1.2.1.2.5 The Reporting ribbon bar

By selecting Reporting from in the menu bar, you will reveal the options to click on to generate a report in the Reports tab in the visualisation area.

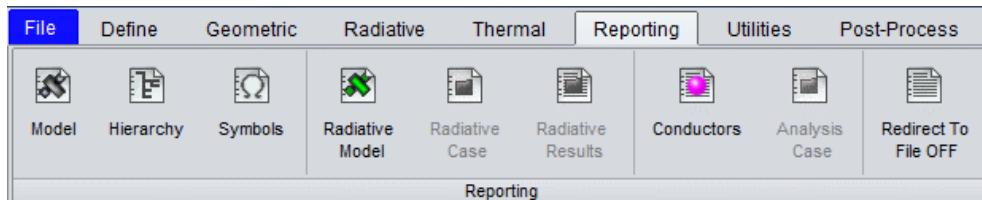


Figure 1-8 Reporting ribbon bar

1.2.1.2.6 The Utilities ribbon bar

The Utilities ribbon bar is comprised of two sections; Converters to enable you to convert your model and the second section to access Software tools.

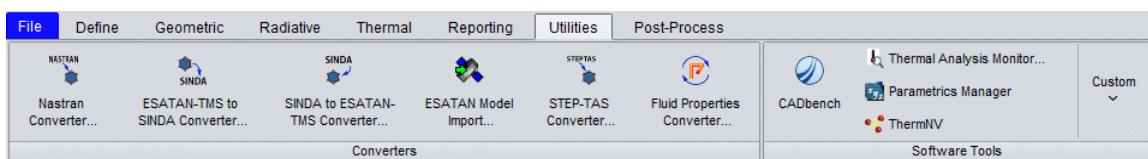


Figure 1-9 Utilities ribbon bar

1.2.1.2.7 The Post-Process ribbon bar

The Post-Process ribbon bar enables you to create and display charts for Post-Processing your Analysis Case data. You can also choose how the visualisation window is split to display the charts in the visualisation as well as update the data displayed by the chart.



Figure 1-10 Post-Process ribbon bar

1.2.1.2.8 the Visualisation Tools

The visualisation toolbar is always present under all tabs of the menu bar within each ribbon bar (to the right) and allows you to control the display aspects of your 3D model.

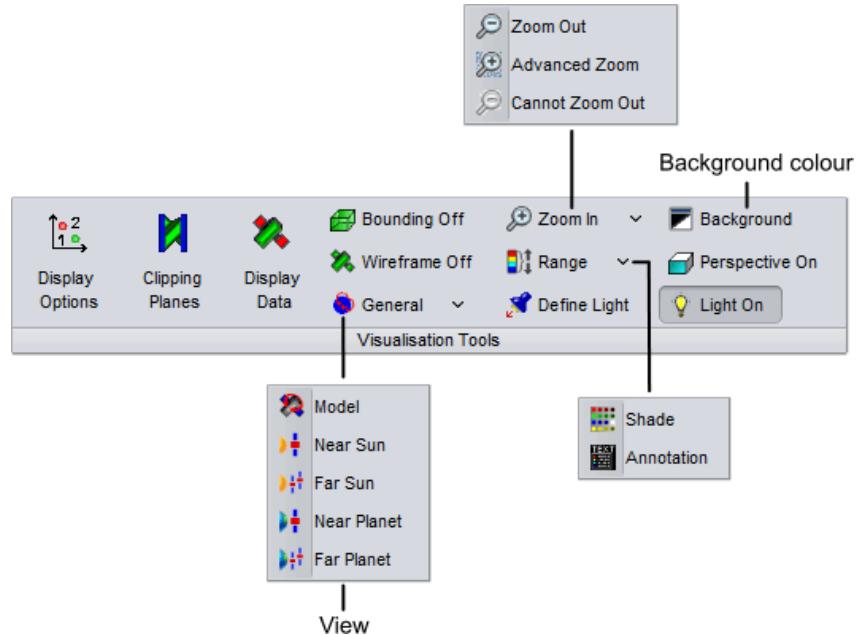


Figure 1-11 Visualisation tools ribbon bar

1.2.1.3 The Model Tree

As you build up your model, the content of the model tree increases to show all the components and entities you have defined, see Figure 1-12 below:

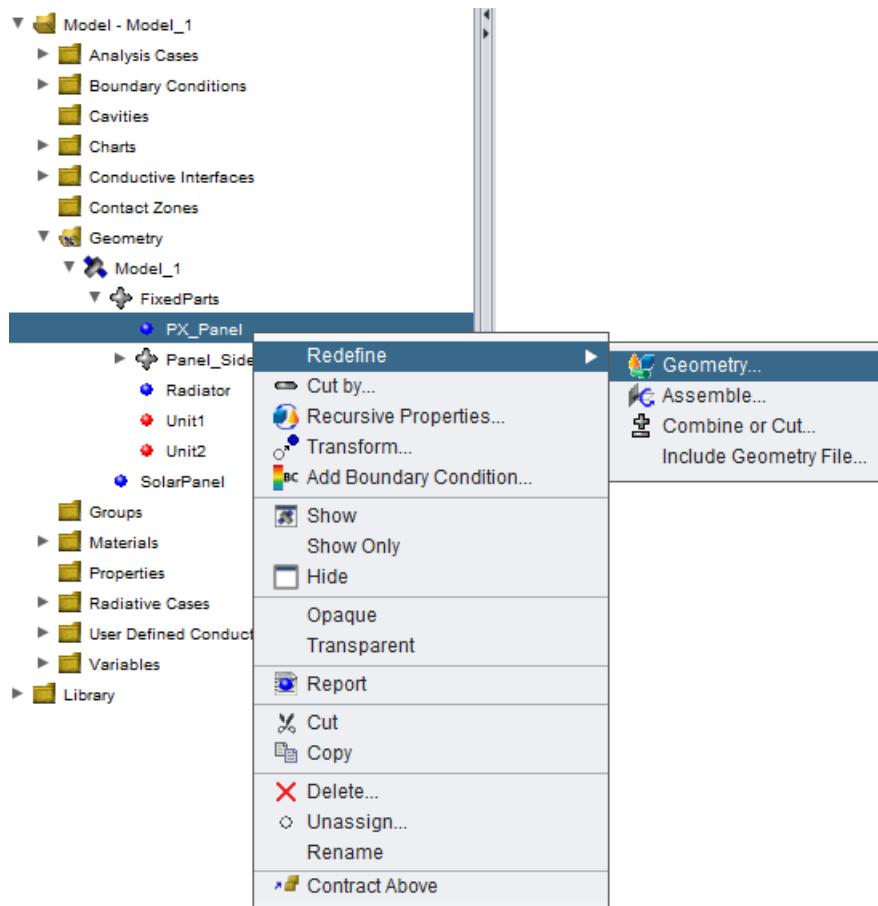


Figure 1-12 Using the model tree

The model tree works like the tree in Windows Explorer. You can double-click folders to open/close them and right-click nodes to activate their drop-down menus.

The options available on the drop-menus can normally be accessed from the Workbench menubar and/or toolbar and the icon for a given option will be the same throughout.

Using the model tree can be a more efficient way of working as you can select the subject of an operation directly and Workbench will activate the relevant module etc. automatically, saving you steps in the process.

1.2.1.4 The Search bar

You can search for contents in the model tree via the search bar located at the bottom of the model tree for quick navigation. Typing in the first few letters; such as “prop”, to locate the item you are looking for will highlight content and/or expand any related folders containing “prop”.

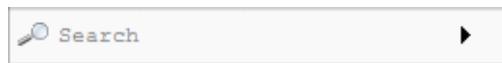


Figure 1-13 Search bar

1.2.1.5 Picking and Properties

Under the search bar, the picking and properties area will be visible. When you select a surface on your model, or a different picking mode such as point or face for example, the Picking and Properties area will expand to show the properties of the surface you have picked and will display as Figure 1-14 below.

Picking and Properties	
Property	Data
Type	Face
Primitive Name	Back_side:face3
Face Number	347
Node Number	-
Attribute	Colour
Attribute Value	CYAN
Optical Coating	White_paint["default"]
Bulk Material	-

Figure 1-14 Picking and Properties area

1.2.1.6 Video Controls

When a Radiative Case is defined, the orbit will be displayed along with the video/animation controls that allow you to start the automated display of orbit positions or display all orbit positions.



Figure 1-15 Video Controls display

1.2.1.7 Visualisation Controls

The visualisation controls enable you to set the model or overlay, or select the picking mode from a click of a button.

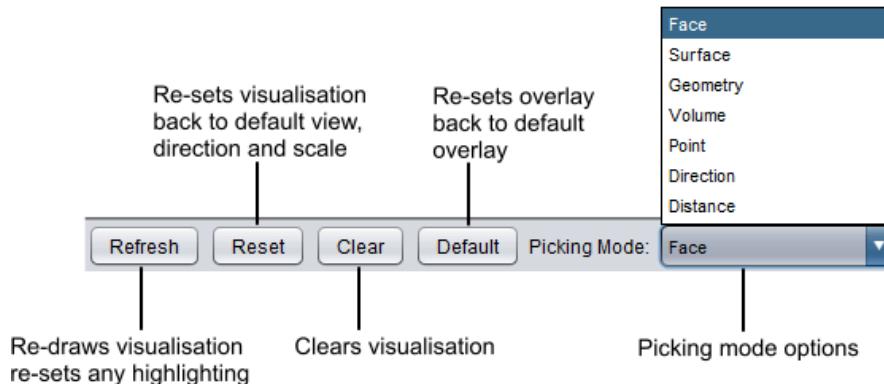


Figure 1-16 Visualisation Controls display

1.2.1.8 The Visualisation

The visualisation fills the majority of the Workbench window and consists of a Visualisation tab where your model will be displayed and a Reports tab where reports will be displayed.



Figure 1-17 The visualisation area

1.2.1.9 Status bar

The status bar runs along the bottom of the main windows and displays what the software is doing, such as the calculations it is performing, and its progress. It also shows the open radiative and/or thermal analysis case, error message and loading bar.



Figure 1-18 Status bar

1.2.1.10 The command history pop-up

Clicking on the error message icon in the status bar will display the warning error which includes information about the error.

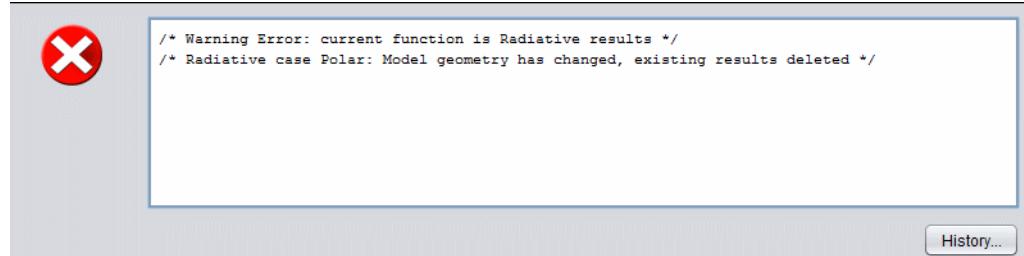


Figure 1-19 Warning error display

The History... button on this warning error displays the command history pop-up below with the relevant tab selected.

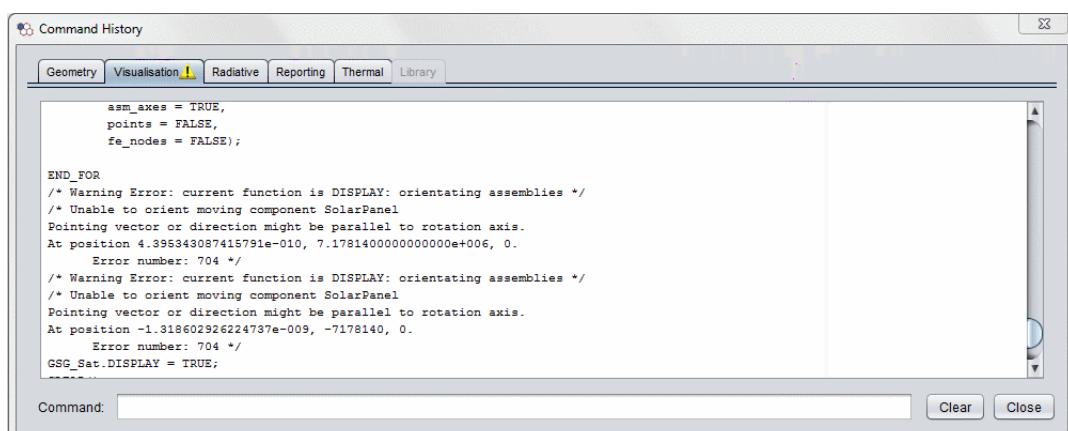


Figure 1-20 Command History pop-up

The Command History window displays the Workbench language that Workbench generates as you complete dialogs. You can also enter language directly in the Command text field in the lower part of the area.

The Command History window also displays any errors that have been generated while you are working.

1.2.1.11 Interactive Axes

The Interactive Axes are at the bottom right of the visualisation window and allow you to view the model from the X, Y, Z axes by clicking on the box or axes.

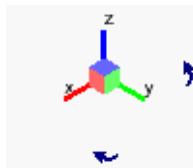


Figure 1-21

1.3 ESATAN-TMS Workbench Modules

The Getting Started Guide is designed to introduce new users to working with Workbench. There are two models to work through and the features covered in each model are shown in Table 1-1 below. The models are aimed at thermal engineers with some but not extensive experience of using Unix and the X Windows System and/or Windows and PCs. It is recommended that the models should be completed in sequence as the next model is more advanced from the last.

- Model 1 is a simple satellite model using Shell, and travels around an orbit. See Chapter 2: “Model 1 - Simple (Shell) Satellite Model”
- Model 2 is a more complex spacecraft using Solid modelling capabilities and Shells. See Chapter 3: “Model 2 - Complex (Solid) Satellite Model”

Feature	Model_1	Model_2
Shell	✓	✓
Solids		✓
Chained Radiative Case		✓
Time dependent steering	✓	✓
Clipping plane	✓	✓
Moving geometry		✓
Time/Temperature dependent properties		✓
ThermNV		✓
Transparency		✓
User Defined Conductors	✓	✓
Vector (orbit position, irregular mesh, etc)		✓

Table1-1 ESATAN-TMS features covered

1.3.1 Before you begin...

You will need to assign a preferred text editor for viewing or editing some of the files generated by Workbench. You can do this as follows:

1. Run Workbench and select File → Preferences
2. Use the Browse button next to the Preferred editor field to navigate and select the executable file for an appropriate text editor - e.g. 'wordpad.exe'.
3. When you have selected the appropriate file click Choose Editor, then click OK.

1.4 Conventions

Throughout this guide certain terms and conventions will be used in instructions:

- 'click' or 'select' = a single click on the left mouse button
- A → B = click menu 'A' and then select item 'B'
- Anything shown in *italics* = a symbol - defined or to be defined
- Anything shown in `courier` font = a filename or a Workbench output

- Field label shown in **bold** = a mandatory parameter

1.5 Acronyms

Table1-2 shows the acronyms that will be used in the instructions throughout this guide.

Acronyms	Definitions
ACG	Auto-generate conductive Generation
BC	Boundary Condition
ESATAN-TMS	European Space Agency Thermal Analysis Network Thermal Modelling Suite
GSG	Getting Started Guide
GUI	Graphical User Interface
HF	Heat Flux
IR	Infra-Red
MCRT	Monte Carlo Ray Tracing
NGN	Non-Geometric Node
OP	Optical Properties
REF	Radiative Exchange Factor
SP	Solar Panel
TMM	Thermal Mathematical Model
UV	Ultra Violet

Table1-2 Acronyms used in this guide

2. Model 1 - Simple (Shell) Satellite Model

2.1 Introduction

This chapter describes the scenario you will be modelling with ESATAN-TMS Workbench as you work through the remaining sections in this chapter. You will begin by defining a simple spacecraft and its polar orbit around Earth. You will then use Workbench to calculate the radiative exchange factors (REFs) and the solar, albedo and planetary heat fluxes (HFs) absorbed by the faces of the model as it travels around the orbit.

2.1.1 The Geometry

The spacecraft consists of a box-shaped main body, with 4 side panels plus 1 partial panel on top. The top of the ‘box’ is completed with a radiator adjoining the partial panel. The spacecraft also has a separate rotating solar panel and there are two electrical units inside the main body. See Figure 2-1

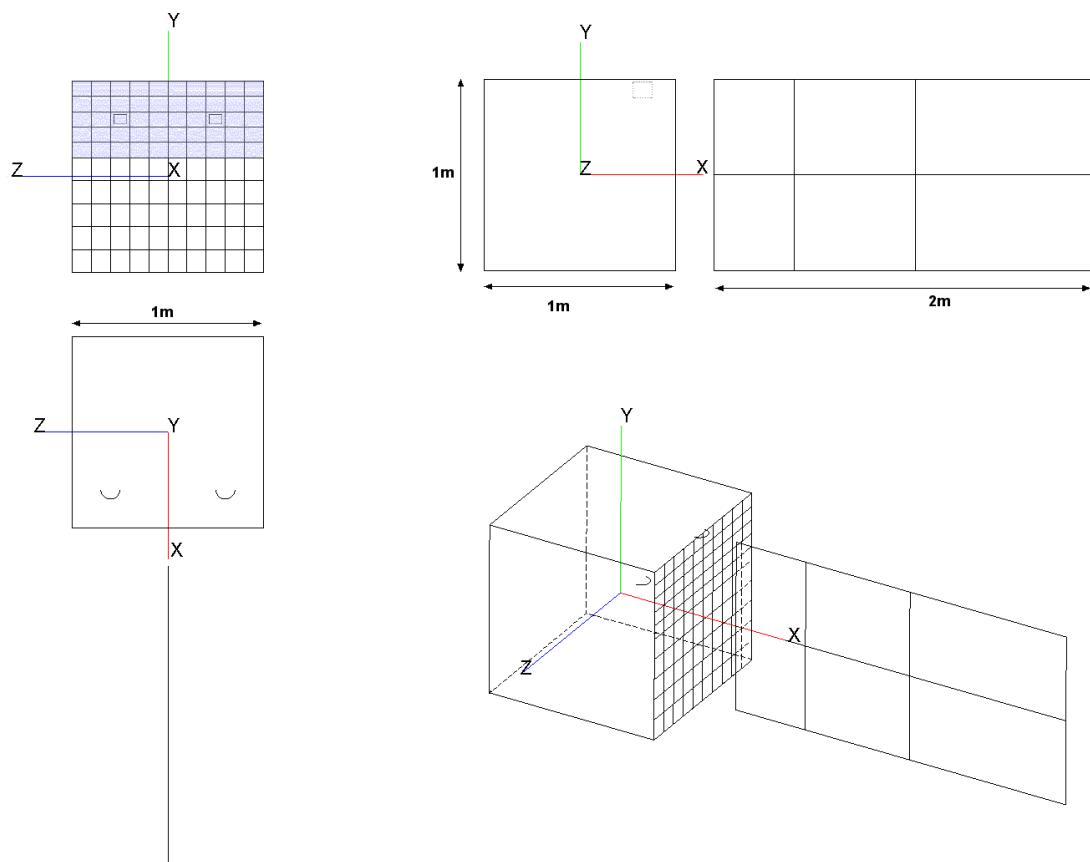


Figure 2-1 Model_1 spacecraft design

The 4 side panels have 1 radiative face on each surface. The top panel and the radiator are both subdivided into 50 faces on each surface. The solar panel is subdivided into 6 faces on each surface.

It is assumed that the panels, radiator and solar panel are made from Aluminium and then coated with one of three different optical coatings, with different thermo-optical properties. The coatings used are Multilayer Insulation (MLI), Solar Cells or Second Surface Mirror (SSM). See Figure 2-2

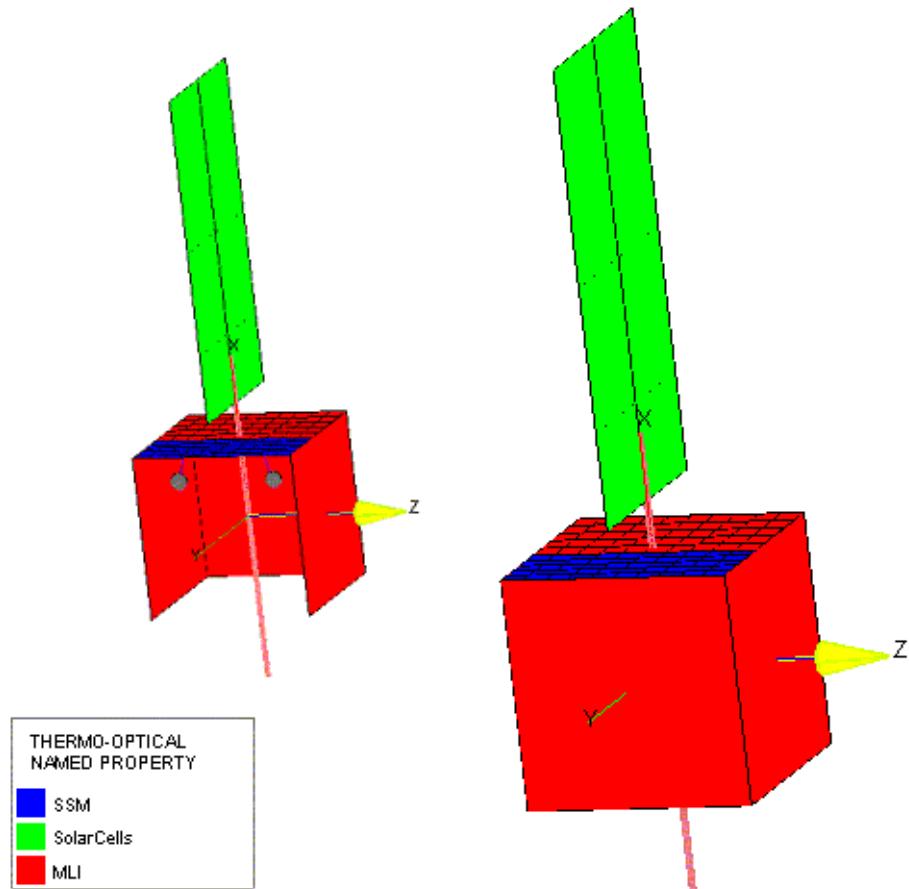


Figure 2-2 Model_1 Optical coatings

2.1.2 The Orbit

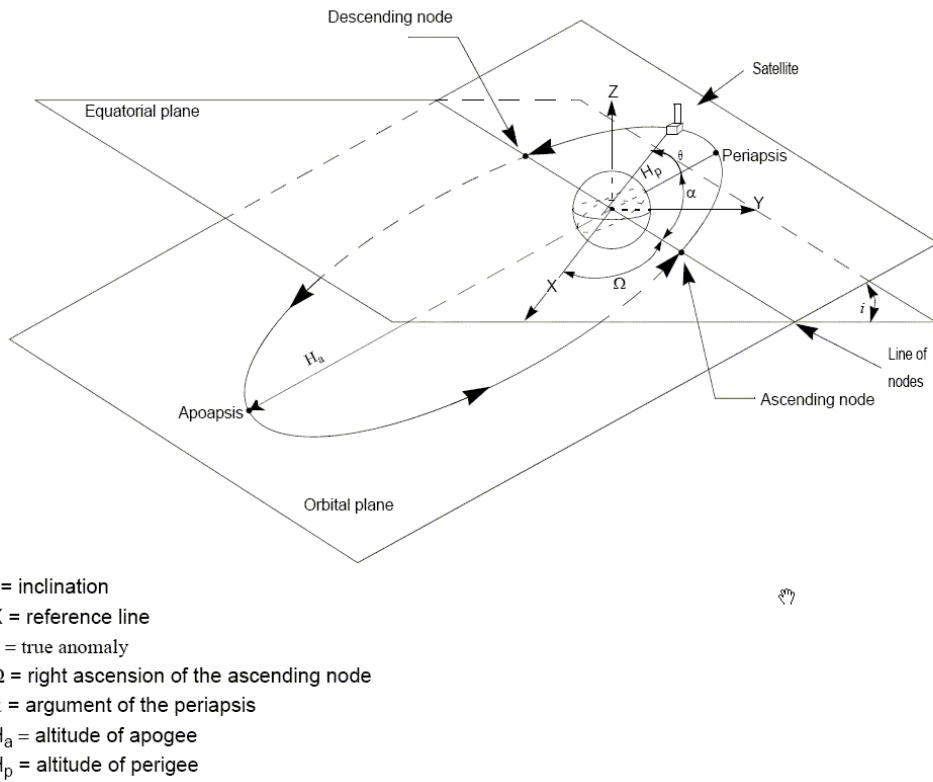


Figure 2-3 Model 1 orbit parameters

The spacecraft will follow a circular polar orbit around the Earth at an altitude of 900 km with an orbit inclination of 88° and a right ascension of 30° . The argument of periapsis (α) will be 0° and there will be 11 orbit positions. The solar panel will always point towards the Sun.

2.1.3 The Analysis

When the orbit is defined, Workbench will automatically calculate the satellite's position and orientation on the orbit at intervals of 32.7273° ($360/11$) as well as at eclipse entry and exit points. Workbench will then use a Monte Carlo ray tracing method to fire 1000 rays from each face and calculate the REFs for the spacecraft. Then the various absorbed heat fluxes (HFs) around the orbit will be calculated.

2.2 Defining the Geometry

In this section you will work through the process of building the geometric model of your spacecraft in Workbench. The main stages in this process are as follows:

- Beginning a new model

- Defining the bulk materials and thermo-optical properties
- Defining the geometry (geometric primitives - rectangles, spheres etc.)
- Defining the properties of the geometry
- Translating and/or combining the geometry
- Assigning the model
- Generating Conductive Interfaces

Note: Throughout the rest of this guide, instructions will be given for completing dialogs etc. The value of a field in a dialog should only be changed if specified in the instructions. Fields should otherwise be left as they appear when the dialog opens.

2.2.1 Defining a New Model

Before you begin, you will need to ensure Workbench has access to a valid ESATAN-TMS Workbench licence file. You can then define the name of your new model as follows:

1. Run ESATAN-TMS Workbench
2. Select File → New
3. Type your model name "*Model_1*" in the text field of the Begin a New Model dialog then click OK

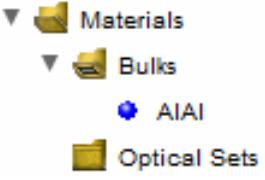
Type your model name as "*Model_2*" when completing
Chapter 3: "Model 2 - Complex (Solid) Satellite Model"

Workbench will display an error message if this model already exists. To proceed, either delete the existing model and repeat steps 1, 2 & 3 or use an alternative name and substitute that name wherever this guide refers to Model 1 or 2.

2.2.2 Defining Material Properties

2.2.2.1 Bulk Materials

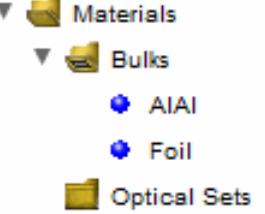
You will begin by defining the bulk material of the main panels and the solar panel - Aluminium Alloy:

<ul style="list-style-type: none"> • Select Define → Bulk Material then enter: • Bulk: = AlAI • Density = 2700 Kg / m³ • Specific Heat = 900 J / Kg / K • Conductivity = 200 W / m / K • Click Apply to complete the definition 	
--	--

The Bulks folder in the model tree will now contain an AlAI node.

Next you will define a set of ‘dummy’ material properties to represent the external reflective foil of the Multi-layer Insulation (MLI), which will be used to cover the outside of the main panels, etc.

The effects of the foil are negligible, relative to the other material properties for this spacecraft, so you can define properties that will produce a capacitance of zero for this material. This will later simplify and speed up the ESATAN-TMS Thermal process because properties with zero capacitance are excluded from the calculations for a transient solution as they are assumed to be in a steady state with the rest of the model.

<ul style="list-style-type: none"> • Amend the parameters from AlAI to: • Bulk: = Foil • Density = 0 • Specific Heat = 0 • Conductivity = 0.01 • Click Apply followed by Close 	
--	--

Selecting Define → Bulk Material again was unnecessary because the dialog was still open. This feature can save you significant time and effort and you can always check the model tree to make sure you have applied a definition before amending the properties for the next definition.

Note: If you select an alternative dialog, e.g. Define → Optical Set, before closing the current dialog Workbench will ask you to confirm your intended action. You can disable this feature by un-checking the **Display warning on closing dialog** option on the Workbench User Options dialog which you can access by selecting File → Preferences

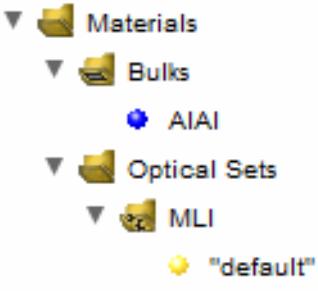
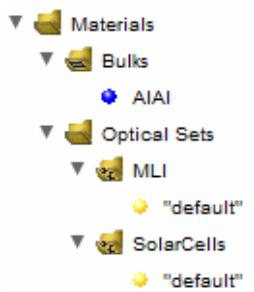
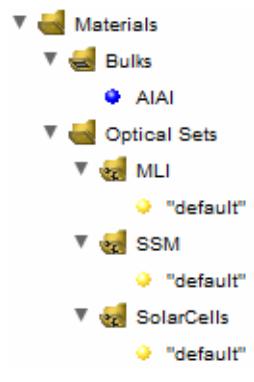
2.2.2.2 Thermo-optical Properties

You can define the thermo-optical properties of the spacecraft materials in the same way as you defined the bulk materials.

Workbench requires the sum of the four coefficients for both the Infrared and the Solar wave band to equal 1.0. If you define just Infrared Emissivity and/or Solar Absorptivity and

leave the remaining fields for each wave band blank (as for MLI and Solar Cells below) Workbench will define the Diffuse Reflectivity for the corresponding wave band to bring the total of the coefficients for that wave band to 1.0.

Define the thermo-optical properties for Multi-layer Insulation (MLI), Solar Cells and Second Surface Mirror (SSM).

<ul style="list-style-type: none"> • Define → Optical Set • Optical: = MLI • Infrared Emissivity = 0.78 • Solar Absorptivity = 0.46 • Click Apply to complete the definition 	
<ul style="list-style-type: none"> • Amend the parameters from MLI to: • Optical: = SolarCells • Infrared Emissivity = 0.84 • Solar Absorptivity = 0.75 • Click Apply to complete the definition 	
<ul style="list-style-type: none"> • Amend parameters from SolarCells to: • Optical: = SSM • Infrared Emissivity = 0.8 • Infrared Specular Reflectivity = 0.2 • Solar Absorptivity = 0.2 • Solar Specular Reflectivity = 0.8 • Click Apply followed by Close 	

2.2.3 Geometry Construction

Defining geometry that make up your geometric model is done in a similar way, but there are two tabs on the Define Geometry dialog - the Geometric tab and the Properties tab. When you first display Define Geometry dialog it defaults to the Geometric tab. See

Figure 2-4

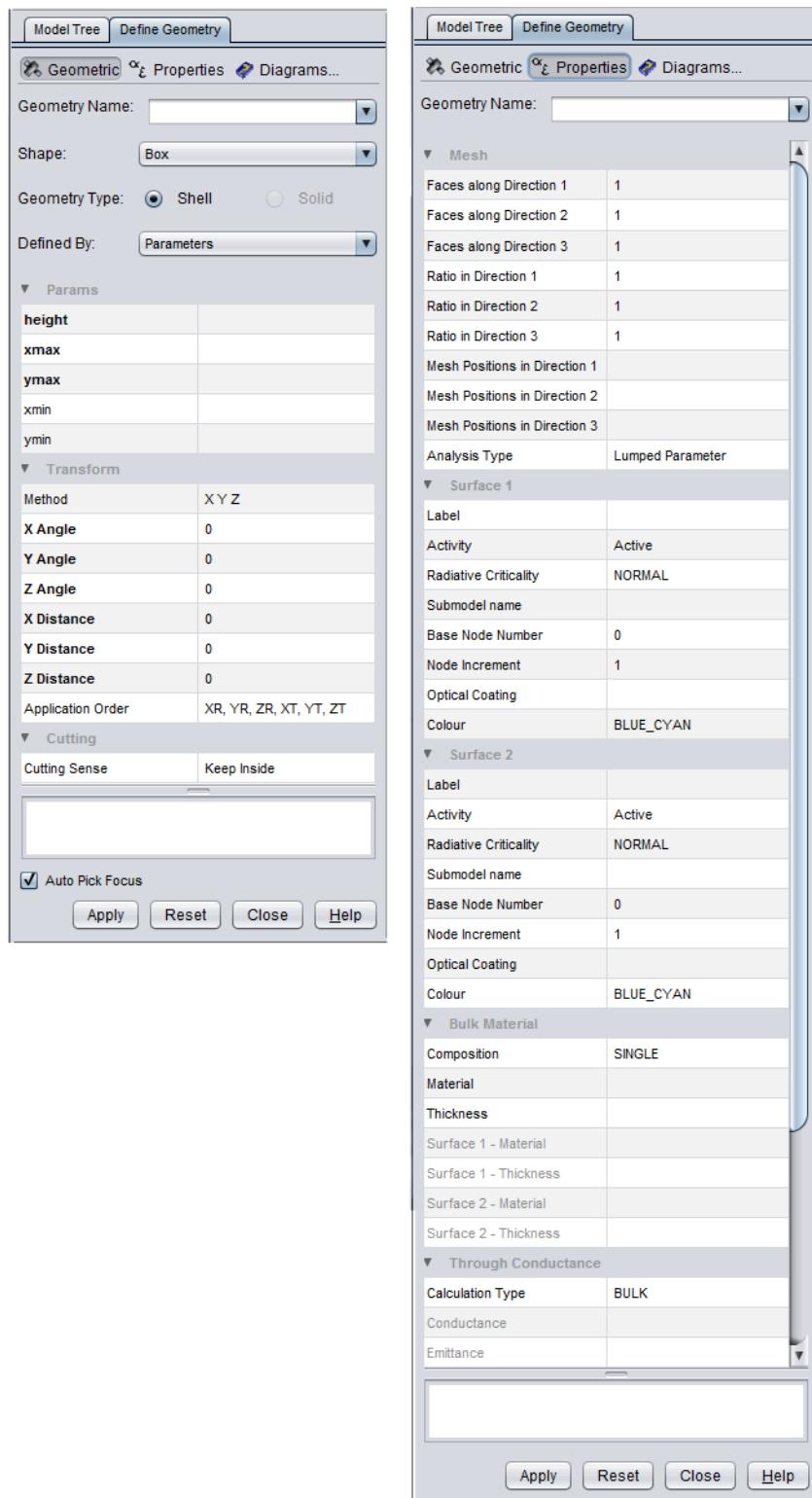
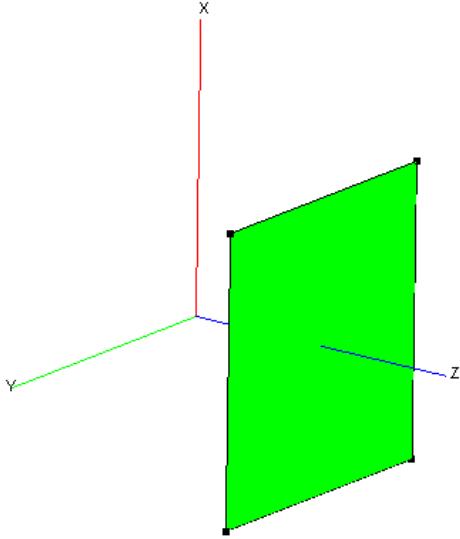


Figure 2-4 The define geometry dialog

2.2.3.1 Defining Geometry

Define the first geometry - Panel (+Z) as follows:

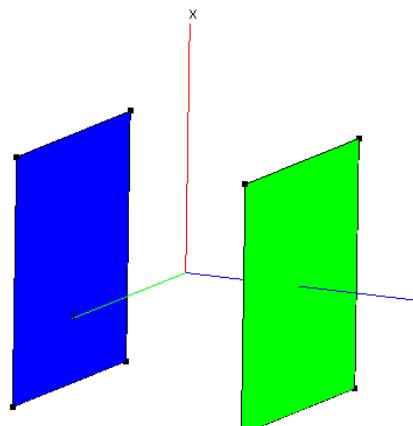
<ul style="list-style-type: none"> • Define → Geometry... • Geometry name: = PZ_Panel • Shape: = Rectangle • xmax = 1.0 • ymax = 1.0 • Switch to Properties tab at top of dialog • Surface 1 Label = PZ_Panel • Surface 1 Base Node Number = 3001 • Surface 1 Optical Coating = MLI • Surface 1 Colour = GREEN • Surface 2 Label = PZ_Panel • Surface 2 Activity = Conductive • Surface 2 Base Node Number = 2001 • Surface 2 Colour = BLUE_CYAN • Bulk Material Composition = DUAL • Surface 1 - Material = Foil • Surface 1 - Thickness = 1.0E-6 • Surface 2 - Material = AlAI • Surface 2 - Thickness = 1.0E-3 • Calculation Type = EFFECTIVE • Emittance = 4.0E-3 • Click Apply to set the geometry definition • Switch to Geometric tab at top of dialog • Reset XT, YT, ZT <ul style="list-style-type: none"> X Distance= -0.5 Y Distance= -0.5 Z Distance= 0.5 • Click Apply to reposition the geometry 	
---	---

You have applied an optical coating to surface 1 of the panel - MLI with a Foil thickness of 1.0E-6. You have applied a different node number but no optical coating to surface 2, which is defined as Aluminium - the same as the bulk of the panel. This panel will therefore show a difference in radiative heat transfer between the inside (surface 2) and the outside (surface 1). You also set the activity of surface 2 to thermally active, i.e. - not radiatively active. This means that the radiative heat transfer on the inside of this panel can be ignored.

To obtain the view shown with the instructions above you can use the mouse controls, as described in Subsection 1.2.1.8: “The Visualisation”:

- Hold down the left mouse button and drag to rotate the model
- Hold down the middle/both mouse buttons and drag to zoom in and out
- Hold down the right mouse button to pan left, right, up or down

The opposite panel - Panel (-Z) will be of the same construction as Panel (+Z). It can therefore easily be defined by using the same parameters and adjusting the translation and rotation to position it opposite to Panel (+Z). Remember to amend the label and node numbers

<ul style="list-style-type: none"> • Redefine PZ_Panel as follows: • Geometry name: = MZ_Panel • Y Angle = 180.0 • X Distance = 0.5 • Y Distance = -0.5 • Z Distance = -0.5 • Switch to Properties tab at top of dialog • Surface 1 Label = MZ_Panel • Surface 1 Base Node Number = 3002 • Surface 2 Label = MZ_Panel • Surface 2 Base Node Number = 2002 • Click Apply to complete the definition 	
--	---

So far you have defined both geometry Panel (+Z) and Panel (-Z) by the default method - Parameters. Geometry can also be defined by Points or by Directions. Please refer to the ESATAN-TMS Workbench User Manual for details on defining geometry by Directions.

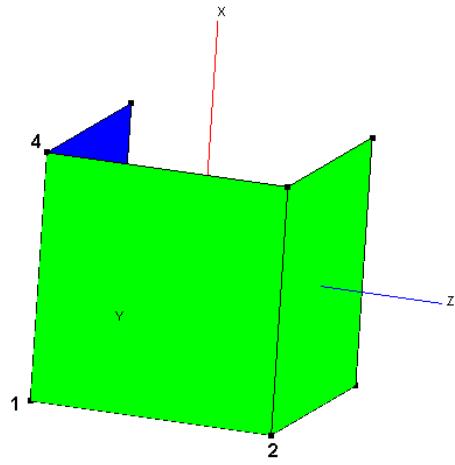
The points definition method uses specific points on the primitive to define it in space and these points can either be entered manually into the appropriate field on the dialog or they can be selected by ‘picking’ with the mouse on the visualisation (you will need to click in each field on the dialog before picking the appropriate point on the visualisation). You can now define Panel (+Y) by points.

For this method you will need to use the Display options icon in the visualisation tools on the ribbon bar to access the Display Options dialog. Check (tick) the Points box under General to switch on displaying of points on the visualisation. You can un-check the Points box to switch it off again when you have finished using points.



- Redefine PZ_Panel as follows:
- Geometry name: = PY_Panel
- Defined By: = Points
- Use  to display points on the geometry (see Figure 1-11)
- Select corner points 1, 2 & 4 with mouse:

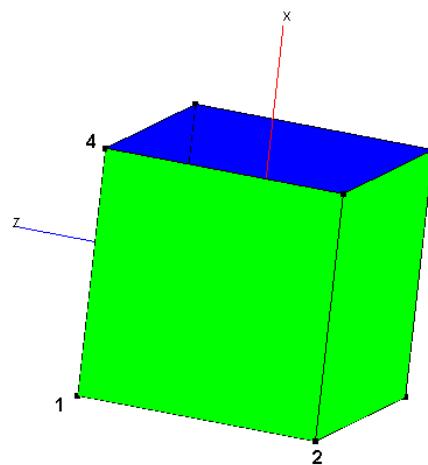
Point 1	= [-0.5, 0.5, -0.5]
Point 2	= [-0.5, 0.5, 0.5]
Point 4	= [0.5, 0.5, -0.5]
- X Distance = 0
- Y Distance = 0
- Z Distance = 0
- Switch to Properties tab at top of dialog
- Surface 1 Label = PY_Panel
- Surface 1 Base Node Number = 3003
- Surface 2 Label = PY_Panel
- Surface 2 Base Node Number = 2003
- Click Apply to complete the definition



Again, you can define the opposite panel, Panel (-Y), by making minimal changes to the definition of Panel (+Y). It is good practice to save your model at regular intervals as you define it. You can define the fourth panel and save your model by following the instructions below

- Redefine PY_Panel as follows:
- Geometry name: = MY_Panel
- Select corner points 1, 2 & 4 with mouse:

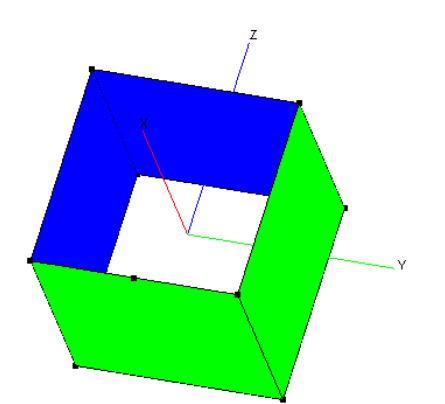
Point 1	= [-0.5, -0.5, 0.5]
Point 2	= [-0.5, -0.5, -0.5]
Point 4	= [0.5, -0.5, 0.5]
- Switch to Properties tab at top of dialog
- Surface 1 Label = MY_Panel
- Surface 1 Base Node Number = 3004
- Surface 2 Label = MY_Panel
- Surface 2 Base Node Number = 2004
- Click Apply followed by Close
- Save your model using:
File → Save



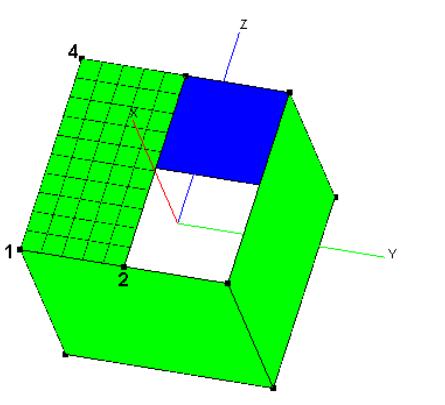
When defining geometry by points you can select the points on the visualisation as you have done here, or you can complete the point fields manually by typing 3 real values separated by commas (Workbench will automatically enter the square brackets). Alternatively, if you have previously defined any point variables, they will be available for selection in the drop-down list for each point field.

2.2.3.2 Defining Point Variables

To parametrize the geometry definition it is useful to introduce a Workbench Point Variable, MyPoint, representing the common point of the top two panels:

<ul style="list-style-type: none"> • Define → Point • Point Variable = Ticked • Name = My_Point • Method = Coordinate • Coordinates = 0.5, 0.0, -0.5 • Click Apply followed by Close 	
--	---

MyPoint can then be used in the definition of both geometry, Panel (+X) and Radiator

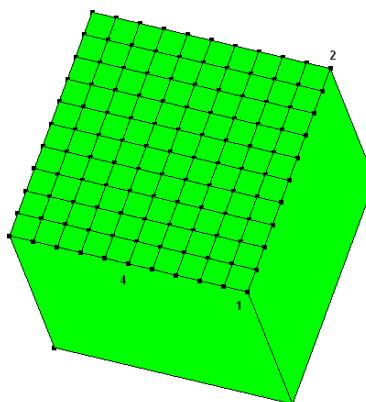
<ul style="list-style-type: none"> • Define → Geometry • Select MY_Panel then redefine it: • Geometry name: = PX_Panel • Defined By: = Points • Select corner points 1, 2 & 4 with mouse: <ul style="list-style-type: none"> Point 1 = [0.5, -0.5, -0.5] Point 2 = MyPoint Point 4 = [0.5, -0.5, 0.5] • Switch to Properties tab at top of dialog • Faces along Direction 1 = 5 • Faces along Direction 2 = 10 • Surface 1 Label = PX_Panel • Surface 1 Base Node Number = 3100 • Surface 2 Label = PX_Panel • Surface 2 Base Node Number = 2100 <p>Click Apply followed by Close</p>	
---	--

You have defined Panel (+X) using MyPoint but you have also divided this panel into 50 separate radiative faces on each surface, 5 in direction 1 and 10 in direction 2. The geometry you have defined until to now all have 1 face per surface. When Workbench outputs the thermal model, Panel (+X) will be represented by a total of 100 thermal nodes. 50 of the nodes will have the thermo-optical properties of MLI assigned to them, representing the outside of Panel (+X). The other 50 nodes will represent the inside of Panel (+X) and will therefore take the properties of the panel bulk material, AlAl. Remember, the radiative properties of the inside of this panel will be ignored as the surface 2 activity is set to thermally active (i.e. radiatively inactive).

For more details on nodal breakdowns, please refer to the ESATAN-TMS Workbench User Manual.

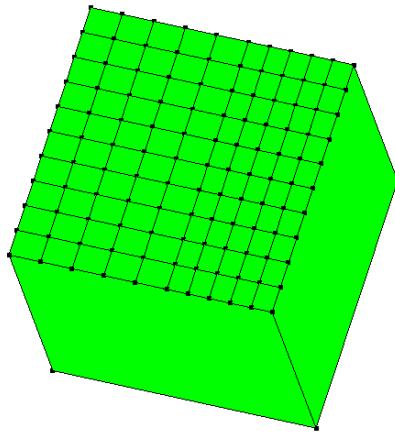
You can now define the radiator as a new geometry.

- Define → Geometry
- Geometry name: = Radiator
- Shape: = Rectangle
- Defined By: = Points
- Select corner points 1, 2 & 4 with mouse:
 - Point 1 = [0.5, 0.5, -0.5]
 - Point 2 = [0.5, 0.5, 0.5]
 - Point 4 = My_Point
 A dialog will inform you that there are multiple items at the picked location, choose Point Variable → My_Point.
- Switch to Properties tab at top of dialog
- Faces along Direction 1 = 10
- Faces along Direction 2 = 5
- Surface 1 Label = Radiator
- Surface 1 Base Node Number = 3500
- Surface 1 Optical Coating = SSM
- Surface 1 Colour = GREEN
- Surface 2 Label = Radiator
- Surface 2 Activity = Conductive
- Surface 2 Base Node Number = 2500
- Surface 2 Colour = BLUE_CYAN
- Bulk Material Composition = DUAL
- Surface 1 - Material = AlAl
- Surface 1 - Thickness = 5.0E-4
- Surface 2 - Material = AlAl
- Surface 2 - Thickness = 5.0E-4
- Calculation Type = EFFECTIVE
- Conductance = 50.0
- Click Apply followed by Close

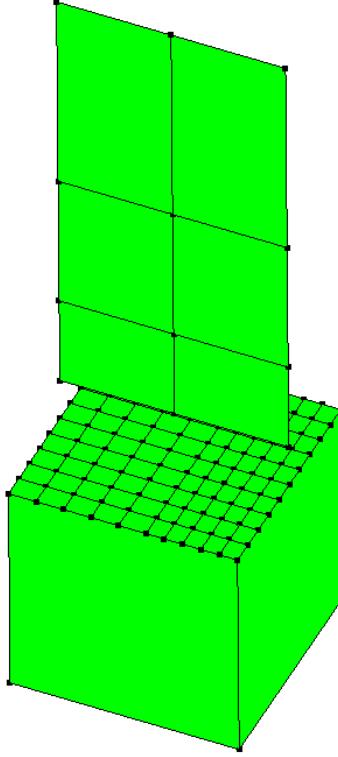


Because you have used the point variable MyPoint in the definition of both Panel (+X) and the radiator, changing MyPoint will affect both geometry. This can save time - in this case it means that when you move MyPoint to reduce the size of the radiator Workbench automatically increases the size of Panel (+X) so the two geometry are still adjoining. Obviously there is the potential to redefine geometry unintentionally when using point variables and you should bear this in mind when deciding where to use them.

- Define → Point
- Select MyPoint then redefine it:
- Y Coordinate = 0.1
- Click Apply followed by Close

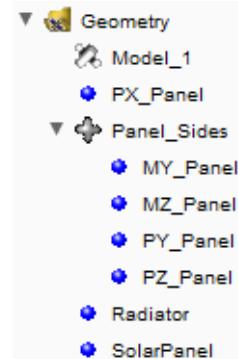


You can now define the other major component of the spacecraft- the solar panel:

<ul style="list-style-type: none"> • Define → Geometry • Geometry name: = SolarPanel • Shape: = Rectangle • Defined By: = Parameters • xmax = 2.0 • ymax = 1.0 • X Distance = 0.5 • Y Distance = -0.5 • Click Apply then redefine: • X Distance = 0.7 • Click Apply again • Switch to Properties tab at top of dialog • Faces along Direction 1 = 3 • Faces along Direction 2 = 2 • Ratio in Direction 1 = 1.5 • Surface 1 Label = SolarPanel • Surface 1 Base Node Number = 5000 • Surface 1 Optical Coating = SolarCells • Surface 1 Colour = GREEN • Surface 2 Label = SolarPanel • Surface 2 Base Node Number = 6000 • Surface 2 Optical Coating = MLI • Surface 2 Colour = BLUE_CYAN • Bulk Material Composition = DUAL • Surface 1 - Material = AIAI • Surface 1 - Thickness = 1.0E-3 • Surface 2 - Material = AIAI • Surface 2 - Thickness = 1.0E-3 • Calculation Type = EFFECTIVE • Conductance = 50.0 • Click Apply followed by Close 	
---	---

2.2.3.3 Combining Geometry

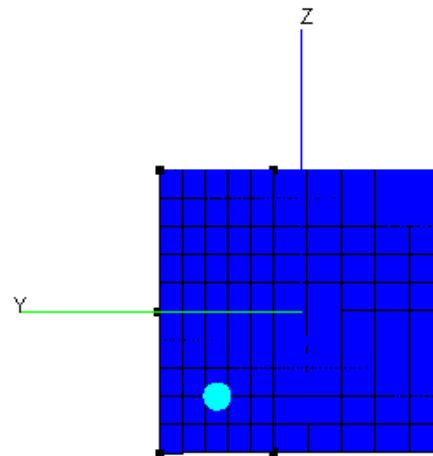
When you have defined all the geometry you can combine them to form components of the spacecraft in a logical hierarchy which will help with navigation and understanding. For this model you will combine the four side panels into a single geometry called *Panel_Sides*:

<ul style="list-style-type: none"> • Select Geometric → Combine or Cut • Target Geometry: = Panel_Sides • Select geometry from Source Geometry (hold down Ctrl key and make selections): PZ_Panel MZ_Panel PY_Panel MY_Panel • Click Add • Click Apply to combine the geometry 	 <pre> ▼ Geometry Model_1 □ PX_Panel ▼ Panel_Sides □ MY_Panel □ MZ_Panel □ PY_Panel □ PZ_Panel □ Radiator □ SolarPanel </pre>
--	---

2.2.3.4 Defining Non Geometric Thermal Nodes

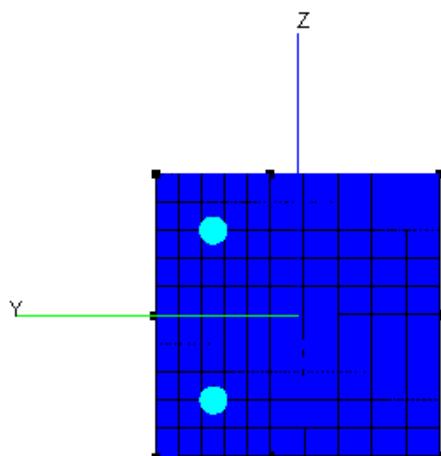
In addition to the main panels and the radiator, this spacecraft also has two electrical units inside the main body. These components do not need to be defined geometrically but still need to be accounted for in the thermal model. Workbench allows for this by supporting the definition of Non Geometric Thermal Nodes - a modelling technique which simplifies the behavior of a component and represents it as a single thermal node in the thermal model. Define Unit1 and Unit 2 as shape Non Geometric Thermal Node:

- Right-click model tree node **Panel_Sides** and select **Hide**
- Click the red face of the Interactive Axes box to view from -X direction
- Define → **Geometry**
- Geometry name: = **Unit1**
- Shape: = Non Geometric Thermal Node
- Enter Origin: = [0.5, 0.5, -0.5]
(bottom left corner point)
- Node Number = 9001
- Label = **Unit1**
- Capacitance Method = **VALUE**
- Value = 5000
- Then click **Apply** followed by **Close**
- Right-click model tree node **Unit 1**, select **Transform**
- Transform Type: = **Relative**
- Method: = **X Y Z**
- X Distance: = -0.2
- Y Distance: = -0.2
- Z Distance: = 0.2
* use TAB key or mouse to change field
- Click **Apply** followed by **Close**



Note that you could have set the translation for Unit1 on the Define Geometry dialog, as done for Unit2 below. However, by using the model tree you achieved the same result via the Geometry Transform dialog.

- Define → **Geometry**
- Geometry name: = **Unit2**
- Shape: = Non Geometric Thermal Node
- Enter Origin: = [0.5, 0.5, 0.5]
(Top left corner point)
- Node Number = 9002
- Label = **Unit2**
- Capacitance Method = **VALUE**
- Value = 5000
- X Distance: = -0.2
- Y Distance: = -0.2
- Z Distance: = -0.2
- Click **Apply** followed by **Close**



The capacitance for a Non Geometric Thermal Node can be defined explicitly as you have done in this example, or calculated from the associated properties.

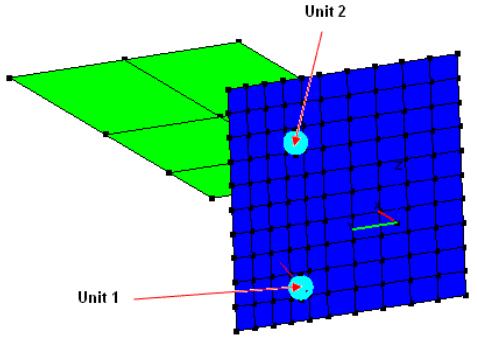
Non Geometric Thermal Nodes can be represented as spheres on the visualisation to aid model validation. An Origin and a Radius can be defined but these are only used for visualisation. If the Origin is not defined then the geometry will not be shown in the visualisation but will still be included in the generated thermal network. The default value for the Radius is 0.05m.

The capacitance will either be CALCULATED using values you specify for the Volume and the Bulk Material or Workbench will use a fixed VALUE for the capacitance which you can enter in the Value field as you have done here.

Non Geometric Thermal Nodes are shown with red icons in the Geometry section of the model tree. Please refer to the ESATAN-TMS Workbench User Manual for more detail on Non Geometric Thermal Nodes.

2.2.3.5 Defining User Defined Conductors

Once you have defined your Non Geometric Thermal Nodes you will likely need to define some corresponding User Defined Conductors. For this model you can define a single conductor from each electrical unit to Panel (+X):

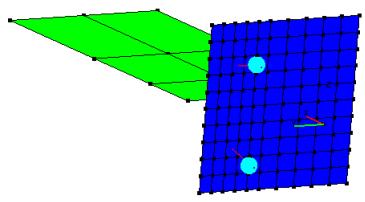
<ul style="list-style-type: none"> • Expand the model tree if necessary to view the contents of the User Defined Conductors folder • Right-click the Conductive folder on the model tree and select Define • Conductor: = Unit1_2_Panel • Click the red face of the Interactive Axes box to view from -X direction if required • Using mouse - Source Reference= Unit1 • Destination Reference: <ul style="list-style-type: none"> = Radiator:face46 (3rd face along and up from bottom left corner) • Definition Method = VALUE • Value = 0.4 • Click Apply followed by Close 	
--	--

By selecting the **Conductive** folder in the model tree you have chosen to define a conductor of type **Conductive**. You can also define **Advective**, **Convective** and **Radiative** conductors in the same way.

You can define the Source and Destination reference as either a Face, a Thermal Node or a Geometry Surface. Here you have chosen the **Source Reference** as a Face (Unit1) and the **Destination Reference** as a Face (Radiator - face 46).

Note: only the Source Reference OR the Destination Reference can be a geometry surface.

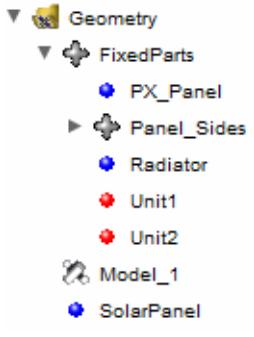
The Value of the conductor can either be specified directly, as you have done here, or it can be calculated from the Advective/Conductive/Convective/Radiative Factors (depending on the type of conductor).

<ul style="list-style-type: none"> • Select Define → User Defined Conductor • Conductor: = Unit2_2_Panel • Click the red face of the Interactive Axes box to view from -X direction if required • Using mouse - Source Reference= Unit2 • Destination Reference: =Radiator:face56 (3rd face along and down from corner) • Definition Method = VALUE • Value = 0.4 • Click Apply followed by Close 	
---	--

Please refer to the ESATAN-TMS Workbench User Manual for more detail on Non Geometric Conductors.

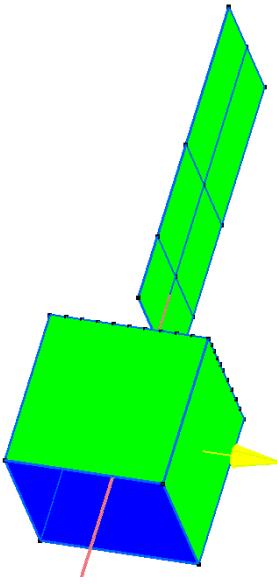
2.2.3.6 Defining Fixed Components of the Geometry

So far all the geometry in your model are fixed in relation to each other. In reality the Solar Panel on this spacecraft will rotate to face the Sun as the spacecraft travels around the orbit. You can define the parts of the model that will remain fixed by combining geometry just as you did for Panel_Sides:

<ul style="list-style-type: none"> • Select Geometric → Combine or Cut • Target Geometry: = FixedParts • Select geometry from Source Geometry (hold down Ctrl'key and make selections) <ul style="list-style-type: none"> Panel_Sides Unit2 Unit1 PX_Panel Radiator • Click Add • Click Apply to combine the geometry 	
--	---

2.2.3.7 Model Assignment

Before you can use the other modules in Workbench your model must be assigned. This process defines which geometry will be used to build the model and also defines the reference and moving components:

<ul style="list-style-type: none">Right-click main model (Model_1) in the model tree, select Assign → Assemble and add:Reference Component = FixedPartsMoving Component = SolarPanelPredefined orientation = True SunPointing Vector = [0.0, 0.0, 1.0] (+Z)Rotation Axis = [1.0, 0.0, 0.0] (+X)Click Apply to complete the definitionFollowed by Close	
---	---

Model assignment is really just a special case of a combination, where the target geometry is the name of the model. As you assign your model you may notice that Workbench briefly displays a message in the status bar informing you that the model has been assigned. The model tree icon will also have changed to a coloured icon.

2.2.3.8 Generating Conductive Interfaces

The final stage in defining the geometric elements of your model is to define the connections where geometry make contact with each other. These connections are known as conductive interfaces and they are used by the thermal analysis process to generate conductive couplings between geometry (e.g. GL conductors in ESATAN-TMS Thermal).

You can define conductive interfaces individually but Workbench can also search them out and define them automatically:

<ul style="list-style-type: none"> • Thermal → Auto Generate Conductive Interfaces • Expand the Conductive Interfaces folder in the model tree • Expand the Fused folder • Click on the first Conductive Interface (ci_1), hold shift and select the last Conductive Interface (ci_9) in the list. • Right-Click and select Set Connection Type • In the Set Conductive Interface Connection Type dialog select: <ul style="list-style-type: none"> • Connect Type = CONTACT • Contact Conductance W/m²K): = 1.0 • Click OK • Use the visualisation to select the interface between PX_Panel and Radiator. <p>The selected Conductive Interface will be highlighted in the model tree (ci_10).</p> <ul style="list-style-type: none"> • Right Click → Redefine to open the Define Conductive Interface dialog • Connect Type = FUSED • Click Apply followed by Close • Save your model using: <p>File → Save</p>	
--	--

Workbench cannot automatically determine the type of contact at each conductive interface so it will set them all to FUSED by default and show them as yellow lines in the visualisation. You can then set the Connect Type to CONTACT (implying a contact conductance over the interface), just as you have done in this example. The Contact interfaces will then appear as orange lines in the visualisation and the Fused interface as a yellow line.

2.2.3.9 Purging the Model

Before you leave the geometry module it is good practice to purge your model. This removes any geometry which do not make up the final description of the model and helps Workbench work more efficiently in future sessions using this model.

You purge the model by selecting File → Purge

In this example there is no redundant geometry so purging the model is unnecessary.

At this point the geometry is fully defined and should be saved using **File → Save**.

2.2.4 The Geometry Logfile

You have seen how the Workbench language statement of each action is ‘echoed’ to the history area. In addition to this, a log file is produced in the directory where Workbench was started.

The logfile contains all the Workbench statements generated. We have noted that a statement is generated each time you finish an action so the logfile therefore contains all the commands required to repeat the definition of the model. In the event of a model being required after it has been deleted the properly cleaned logfiles should be run to restore the model data files.

The name of the logfile is formed from the model name together with a letter denoting the Workbench module producing the logfile. If a logfile already exists with the same name the new file is appended to the existing file and there will be two or more sections to the log file separated by an END_MODEL/BEGIN_MODEL pair between comments.

The logfile produced by the Workbench geometry session you have just run will be called *Model_1_g.log*. It will be located in the user subdirectory of the *Model_1* subdirectory of the *ESATAN-TMS_Models* directory. The exact location of that directory can be found in the Workbench User Options dialog (**File → Preferences**).

Once a model is correct, the geometry log file should be kept as a record of how to build that model. However, because you have re-edited the model, a copy of the geometric model log file should really be edited and ‘cleaned’ to make it simpler.

Note: You should always use a copy of the log file in case something happens while you are editing and you lose the file.

The easiest way to do this is to take a copy of the log file (for this example, call the copy *Model_1.erg*) and clean it of any errors, repeat definitions, etc. that you may have generated in completing this example (see Subsection 2.2.4.1: “Cleaning the Logfile” for details). Then use the batch version of the geometry module (*esrdg*), and redirect its input from this cleaned copy of the logfile (also called an Workbench geometry input file):

```
$ esrdg < Model_1.erg
```

Note: Ensure the *ESATAN_TMS_DIR* Environment Variable is set to match the current ESATAN-TMS installation.

The Workbench geometry input file can also be loaded from the user interface using **File → Import → Geometry**. (Note that the file extension ‘.erg’ is adopted as the standard for a Workbench geometry input file). The above command will perform everything you have done in this chapter without the need for any interaction. As you can see, logfiles are valuable things to keep and in general they should not be edited or deleted.

IMPORTANT NOTES: The log file itself should not be used as an input file as the software will then be writing to the end of the file while reading down from the top, thus creating an infinite loop. If Workbench displays errors while trying to write the BEGIN_MODEL command to the log file, you probably do not have write permission to the ESATAN-TMS_Models directory. Use File → Preferences and reset the User model directory.

2.2.4.1 Cleaning the Logfile

The act of cleaning a log file copy is best described as reducing the log file copy to the minimum number of valid Workbench statements required to build the model correctly. The general approach is to work from both ends of the log file at the same time using the following method:

1. Start at the command after the BEGIN_MODEL line at the top of the file
2. Check from the end of the file for a command with the same left hand side
3. If you find one, replace the first occurrence of the command with the most recent occurrence (nearest the end of the logfile) - this will be the correct version of that same command
4. Delete all occurrences of the command except the one you have just replaced
5. Repeat steps 2 to 4 until you have progressed right through the file.

2.3 Running the Radiative Case

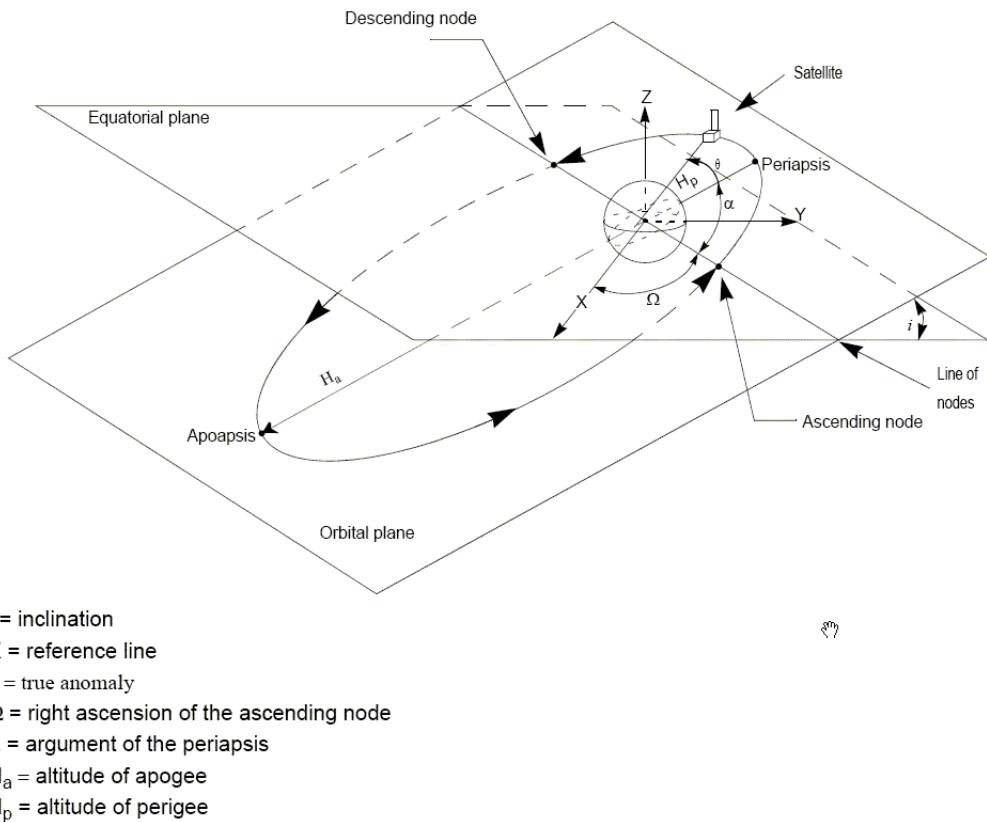


Figure 2-5 Orbit parameters

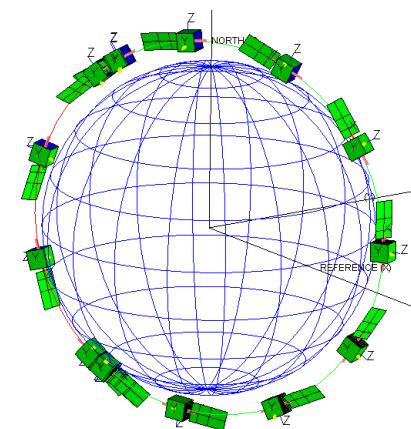
Once you are happy with the definition of the geometry for your model you can use the Radiative module in Workbench to define the orbit parameters and run the radiative calculations.

Workbench makes it extremely easy to run a range of simulations with different parameters while keeping a clear track of your work through the use of Radiative Cases. Each Radiative Case will relate to a single radiative analysis for a single spacecraft with a single behaviour definition in a single orbit. Any changes to the spacecraft, its behaviour, the orbit or the analysis run will require an additional Radiative Case.

2.3.1 Radiative Case Definition

To begin defining your Radiative Case you can either select Radiative → (Re)define from the menubar or you can use the model tree for a short-cut to the Radiative Case Dialog as described below:

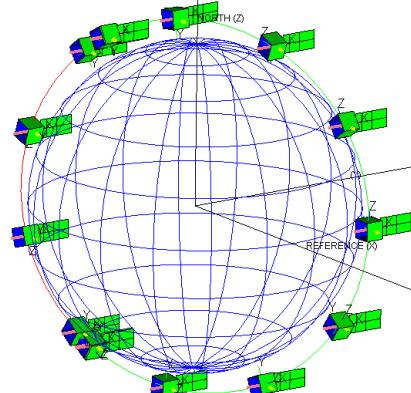
- Right-click Radiative Cases in the model tree and select Define...
- Radiative Case: = Polar
- Select Orbit at the top of the dialog
- Set
 - Altitude of Apogee = 900,000 (m)
 - Altitude of Perigee = 900,000 (m)
 - Inclination = 88 (deg)
 - Right Ascension = 30 (deg)
 - Number of positions = 11
- Click Apply to set the definition



Here you have defined the orbit as described in Section 2.1 - your spacecraft will follow a circular polar orbit around the Earth at an altitude of 900 km with an orbit inclination of 88° and a right ascension of 30°. The argument of periapsis (α) will be 0° and there will be 11 orbit positions.

The next step is to define the behaviour of your spacecraft:

- Select Pointing at the top of the dialog
- Under Primary Pointing set
 - Pointing Vector = +X axis [1,0,0]
 - Pointing Direction = GENERAL
 - General Direction = +Y axis [0,1,0]
- Under Secondary Pointing set
 - Pointing Vector = +Y axis [0,1,0]
 - Pointing Direction = NADIR
- Click Apply to change the definition
- Click OK if a Workbench Information or error message appears
- Click Close to exit the dialog



A spacecraft is defined with respect to the Model Co-ordinate System (MCS) and must then be oriented with respect to the The Inertial Coordinate System (ICS). There are three ways to do this in Workbench but for this example we will use pointing vectors and directions as this is the recommended method for normal use. Please refer to the Workbench User Manual for more information on the MCS, ICS and alternative methods of orienting your spacecraft.

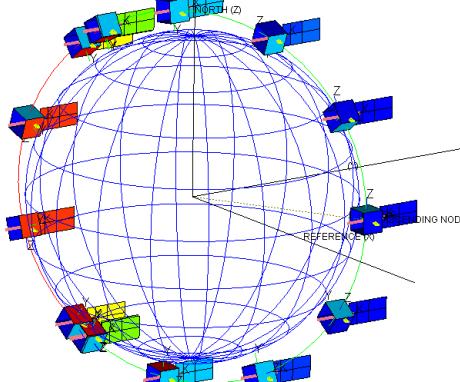
Both the Primary and the Secondary Pointing Vectors are given in the MCS but when using the GENERAL pointing direction the pointing vector specified for the spacecraft will be

aligned to the ICS as specified by the general direction. By default, the craft is oriented to the zenith and normal to the orbit.

In this example the Sun will be located along the +X axis of the ICS. Our model should always have its solar panel facing the Sun so you have set the Primary Pointing Vector to the +X axis (of the MCS) and the Pointing Direction to GENERAL with General Direction set to the +Y axis (of the ICS). You have also set the Secondary Pointing Vector to the +Y axis and the Secondary Pointing Direction to the NADIR to define the orientation of the spacecraft in its orbit (PY_Panel will always face the centre of the planet).

2.3.2 Executing a Radiative Case

Now you have defined your orbit and the spacecraft attitude you are ready to execute your radiative case and post-process the results. To do this you can either select Radiative → Execute in the ribbon bar or you can use the model tree to the Execute Dialog as described below:

<ul style="list-style-type: none"> Right-click Polar under Radiative Cases in the model tree and select Execute... Click Execute at the bottom of the dialog Wait for Workbench to finish the execution Select the Display Data icon in the ribbon bar. Category: = Radiative cases Display: = Heat Flux (W/m²) Mode: = Orbital Results Result: = Planet Flux Model Scale: = 120 Click Display at the bottom of the dialog 	
---	---

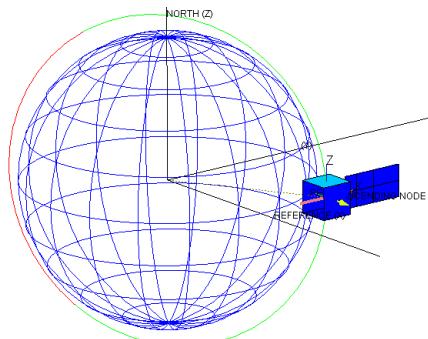
You can use the Execute Dialog to specify a range of parameters for your calculations but for this example the default settings are sufficient so it was not necessary to change anything.

Workbench will have processed a large amount of generated language during the calculations and will also have placed a small red folder called 'Polar' in the status bar. The display in the status bar indicates that Workbench currently has the Radiative Case 'Polar' open. When a Radiative Case is open Workbench can make assumptions about data you would otherwise have to generate and enter yourself so you can see that using Radiative Cases can make your work simpler and save you time.

In the above steps you also used the Display Data dialog to set suitable parameters which enabled you to verify some of the results of your calculations using the visualisation. Here you can clearly see that the planet absorbed flux is greatest when the spacecraft is behind the planet (in relation to the Sun).

You can also use the animation controls to view the model step-by-step at each point around the orbit by following the steps below:

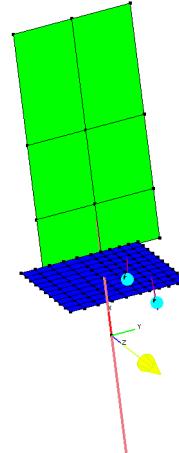
- Change the Model Scale: = 200
- Click Display at the bottom of the dialog
- Click Light off/on icon to turn it off
- Use animation controls (see Figure 1-15) to view flux around the orbit
- Stop animation and return to dialog
- Click Close to exit the dialog



Here you can view the results in more detail at each orbit position and see how they change as the spacecraft moves from one position to the next. You can view the spacecraft without viewing the planet, which can be helpful if the planet is obscuring your view at certain points or the visualisation is too crowded with information. You can remove and reinstate the display of the Model, the Orbit, the Periapsis and the Ascending node in the same way.

For the next section in this guide, Subsection 2.4: “Preparing for Thermal Analysis Case Control”, you will need to reset the view of the model as follows:

- Select Display Data in the ribbon bar
- Category: = Geometry
- Display: = Activity
- Mode: = Single Result
- Click Display at the bottom of the dialog
- Alternatively, you can select Default in the visualisation toolbar
- Click Display
- Right-click model tree node Panel_Sides and select Hide
- Use the mouse on the visualisation to make the view angle similar to that shown on the right.



2.4 Preparing for Thermal Analysis Case Control

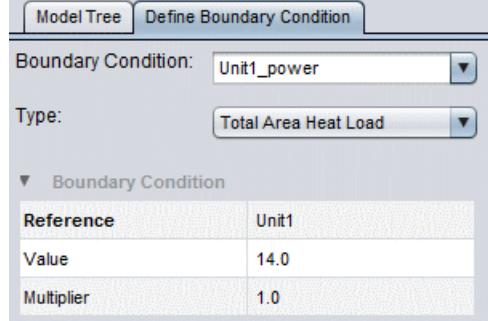
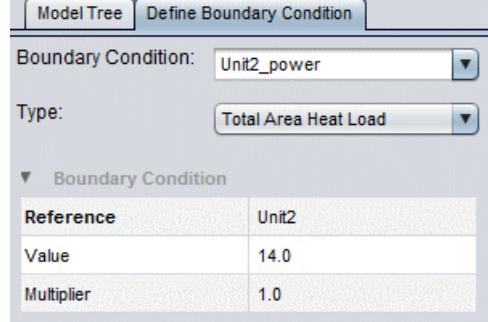
When you have calculated and verified the radiative results, the next step in the modelling process is to run the thermal model (thermal solution). This is done via the Thermal Module. Using this module you can have ESATAN-TMS Workbench calculate conductive couplings and output a thermal model in terms of a thermal network of nodes, conductances

and materials, ready for input to a thermal analyser. The thermal analyser can then use this data to predict steady-state or transient temperature solutions using the thermal network technique.

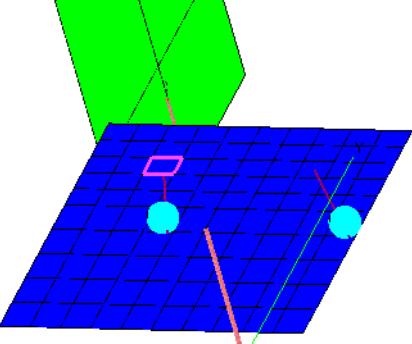
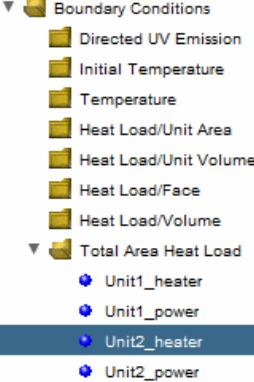
Before you begin defining your Analysis Case(s) you should make sure you have defined any boundary conditions you require as described below.

2.4.1 Defining Boundary Conditions

In this example you will start by returning to the Geometry module (via a model tree short-cut) to define boundary conditions for the two electrical units inside your spacecraft.

<ul style="list-style-type: none"> Right-click Unit1 geometry in the model tree and select Add Boundary Condition Or Define → Boundary Conditions Boundary Condition: = Unit1_power Type: = Total Area Heat Load Reference should read Unit1 Value = 14 (Watts) Click Apply to complete the definition 	
<ul style="list-style-type: none"> Edit the parameters for Unit1_power to: Boundary Condition: = Unit2_power Reference = Unit2 Value = 14 (Watts) Click Apply to complete the definition 	

When you have defined boundary conditions Unit1_power and Unit2_power you can modify these definitions as shown below to create a second set of boundary conditions - Unit1_heater and Unit2_heater:

<ul style="list-style-type: none"> • Edit the parameters for Unit2_power to: • Boundary Condition: = Unit1_heater • Type: = Total Area Heat Load • Select the Reference field, then pick the radiator face at the location where Unit1 is connected (see image right) - Reference should now read Radiator:face46 • Value = 6 (Watts) • Click Apply to complete the definition 	
<ul style="list-style-type: none"> • Edit the parameters for Unit1_heater to: • Boundary Condition: = Unit2_heater • Type: = Total Area Heat Load • Select the Reference field, then pick the radiator face at the location where Unit2 is connected (see image right) - Reference should now read Radiator:face56 • Value = 6 (Watts) • Click Apply to followed by Close 	

You are now ready to begin using the Thermal module.

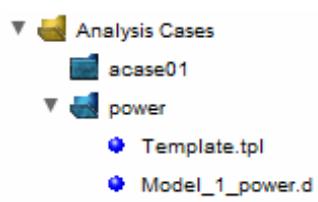
2.5 Solving with ESATAN-TMS Thermal

If you have defined a Radiative Case for your model, as in this example, the thermal model will be controlled and run via an Analysis Case. The thermal model can be constructed automatically from a template file that contains the structure of the analysis file, reference to the ESATAN-TMS Workbench results, logic for controlling the run and reference to any externally included data. The analysis file will then be used by the thermal analyser to solve the thermal model, the results of which can then be post-processed in Workbench.

2.5.1 Analysis Case Definition

2.5.1.1 Applying Boundary Conditions to Your Analysis Case

You can begin defining your Analysis Case as follows:

<ul style="list-style-type: none"> • Thermal → (Re)define • Analysis Case: = power • Solver: = ESATAN • Analysis Case Type:= Single Radiative Case • Radiative Case = Polar • Select <input type="button" value="..."/> next to the Boundary Conditions field to open the Select Boundary Conditions dialog. • Select (tick) Unit1_power and Unit2_power then click OK • Click Apply to define the Analysis Case 	
--	--

Here you have defined a new Analysis Case, ‘power’, and indicated that you will use ESATAN-TMS Thermal as your thermal solver. You have also associated the Polar Radiative Case and the Unit1_power and Unit2_power boundary conditions with this Analysis Case.

When you create a new Analysis Case or redefine an existing case, the case is automatically opened so that it can be used as soon as you have completed (re)defining it. When an Analysis Case is open, the status bar located at the bottom of your Workbench window displays the name of the Analysis Case adjacent to a blue Analysis Case folder. The first Radiative Case associated with the Analysis Case will also be opened.

2.5.1.2 Solution Control

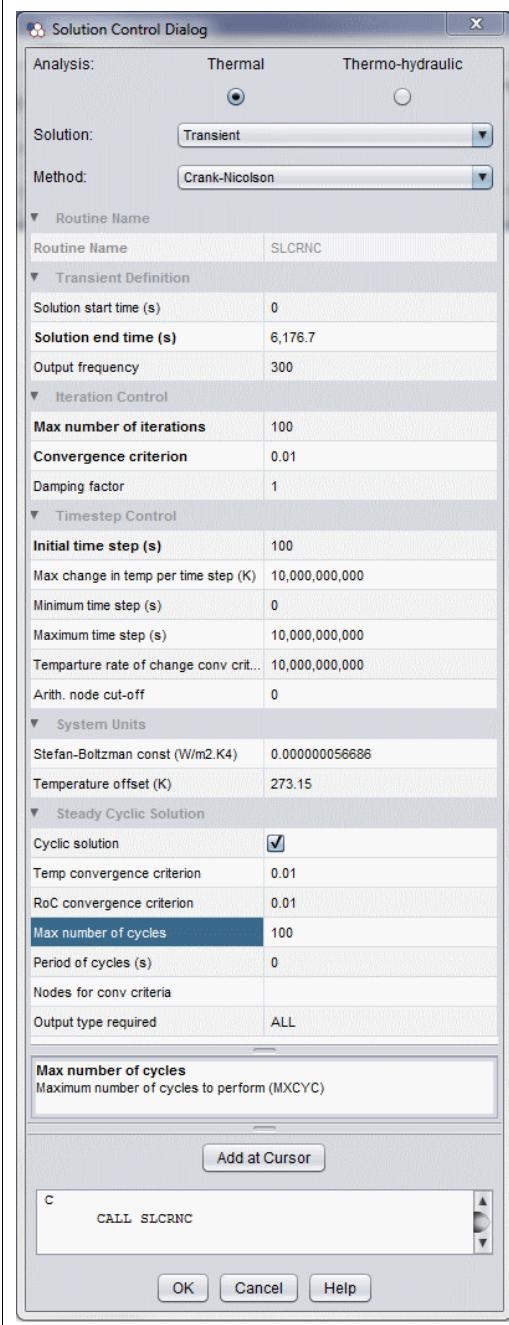
The next step is to define the solution control. From the Define Analysis Case dialog, you can access the Solution Control Dialog to select as many consecutive solution calls as you require, plus all the control constants needed. In our thermal analysis we will first perform a steady-state solution based on the averaged orbital conditions, followed by a transient analysis over a number of orbits to achieve a cyclically repeating result (that is, the temperatures repeat within a given margin between one orbit and the next). For your model you can do this as shown below:

- Select  next to the Solution Control field in the c to open the Solution Control Dialog..
- Ensure the following are set correctly:

Solution:	= Steady State
Solution Routine:	= SOLVFM
- Click Add at Cursor
- Change:

Solution	= Transient
----------	-------------
- Ensure the following is set correctly:

Routine Name:	= SLCRNC
Solutions end time (s):	= 6176.7
Output frequency:	= 300
Initial time step (s):	= 100
- Select (tick) Cyclic solution
- Set Maximum number of cycles = 100
- Click Add at Cursor
- Click Apply in the Define Analysis Case dialog.



Firstly you chose Steady State as the Solution Type and SOLVFM as the Solution Routine (see the help menu in the ESATAN-TMS Thermal library routines for a complete description of SOLVFM). This is all that is required for the steady-state solution in this example as the default settings are sufficient for the remaining parameters.

Next you defined the Transient solution, choosing the solution routine SLCRNC. You set Solutions end time (TIMEND) which is set to the orbit period. The Output frequency (OUTINT) was set to give 300 outputs per orbit and Initial time step (DTIMEI) to 100.

Selecting the Cyclic solution (SOLCYC) option calls this meta-solver that will run over a maximum of 100 orbits (MXCYC = 100) to find the repeating condition.

Additionally you could run a transient analysis over a single orbit to find the maximum and minimum temperatures in orbit. To do this you would repeat the parameter settings for the transient case but un-check the Cyclic solution option before adding to the execution block.

2.5.1.3 Reviewing a Template File

You can review the resulting Template File as follows:

<ul style="list-style-type: none"> • In the model tree, expand Analysis Case > power • Right-click Template.tpl and select Edit/View (set preferred editor in File → Preferences if required) • Review file as required 	
---	--

For a more complex analysis you would modify the Template at this stage to include any additional data required. The analysis file can then be automatically recreated each time the Radiative Case is changed and re-run.

2.5.1.4 Reviewing an Analysis File

You can review your Analysis File (power.d) just as you did for your Template File, see Subsection 2.5.1.3: “Reviewing a Template File”. It is also advisable to save your model at this stage.

2.5.2 Running the Thermal Analysis

You are now ready to run the thermal analysis. You can do this by following the steps below:

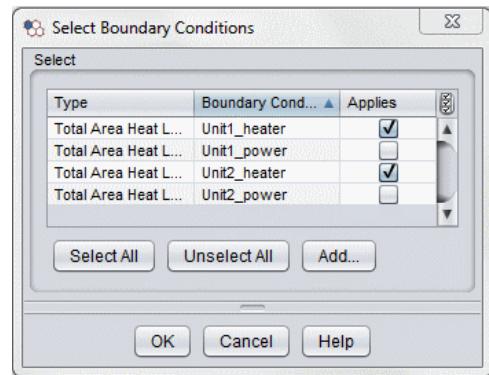
<ul style="list-style-type: none"> • In the Define Analysis Case dialog: • Select Run Analysis • Click Apply followed by Close 	
---	--

the Analysis Monitor will open with the results.

2.5.3 Running Additional Thermal Analyses

When ESATAN-TMS has completed the Thermal Analysis for the ‘power’ analysis case you can then create and run a new ‘heat’ analysis case by redefining the ‘power’ case as follows:

- Thermal → Analysis Case → (Re)define
- Analysis Case: = heat
- Solver: = ESATAN
- Analysis Case Type:- Single Radiative Case
- Radiative Case = Polar
- Select  next to Boundary Conditions
- Un-select Unit1_power and Unit2_power
- Select Unit1_heater and Unit2_heater
- Click OK to return to the Define Analysis Case dialog.
- Click Apply to define the Analysis Case followed by Close



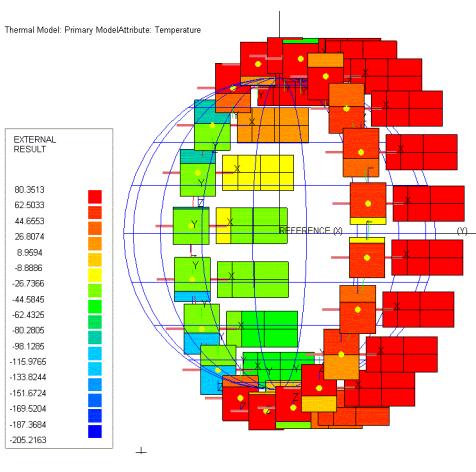
Here you have defined an additional Analysis Case and associated an alternative set of Boundary Conditions with this case. You can now repeat all the steps described in section 2.5.1.2 through to section 2.5.2 to complete and run your ‘heat’ Analysis Case.

When ESATAN-TMS Thermal has completed the thermal analyses for the ‘power’ and ‘heat’ Analysis Cases you can post-process your results using Workbench.

2.6 Post-processing the Thermal Analysis Results

2.6.1 Post-Process on geometry

To post-process your thermal analysis results you must first import them into ESATAN-TMS Workbench as shown below:

<ul style="list-style-type: none"> • In the Model Tree right click on the main model and select Show. • Now right click on the power Analysis Case and select Open • Expand the Result Files folder • Right click on the file Model_1_POWER.TMD2 (this is the transient result file) and select Post-Process... • Select (tick) Albedo Flux, Planet Flux and Solar Flux • Display As: <input checked="" type="checkbox"/> = Orbital Results • Model Scale: <input checked="" type="checkbox"/> = 120 • Select Results at the top of the dialog • Select Time Steps at the top of the dialog: Start time: 0 End time: 6176.70 • Click Display at the bottom of the dialog 	
---	--

Having imported your results data for the ‘power’ Analysis Case you can use the Display Data dialog to visualise your results. You can alter a range of parameters on this dialog to display and re-display different aspects of your results in the visualisation.

In the example above you have chosen to post-process the data for the thermal nodes (this is the default option) and to view the results around the orbit for all cycles. You set the model scale to 120 to make it easier to view in relation to the planet on the visualisation. You have then chosen to view the temperature results for your thermal model.

Study your results on the visualisation and make sure you are happy they are what you would expect for the example you have modelled and repeat the analysis for the results of the ‘heat’ Analysis Case if you wish.

2.6.2 Post-Process using charts

Charts can be created to visualise the thermal model and result data. For this example we will create the following charts,

1. Plot of the solar power input to the solar cells (Attribute Chart - Line Chart)
2. Plot of the radiator temperature (average, minimum and maximum) (Attribute Chart - Min-Max Chart)
3. Plot of the radiator heat balance (Attribute chart - Line Chart)
4. Comparison of the temperatures of the units for the 2 cases (Delta Chart)
5. Unit temperatures against specified limits (Limits Chart)

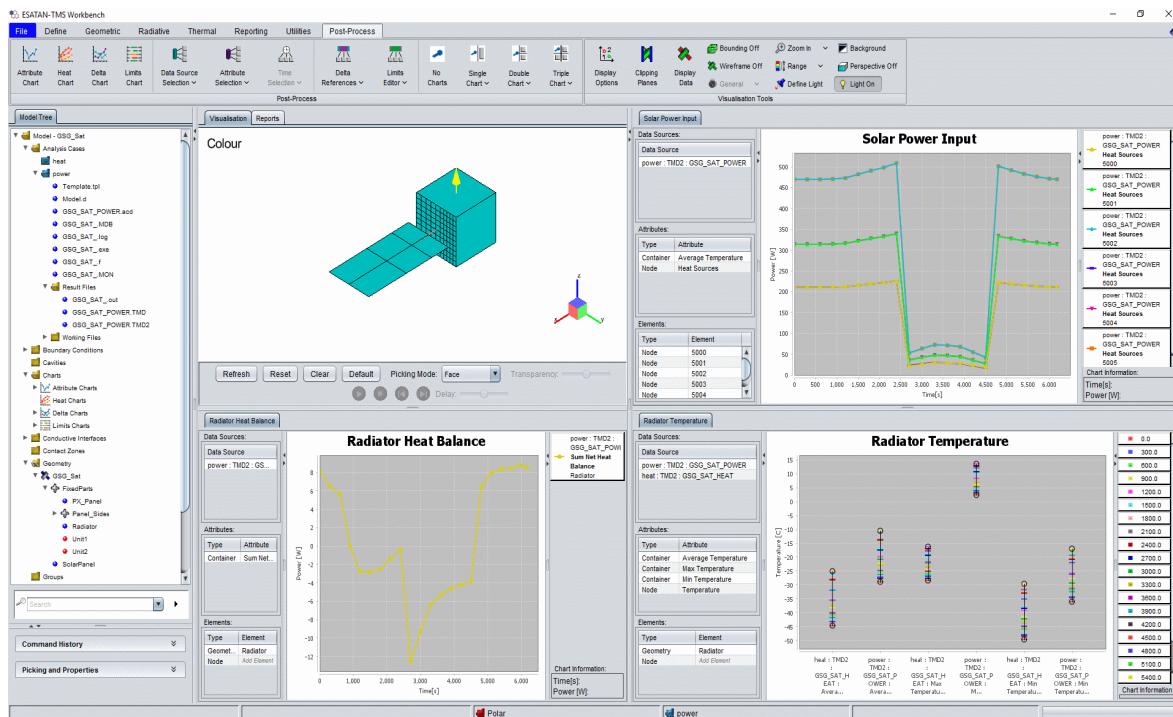


Figure 2-6 Triple Chart view

2.6.2.1 Solar power input to the solar cells (Attribute Chart - Line Chart)

To open the Analysis Case power:

- Select Thermal → power to open the Analysis Case.

This will expand the folder in the model tree.

- Expand the Result's file → right-click on MODEL_1_POWER.TMD2 → select New Chart → Attribute Chart.

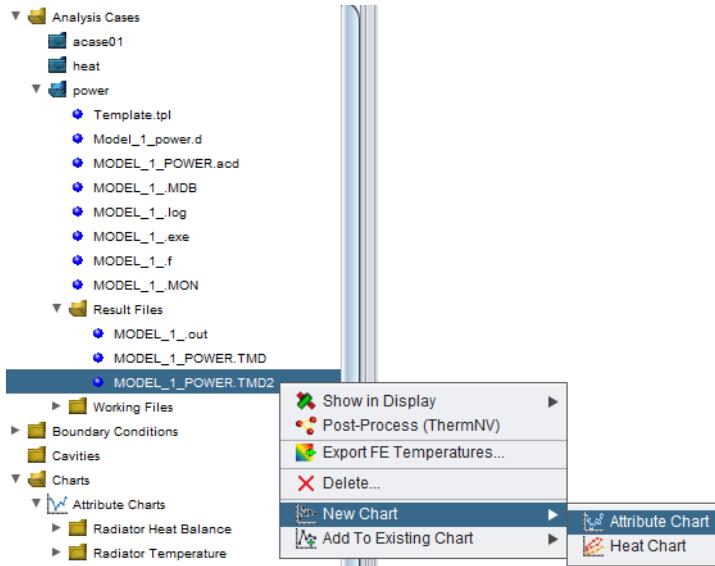


Figure 2-7 Add new attribute chart

This will create a single attribute chart to the right-hand side of the visualisation area.

Information: Workbench can be configured to display up to 3 chart areas using the buttons on the Post-Process tab Single Chart, Double Chart or Triple Chart. Within each chart window there can be multiple charts, each displayed on a separate tab. The charts can be easily moved from one window to another by dragging the chart from its tab

The Data Sources panel will automatically update showing the TMD file selected.

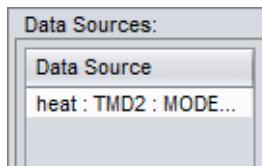


Figure 2-8 Data Sources panel

Edit the attributes displayed, by right-clicking in the Type field of the Attributes panel and select Edit; this will display the Attribute Selection dialog.

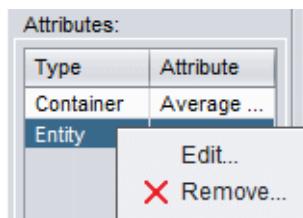


Figure 2-9 Edit Attributes

Remove the display of temperature for the nodes, by selecting Temperature in the right-hand list on the Node(s) tab (within the Attributes to Report list), and selecting the  arrow to remove the Temperature from the list. Select Heat Sources from the Attributes panel and click  to add it to the Attributes to Report panel as shown in Figure 2-10 .

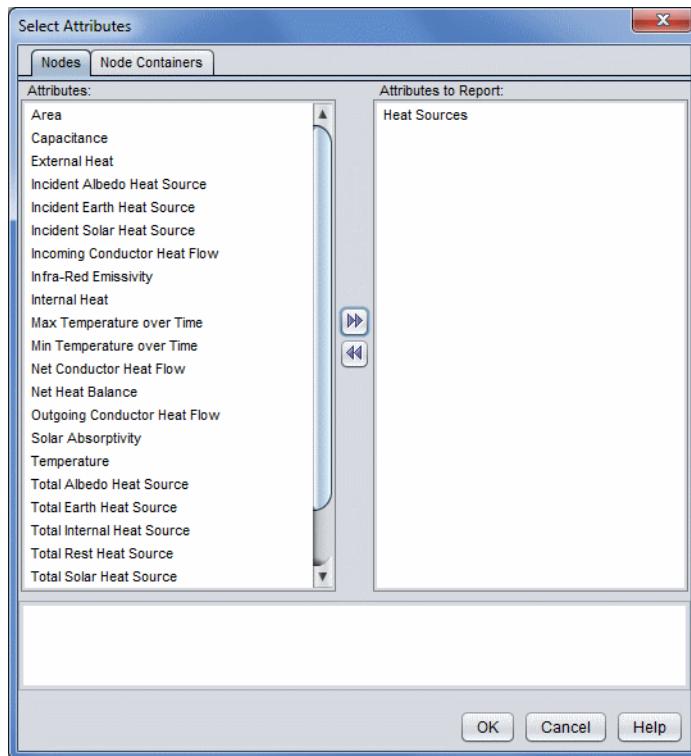


Figure 2-10 Attribute Selection Dialog

In the Elements panel in the chart information area, click in the Type field (last row in green), and select Surface. Then click in the Element field (last row in green with cell label Add Element), the Picking Mode in the visualisation will automatically change. Click on the SolarPanel surface 1 from within the visualisation (surface 1 is the side of the solar panel with the solar cells).

Elements:	
Type	Element
Surface	SolarPanel:surface1
Surface	Add Element

Figure 2-11 Elements Panel

Information: This will add the surface to the chart as a Container. A Container is an entity which contains multiple thermal nodes; a container can be FE Face, Surface, Geometry or a Group.

Expand the surface as the constituent thermal nodes, right-click on the Element field → select **Expand as Nodes**. This will display the individual nodes, rather than a single graph for the Container.

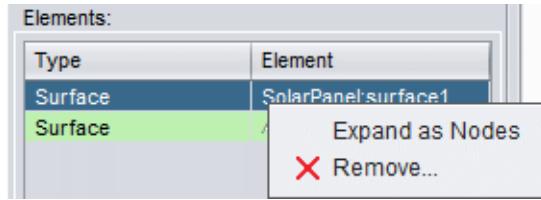


Figure 2-12 Expand as Nodes

The chart currently has a default name and title **Attribute Chart 1**. Rename the chart by right-clicking on the chart tab → select **Rename** → enter **Solar Power Input** → press **Enter** on the keyboard. By default, the title of the chart is set to the chart type name.

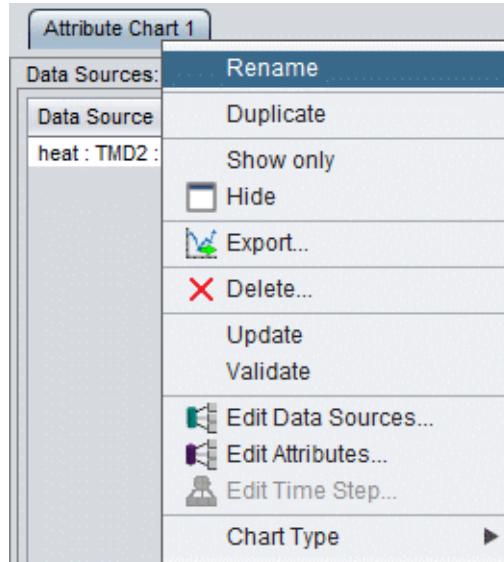


Figure 2-13 Rename Chart

Further customisation of the chart can be made by right-clicking on the chart and selecting **Properties**; this will display the **Chart Properties** dialog. The axis labels can be defined, along with other chart properties such as the line type, colour and markers.

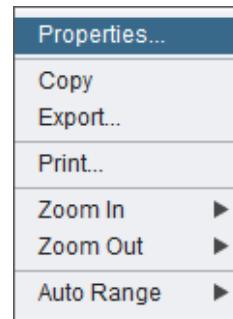


Figure 2-14 Chart menu

Change the label of the Y axis by selecting the Plot tab and the sub-tab Range Axis.

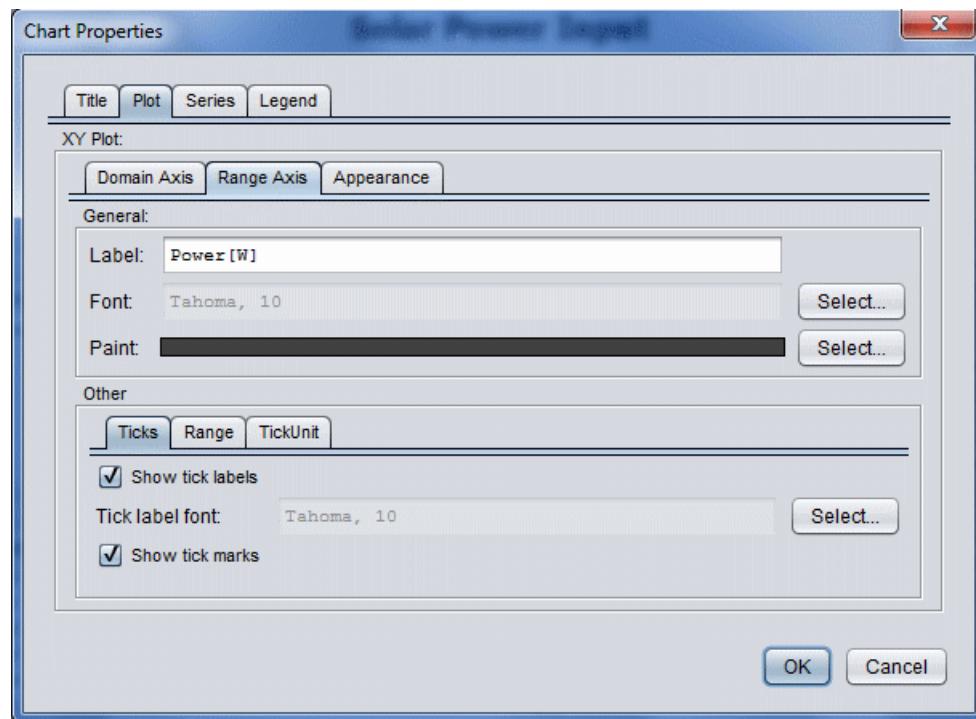


Figure 2-15 Chart Properties dialog

The final chart should look as follows.

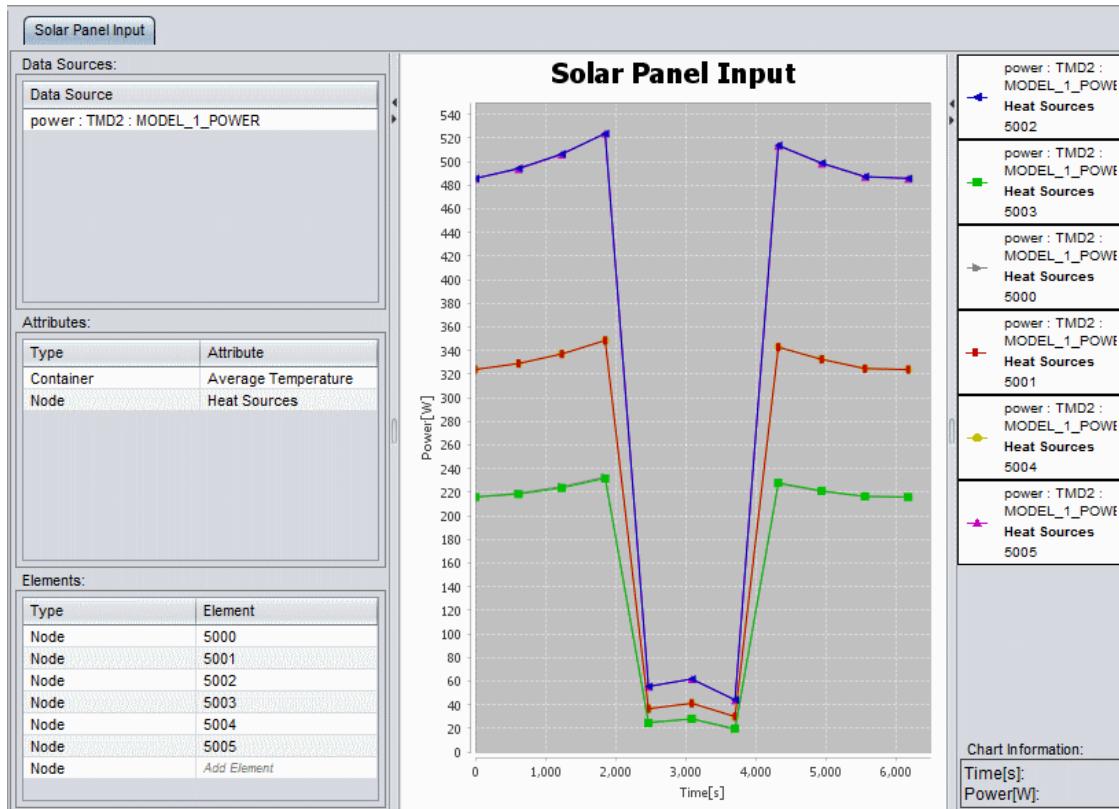


Figure 2-16 Solar Power Input chart

The period of eclipse can be clearly seen, where the solar energy received on the solar panel drops

The image can be exported to a PNG, CSV or XML file by right clicking on the chart tab → Export... The Export dialog will display, type in the File Name and select the required format in the Files of Type drop down list.

Or right-click on the chart → Export... to access the Export dialog.

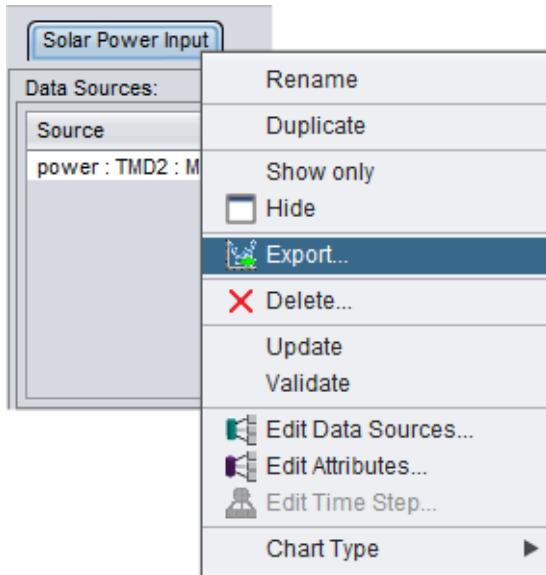


Figure 2-17 Export as Image menu

Save the model; this will also save the chart data.

2.6.2.2 Radiator temperature (Attribute Chart - Min-Max Chart)

As in the previous example (section 2.6.2.1), open the Analysis Case power and create a new attribute chart for the results file MODEL_1_POWER.TMD2.

Set the attributes to display by right-clicking in the Attributes field and selecting Edit. This time a Container is being displayed, so switch to the Node Containers tab → select Max Temperature [Node Based] and Min Temperature [Node Based] from the left-hand column (you can select multiple items in the list by holding the Ctrl button down) → add to the Attributes to Report list using the arrow.

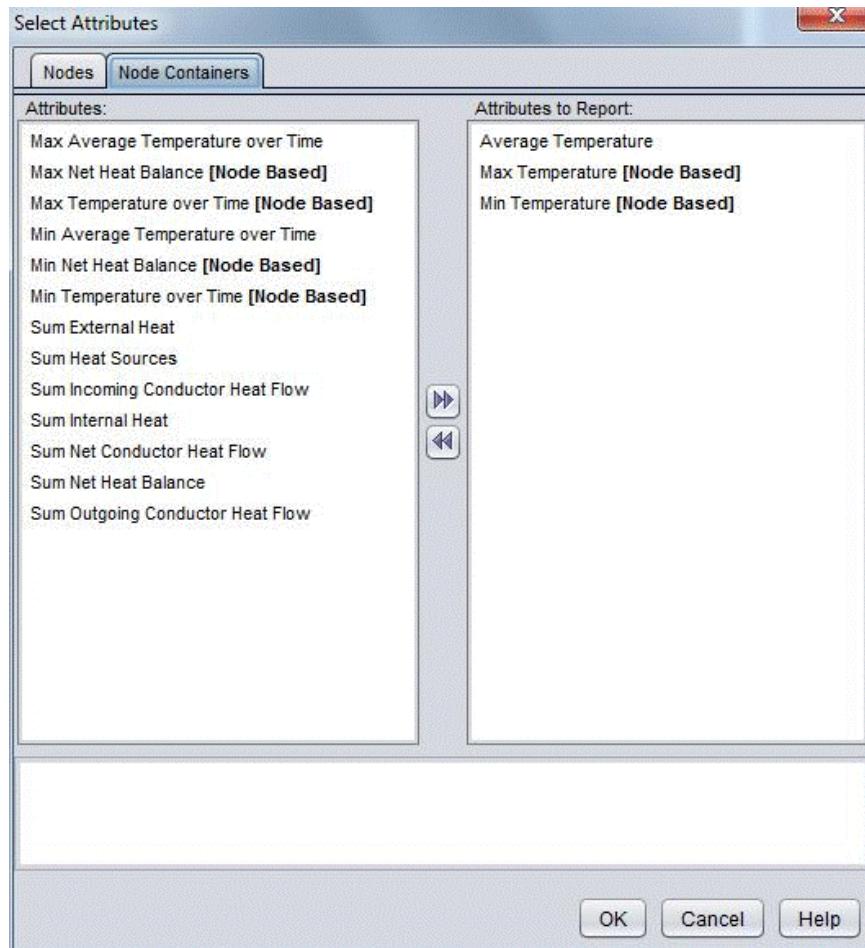


Figure 2-18 Node Containers

Add the geometry Radiator to the chart by setting the Type field to Geometry in the Elements panel, click on the Add Element field next to the Geometry selection just made and select Radiator from the drop-down list.

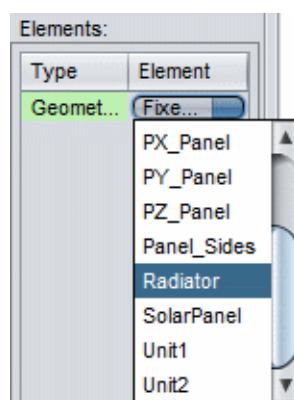


Figure 2-19 Geometry selection

Set the chart type to be a Min-Max Chart by selecting Chart Type → Min Max from the chart menu, as shown below.

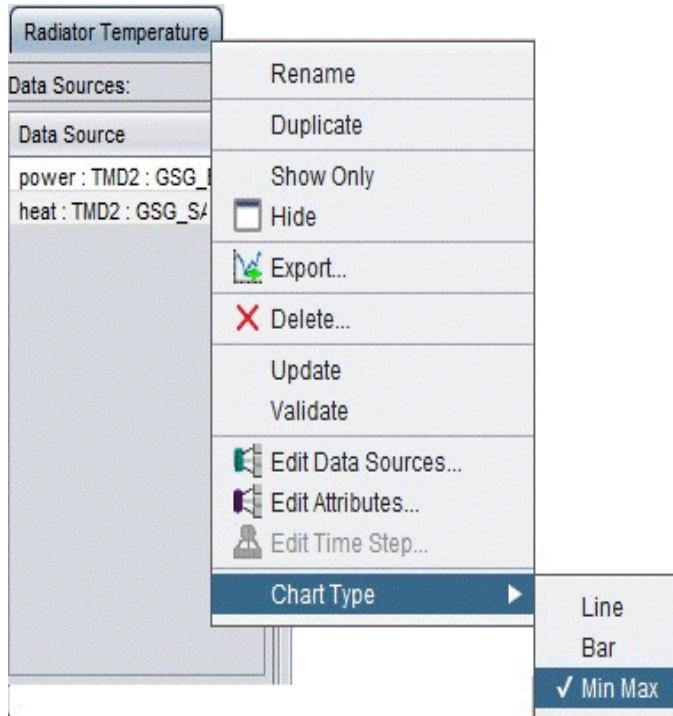


Figure 2-20 Min-Max Chart

As in the previous example, the chart name, title and axis can be customised, set the following properties.

- Set the chart name to Radiator Temperature
- Set the chart Y axis (Range) to Temperature[C]

Additional data sources can be added to the chart, therefore for this example we want to compare the results for the heat case on the same chart. To add this case to the chart, simply open the Analysis Case Heat in the model tree and right-click on the results file MODEL_1_HEAT.TMD2 and select Add to Existing Chart → Radiator Temperature. The chart will be updated with the new results set.

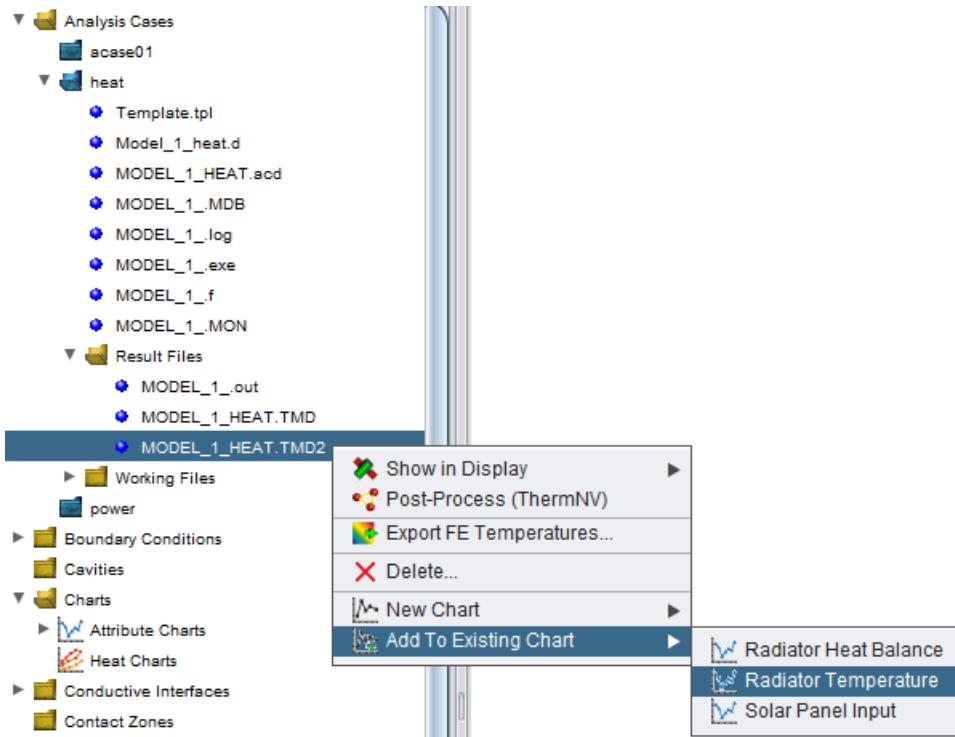


Figure 2-21 Add TMD file to existing chart

Save the model.

The final chart should look like the following diagram.

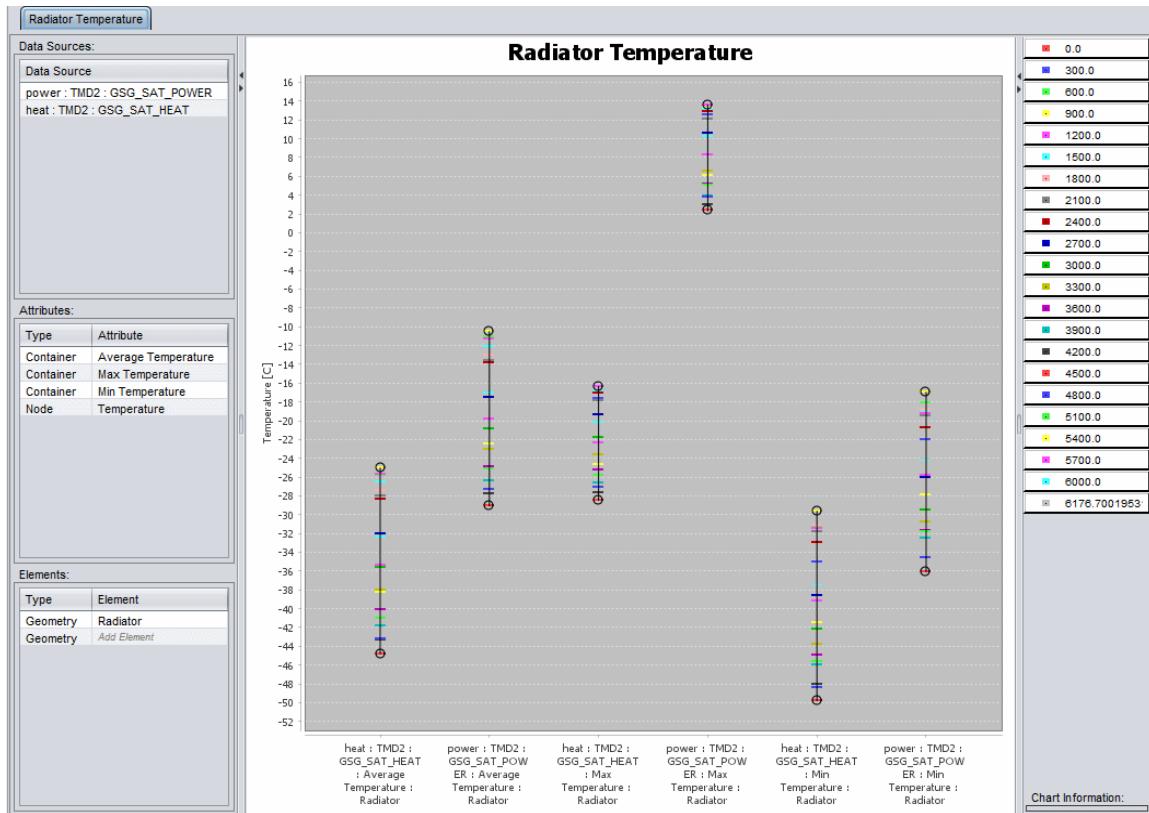


Figure 2-22 Radiator Temperature chart

2.6.2.3 Radiator heat balance (Attribute Chart - Line Chart)

Create a new Attribute chart selecting the data source Analysis Case power, result file MODEL_1_POWER.TMD2.

Add the geometry Radiator to the chart by selecting Radiator in the model tree, right-click and select Add to Existing Chart → Attribute Chart 1.

Edit the attributes that are displayed by selecting the Attribute Selection icon from the Post-process ribbon bar and select Attribute Chart 1; this will display the Attribute Selection dialog. Set the attribute for a Container to be Sum Net Heat Balance.

Set the name of the chart to be Radiator Heat Balance and set the Y axis label to be Power[W].

Save the model.

The final chart should look like the following

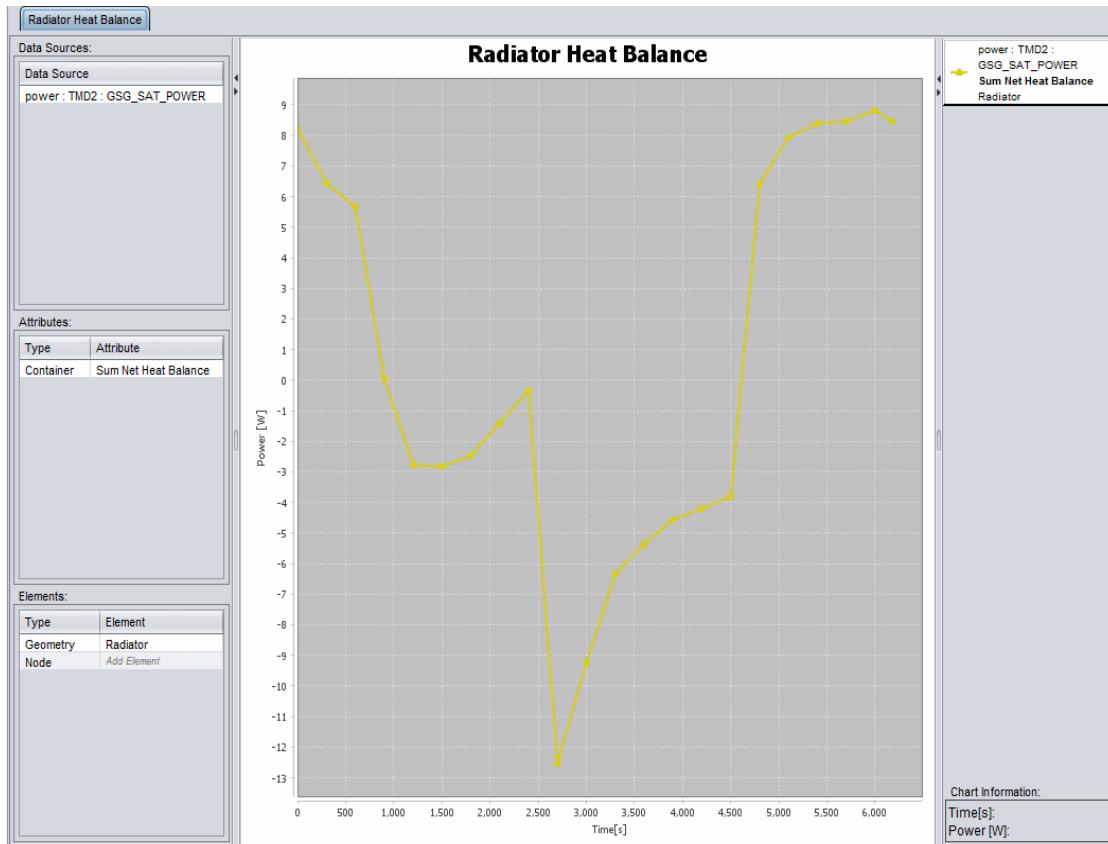


Figure 2-23 Radiator Heat Balance chart

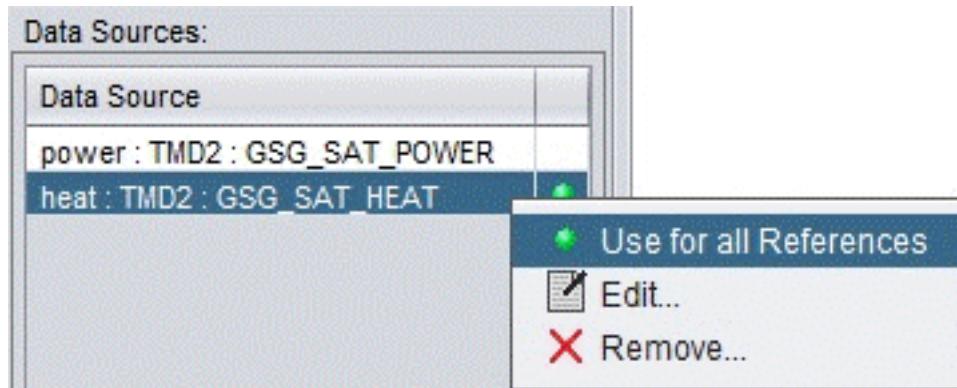
2.6.2.4 Unit Temperatures (Delta Chart)

The following example creates a Delta Chart to compare the temperature change of the two units with respect to the power and heat Analysis Cases.

Create a new Delta Chart and add the two transient cases (TMD2 data sets) for both the heat and power Analysis Cases.

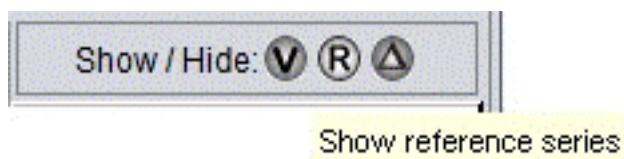
Add the geometry Unit1 and Unit2 to the chart by setting the element type to Geometry and selecting the two units from the drop-down (note that the units are added to the chart as Containers and therefore the Node Container attributes apply, even though the units are represented by a single thermal node).

The reference (the value that the element / attribute is compared against) can be a fixed value, a user-defined set of time / value pairs or the reference can be any one of the data sets on the chart. In this case we are going to compare the results against the heat Analysis Case. Select the heat case to be the reference by right-clicking on the heat data source and selecting “Use for all References”; see the following figure



Change the Chart name to “Unit Temperature” and the chart range axis label to “Temperature [C]”.

For each Element / Attribute pair the Delta Chart shows three series; the Reference (the value the element / attribute is compared against), the element / attribute value and the difference between the value and the reference. The user can choose to show or hide any of the three series. In our case the chart is going to show only the value and the delta series, by un-selecting the icon “R” just above the legend.



The final chart should look like the following

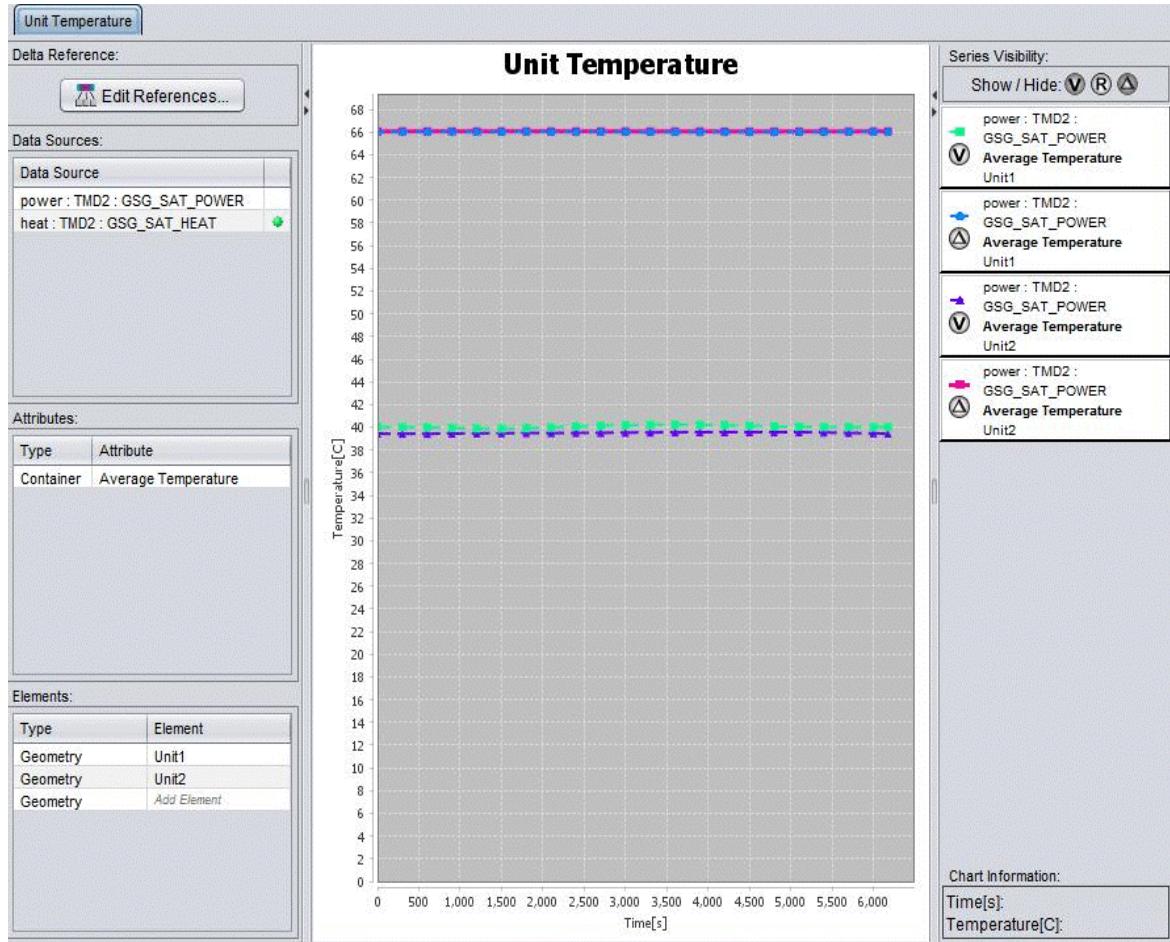


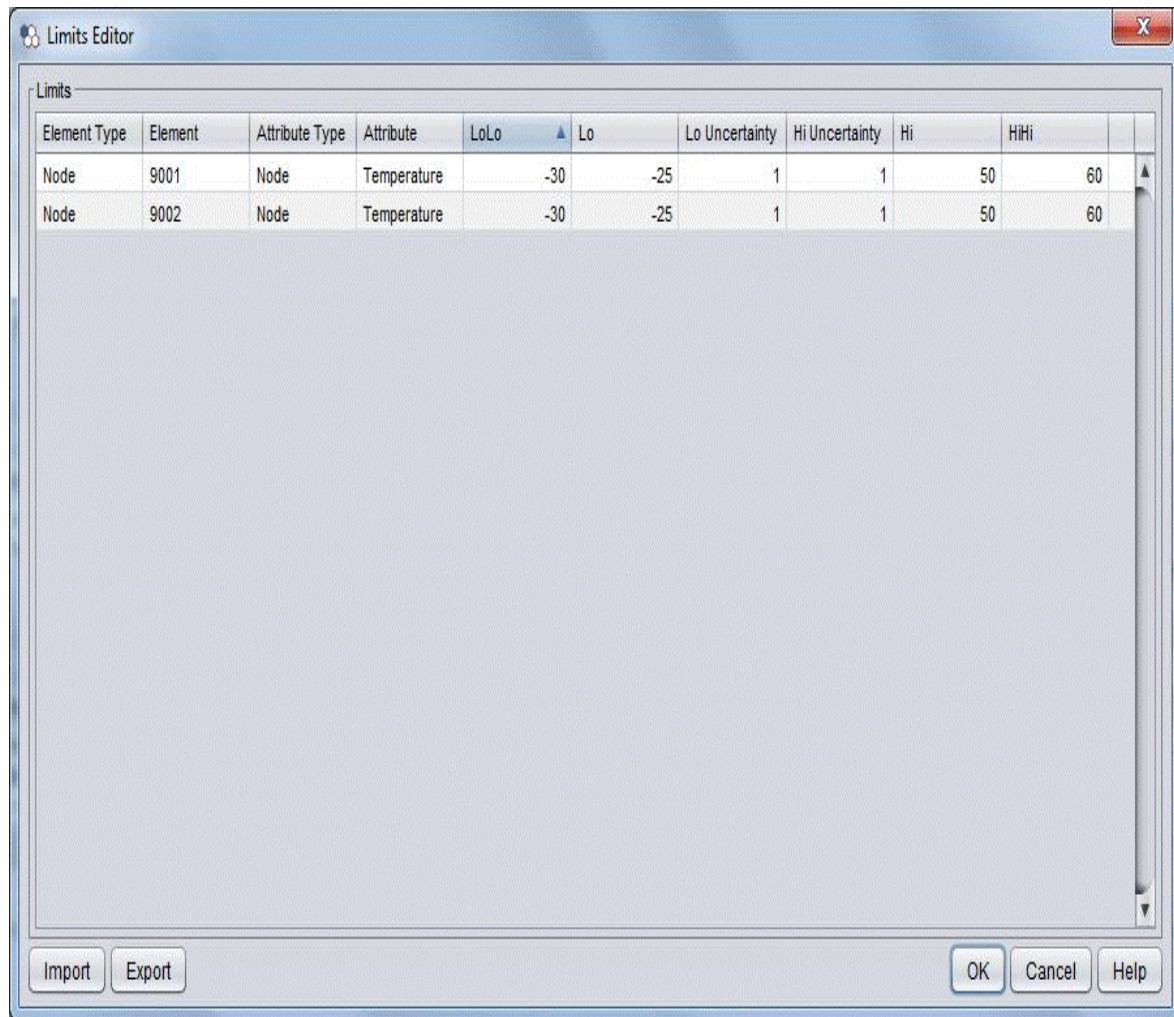
Figure 2-24 Final Unit Temperature Chart

2.6.2.5 Unit Temperatures (Limits Chart)

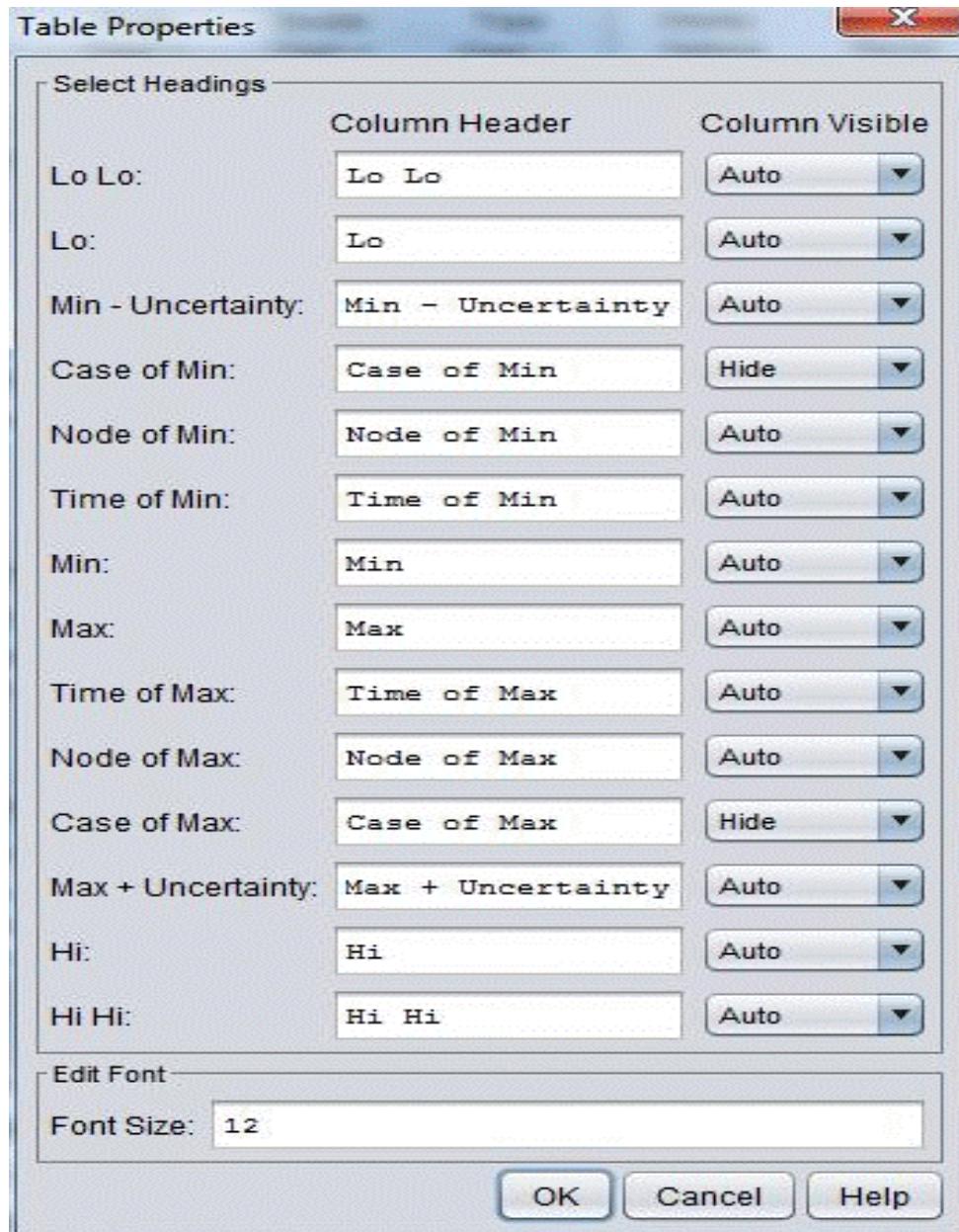
The same chart created in section 2.6.2.4 shall be created but this time using a Limits Chart to compare the temperatures of the units against predefined temperatures.

Create a new Limits Chart, add the two transient data sources (TMD2) for both the heat and power Analysis Cases and add the Geometry Unit1 and Unit2 to the chart. This will create a Limits Chart showing the table and chart view. The user has control over whether one of both of the views are displayed by selecting the icons next to the “Edit Limits...” button. For our case we will leave both of the views displayed (the default).

The next step is to define the limits for each of the element / attributes displayed; this is defined using the Limits Editor, selected from the Edit Limits button or Edit Limits from the chart menu. Enter the limits as defined within the following image.

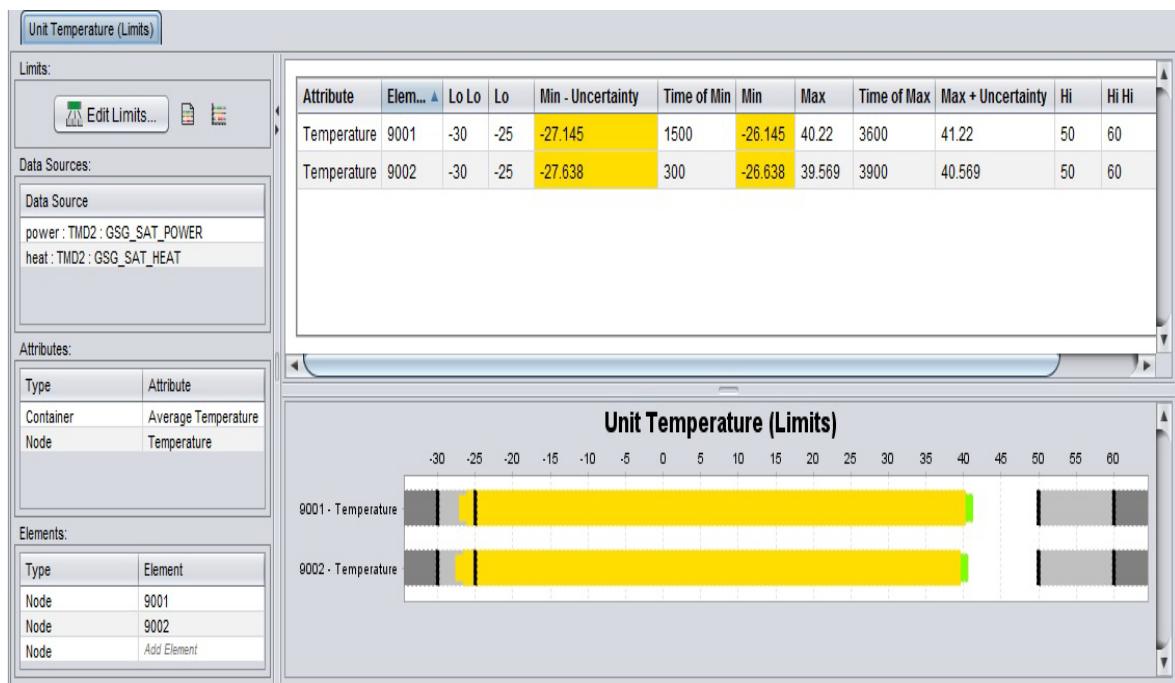


The Limits Table column titles and whether the columns are displayed in the table can be defined using the Table Properties dialog. The Table Properties dialog can be selected from the chart menu or right-clicking on the header of the Chart Table. In this case the names have been left as default but the columns "Case of Min" and "Case of Max" have been hidden.



Rename the chart "Unit Temperature (Limits)

The final chart should look like the following,



3. Model 2 - Complex (Solid) Satellite Model

3.1 Introduction

It is recommended you work through “Model 1 - Simple (Shell) Satellite Model” to familiarise yourself with the Graphical User Interface (GUI), shell definition, Radiative Case, Analysis Case and moving geometry before moving on to this intermediate guide for solid modelling capabilities.

This chapter describes the scenario you will be modelling with ESATAN-TMS Workbench as you work through the remaining sections in this chapter. You will define a spacecraft using different methods to “Model 1 - Simple (Shell) Satellite Model” and the orbit will be separated in to three orbits; circular, transfer and eccentric.

- Circular orbit is with time dependent steering
- Transfer orbit is with a vector to define the position number
- Eccentric orbit is with the sunlight and eclipse position

You will define this orbit after the assembly with its moving component pointing to the True Sun. You will then use Workbench to calculate the Radiative Exchange Factors (REFs) and the Solar, Albedo and Planetary Heat Fluxes (HFs) absorbed by the faces of the model as it travels around the orbit. Thereafter, you will display the temperature in Workbench and post-process the results in ThermNV to get the temperature evolution of the satellite around all orbits.

3.1.1 The Geometry

The spacecraft consists of a structure with two solar panels, two antennas, some boxes you will apply Boundary Conditions (BC) to, and a cold finger used to reduce the temperature of the antennas. See Figure 3-1.

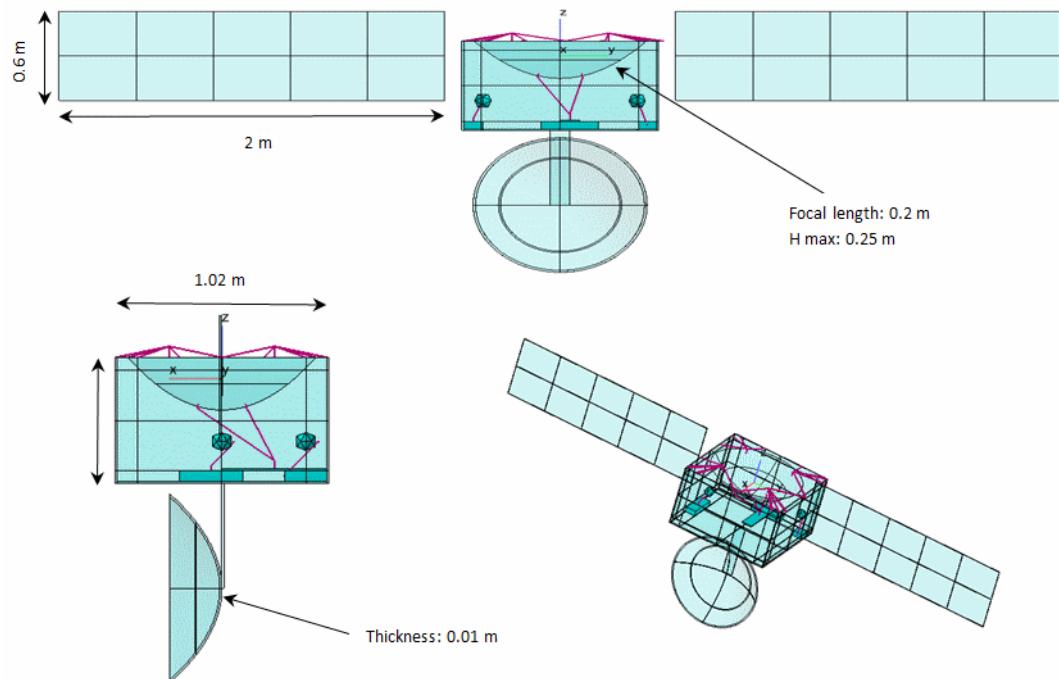


Figure 3-1 Model 2 spacecraft design

All parts (surfaces) of the satellite are active except the internal surfaces.

Two Non-Geometric Thermal Nodes (NGN) must be created in order to set the BC on the boxes and there are two ways of doing this. The Solar Cells are assumed to be copper with their optical properties on each side. As the Solar Panels (SP) have a large thickness, Kapton has been added to each SP corner as shown in Figure 3-2. The structure is composed of Aluminium with Black Paint properties on the outer side and White Paint properties on the inner side. The internal boxes are aluminium with Optical Properties (OP).

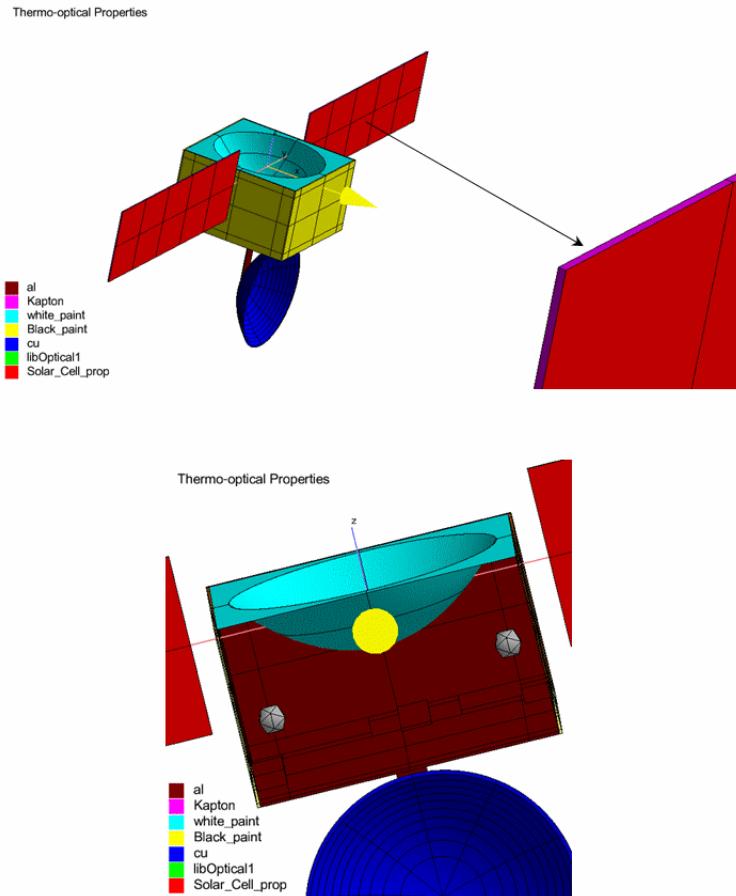


Figure 3-2 Model 2 optical coatings

Aluminium OP has been set for all components except for the upper plate on the inside of the satellite, and the antenna consists of white paint properties.

3.2 The Orbit

The spacecraft will follow a circular polar orbit around the planet Mars at the altitude of 900km, with an inclination of 97.8° and the right ascension of 10° . In the first orbit, you will try to get information from the Sun, get power and send information to the planet at the same time.

You will perform a Hohmann orbital transfer shown in Figure 3-3 which will be from the end of the first circular orbit to the beginning of the last orbit and define a vector at the position you want the results for the radiative calculations.

In the last orbit, the value for the beginning and the end of the eclipse and sunlight area will be retrieved.

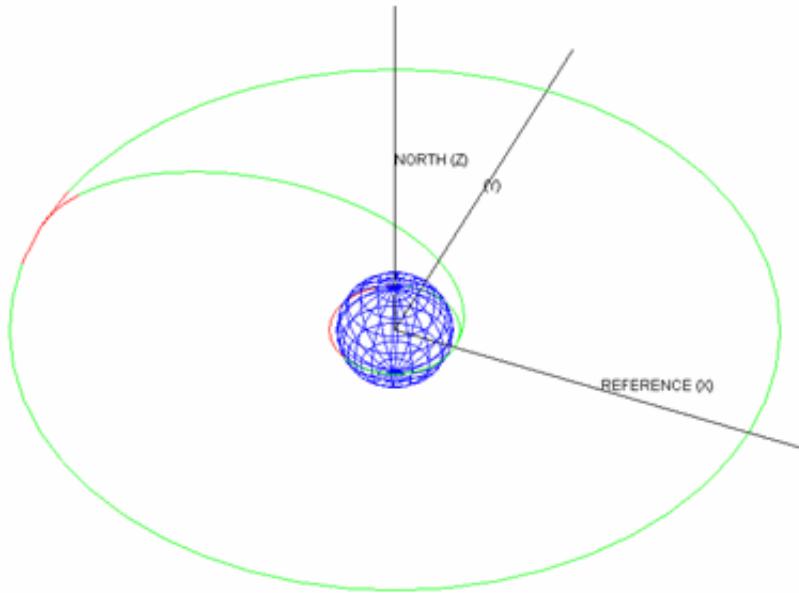


Figure 3-3 Hohmann orbit definition

Note that the Hohmann orbit is an elliptical orbit used to transfer between two circular orbits of different altitudes although the last orbit is eccentric.

It is assumed that you are aware of inclination, true anomaly, period, right ascension of the ascending node, apogee and perigee before starting this model.

3.3 Defining the Geometry

See Section 2.2: “Defining the Geometry” for more information.

3.3.1 Defining a New Model

See Subsection 2.2.1: “Defining a New Model” and name your model ‘*Model_2*’.

3.3.2 Defining Material Properties

You will begin by defining the bulk material using Define → Bulk Material or from the model tree by expanding the Materials and double-clicking on the Bulks folder. Define the bulk materials to Figure 3-4

<input type="button" value="Model Tree"/> <input type="button" value="Define Bulk Material"/>	
Bulk:	Silcell
Description:	<input type="text"/>
Type:	Isotropic
▼ Bulk	
Density (kg/m3)	8900.0
Specific Heat (J/kg.K)	900.0
Conductivity (W/m.K)	20.0

<input type="button" value="Model Tree"/> <input type="button" value="Define Bulk Material"/>	
Bulk:	Copper
Description:	<input type="text"/>
Type:	Isotropic
▼ Bulk	
Density (kg/m3)	4200.0
Specific Heat (J/kg.K)	200.0
Conductivity (W/m.K)	10.0

<input type="button" value="Model Tree"/> <input type="button" value="Define Bulk Material"/>	
Bulk:	Aluminuim
Description:	<input type="text"/>
Type:	Isotropic
▼ Bulk	
Density (kg/m3)	2700.0
Specific Heat (J/kg.K)	800.0
Conductivity (W/m.K)	320.0

Figure 3-4 Bulk materials required

When the model is defined, the bulk material can be checked using the Display Data icon in the ribbon bar and selecting **Bulks** as the Category and **Display**. you can reproduce this step in order to verify whether everything has been defined correctly.

Bulk Properties

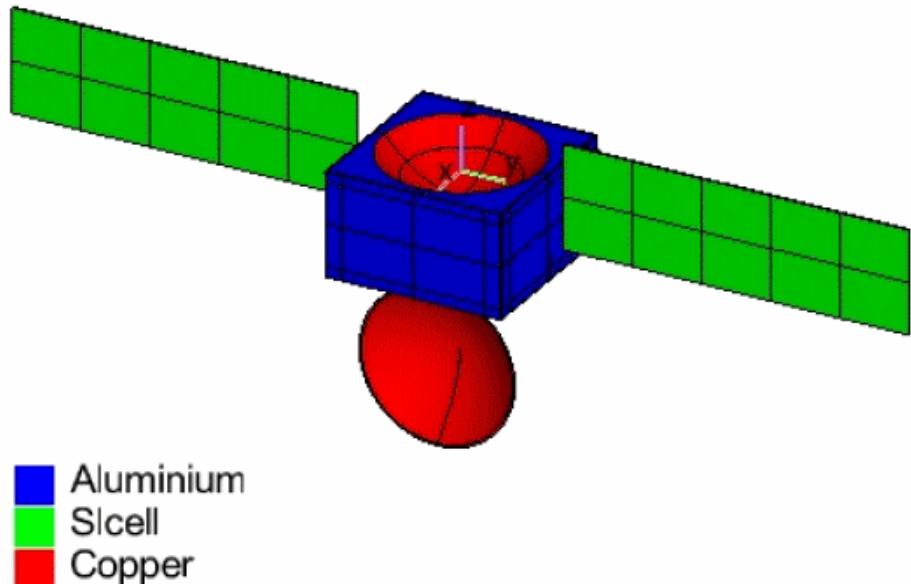


Figure 3-5 Bulk material on the model

3.3.3 Defining optical properties

You can report the optical coating and the material by right-clicking on the material and selecting **Report**.

The optical overlay is shown in the Subsection 3.3.4: “Geometry Construction” and the process to display the Optical Properties (OP) of your model is the same as bulk materials.

Define the thermo-optical properties for Black Paint, Kapton, Copper (cu), Aluminium (al), Spacecraft Properties (SC_Properties) and White Paint by selecting **Define** → **Optical Set** or by expanding the **Materials** folder in the model tree and double-clicking on **Optical Sets**. Define the thermo-optical properties as Figure 3-6.

Model Tree		Define Thermo-Optical									
Optical:	Black_Paint	Optical:	al								
Description:											
Environment Property Environment "default"											
Infrared <table border="1"> <tr><td>Emissivity</td><td>0.94</td></tr> <tr><td>Transmissivity</td><td>0</td></tr> <tr><td>Specular Reflectivity</td><td>0</td></tr> <tr><td>Diffuse Reflectivity</td><td>0.06</td></tr> </table>				Emissivity	0.94	Transmissivity	0	Specular Reflectivity	0	Diffuse Reflectivity	0.06
Emissivity	0.94										
Transmissivity	0										
Specular Reflectivity	0										
Diffuse Reflectivity	0.06										
Solar <table border="1"> <tr><td>Absorptivity</td><td>0.96</td></tr> <tr><td>Transmissivity</td><td>0</td></tr> <tr><td>Specular Reflectivity</td><td>0</td></tr> <tr><td>Diffuse Reflectivity</td><td>0.04</td></tr> </table>				Absorptivity	0.96	Transmissivity	0	Specular Reflectivity	0	Diffuse Reflectivity	0.04
Absorptivity	0.96										
Transmissivity	0										
Specular Reflectivity	0										
Diffuse Reflectivity	0.04										
Optical:	Kapton	Optical:	SC_Properties								
Description:											
Environment Property Environment "default"											
Infrared <table border="1"> <tr><td>Emissivity</td><td>0.18</td></tr> <tr><td>Transmissivity</td><td>0</td></tr> <tr><td>Specular Reflectivity</td><td>0</td></tr> <tr><td>Diffuse Reflectivity</td><td>0.82</td></tr> </table>				Emissivity	0.18	Transmissivity	0	Specular Reflectivity	0	Diffuse Reflectivity	0.82
Emissivity	0.18										
Transmissivity	0										
Specular Reflectivity	0										
Diffuse Reflectivity	0.82										
Solar <table border="1"> <tr><td>Absorptivity</td><td>0.12</td></tr> <tr><td>Transmissivity</td><td>0</td></tr> <tr><td>Specular Reflectivity</td><td>0</td></tr> <tr><td>Diffuse Reflectivity</td><td>0.88</td></tr> </table>				Absorptivity	0.12	Transmissivity	0	Specular Reflectivity	0	Diffuse Reflectivity	0.88
Absorptivity	0.12										
Transmissivity	0										
Specular Reflectivity	0										
Diffuse Reflectivity	0.88										
Optical:	cu	Optical:	White_Paint								
Description:											
Environment Property Environment "default"											
Infrared <table border="1"> <tr><td>Emissivity</td><td>0.7</td></tr> <tr><td>Transmissivity</td><td>0</td></tr> <tr><td>Specular Reflectivity</td><td>0</td></tr> <tr><td>Diffuse Reflectivity</td><td>0.3</td></tr> </table>				Emissivity	0.7	Transmissivity	0	Specular Reflectivity	0	Diffuse Reflectivity	0.3
Emissivity	0.7										
Transmissivity	0										
Specular Reflectivity	0										
Diffuse Reflectivity	0.3										
Solar <table border="1"> <tr><td>Absorptivity</td><td>0.5</td></tr> <tr><td>Transmissivity</td><td>0</td></tr> <tr><td>Specular Reflectivity</td><td>0</td></tr> <tr><td>Diffuse Reflectivity</td><td>0.5</td></tr> </table>				Absorptivity	0.5	Transmissivity	0	Specular Reflectivity	0	Diffuse Reflectivity	0.5
Absorptivity	0.5										
Transmissivity	0										
Specular Reflectivity	0										
Diffuse Reflectivity	0.5										

Figure 3-6 Optical properties material required

Refer to Figure 3-2 to see the colour of the optical coatings.

3.3.4 Geometry Construction

There are several ways the geometry definition can be complete;

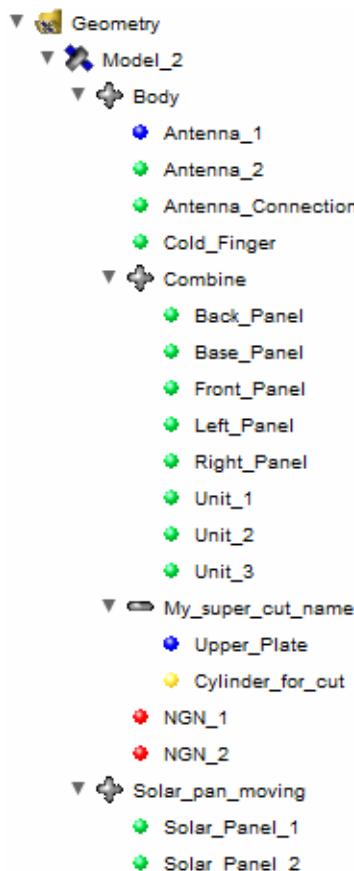


Figure 3-7 Model tree for the entire Model

This model is composed of three shells, one used for a cut, two Non-Geometric Thermal Nodes (NGN) used for the Boundary Conditions and thirteen solids for the structure modelling.

An assembly is required, therefore you will create two combinations with one fixed part and one moving part. See Figure 3-8 below and Section 2.2: “Defining the Geometry”.



Figure 3-8 Model tree combined

You will create a combination named Body which is the reference component and the Solar_pan_moving combination which is the moving component.

The procedure for this model will be to define all the geometry including NGN, to then combine all of them into a model including the cut. The next step will be to combine all the geometry as a Body and Solar_pan_moving combination.

Please check that you have defined all the material properties and optical properties before moving on to defining the geometry definition as follows.

Define a Real Vector as by selecting Define → Real Vector....and entering the details in Figure 3-9

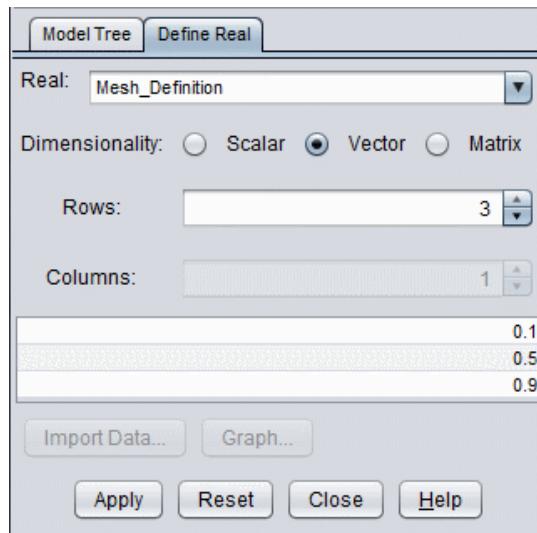


Figure 3-9 Define Vector

Now define the body definition by selecting Define → Geometry and entering the definitions in Figure 3-10 and Figure 3-11.

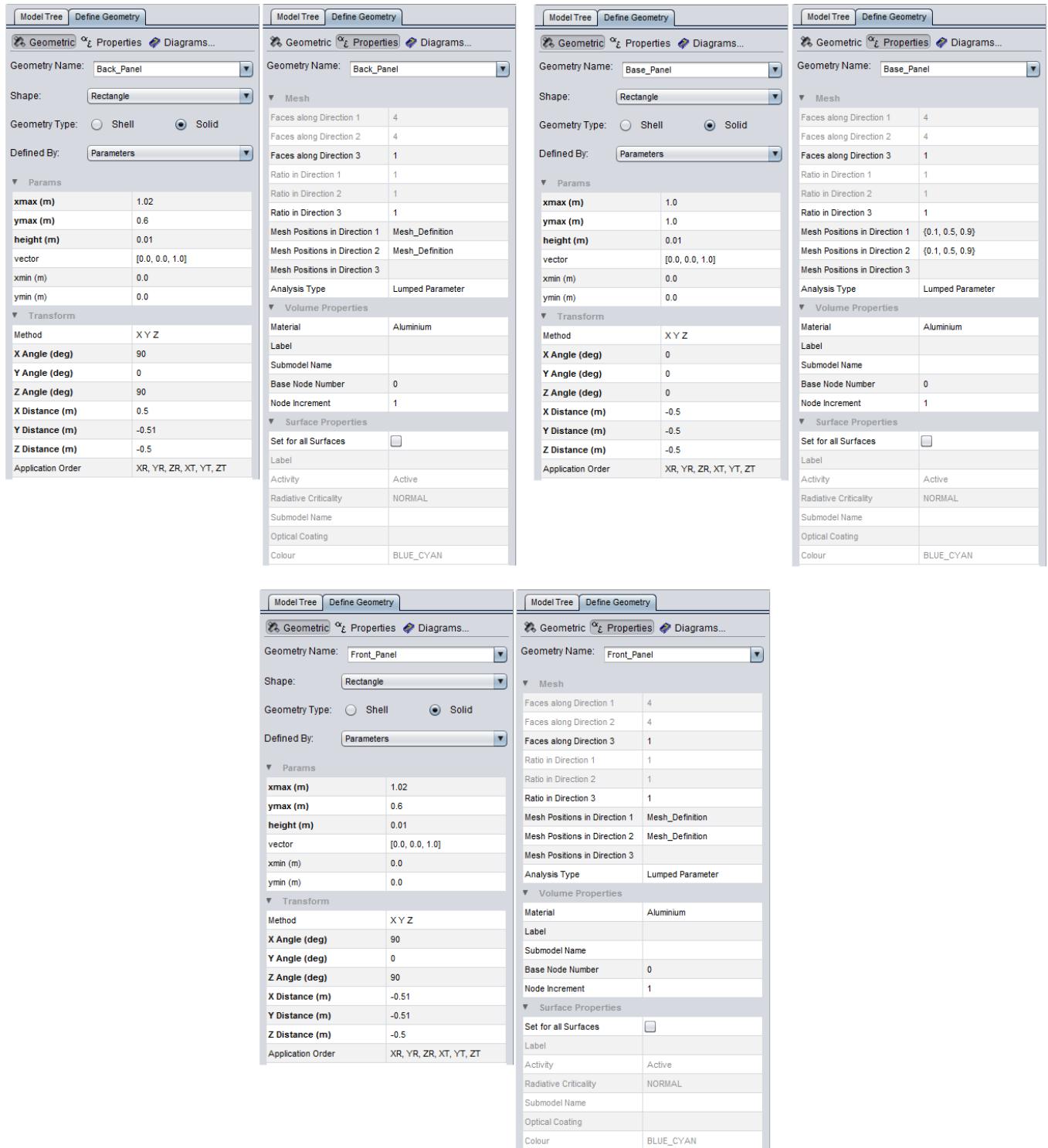


Figure 3-10 Body Definition

Model Tree **Define Geometry**

Geometric Properties Diagrams...

Geometry Name: Right_Panel

Shape: Rectangle

Geometry Type: Shell Solid

Defined By: Parameters

Params

xmax (m)	1.0
ymax (m)	0.6
height (m)	0.01
vector	[0.0, 0.0, 1.0]
xmin (m)	0.0
ymin (m)	0.0

Transform

Method	X Y Z
X Angle (deg)	90
Y Angle (deg)	0
Z Angle (deg)	0
X Distance (m)	-0.5
Y Distance (m)	0.51
Z Distance (m)	-0.5

Application Order: XR, YR, ZR, XT, YT, ZT

Model Tree **Define Geometry**

Geometric Properties Diagrams...

Geometry Name: Right_Panel

Mesh

Faces along Direction 1	4
Faces along Direction 2	4
Faces along Direction 3	1
Ratio in Direction 1	1
Ratio in Direction 2	1
Ratio in Direction 3	1
Mesh Positions in Direction 1	Mesh_Definition
Mesh Positions in Direction 2	{0.1, 0.5, 0.9}
Mesh Positions in Direction 3	
Analysis Type	Lumped Parameter

Volume Properties

Material	Aluminum
Label	
Submodel Name	
Base Node Number	0
Node Increment	1

Surface Properties

Set for all Surfaces	<input type="checkbox"/>
Label	
Activity	Active
Radiative Criticality	NORMAL
Submodel Name	
Optical Coating	
Colour	BLUE_CYAN

Model Tree **Define Geometry**

Geometric Properties Diagrams...

Geometry Name: Left_Panel

Shape: Rectangle

Geometry Type: Shell Solid

Defined By: Parameters

Params

xmax (m)	1.0
ymax (m)	0.6
height (m)	0.01
vector	[0.0, 0.0, 1.0]
xmin (m)	0.0
ymin (m)	0.0

Transform

Method	X Y Z
X Angle (deg)	90
Y Angle (deg)	0
Z Angle (deg)	0
X Distance (m)	-0.5
Y Distance (m)	-0.5
Z Distance (m)	-0.5

Application Order: XR, YR, ZR, XT, YT, ZT

Model Tree **Define Geometry**

Geometric Properties Diagrams...

Geometry Name: Left_Panel

Mesh

Faces along Direction 1	4
Faces along Direction 2	4
Faces along Direction 3	1
Ratio in Direction 1	1
Ratio in Direction 2	1
Ratio in Direction 3	1
Mesh Positions in Direction 1	Mesh_Definition
Mesh Positions in Direction 2	Mesh_Definition
Mesh Positions in Direction 3	
Analysis Type	Lumped Parameter

Volume Properties

Material	Aluminum
Label	
Submodel Name	
Base Node Number	0
Node Increment	1

Surface Properties

Set for all Surfaces	<input type="checkbox"/>
Label	
Activity	Active
Radiative Criticality	NORMAL
Submodel Name	
Optical Coating	
Colour	BLUE_CYAN

Figure 3-11 Body Definition (Continued)

As it is described in Figure 3-10 and Figure 3-11, the Real Vector defined for the mesh position is used for the mesh directions 1 and 2 of the geometry. However, the direct use of literal values is implemented, therefore the literal values can be directly entered in the field (0.1, 0.5, 0.9 instead of referencing a Real Vector). The software will recognize the literal values and convert to the format {0.1, 0.5, 0.9}.

The spacecraft structure is made of Aluminium and the OPs have been defined in Figure 3-6. To change the OP, you can select all surfaces with the same OP (hold the Ctrl key when selecting each surface). Set the Picking Mode to Surface, then right-click on a selected surface to define the OP using the Surface Properties... dialog. The OP can be verified using the Pre-Process Radiative Model dialog.

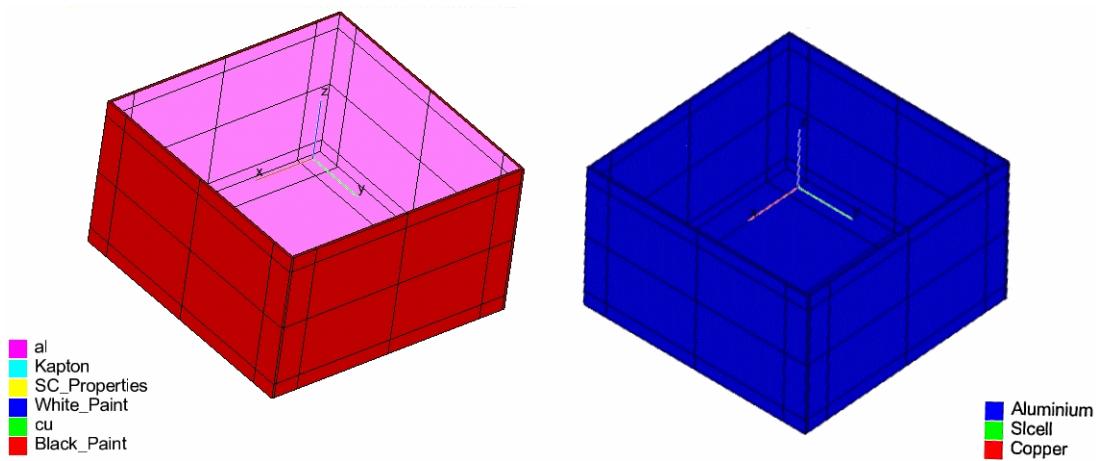


Figure 3-12 Body geometry with Ops and material displayed

Define the solar panels as per Figure 3-13 and Figure 3-14.

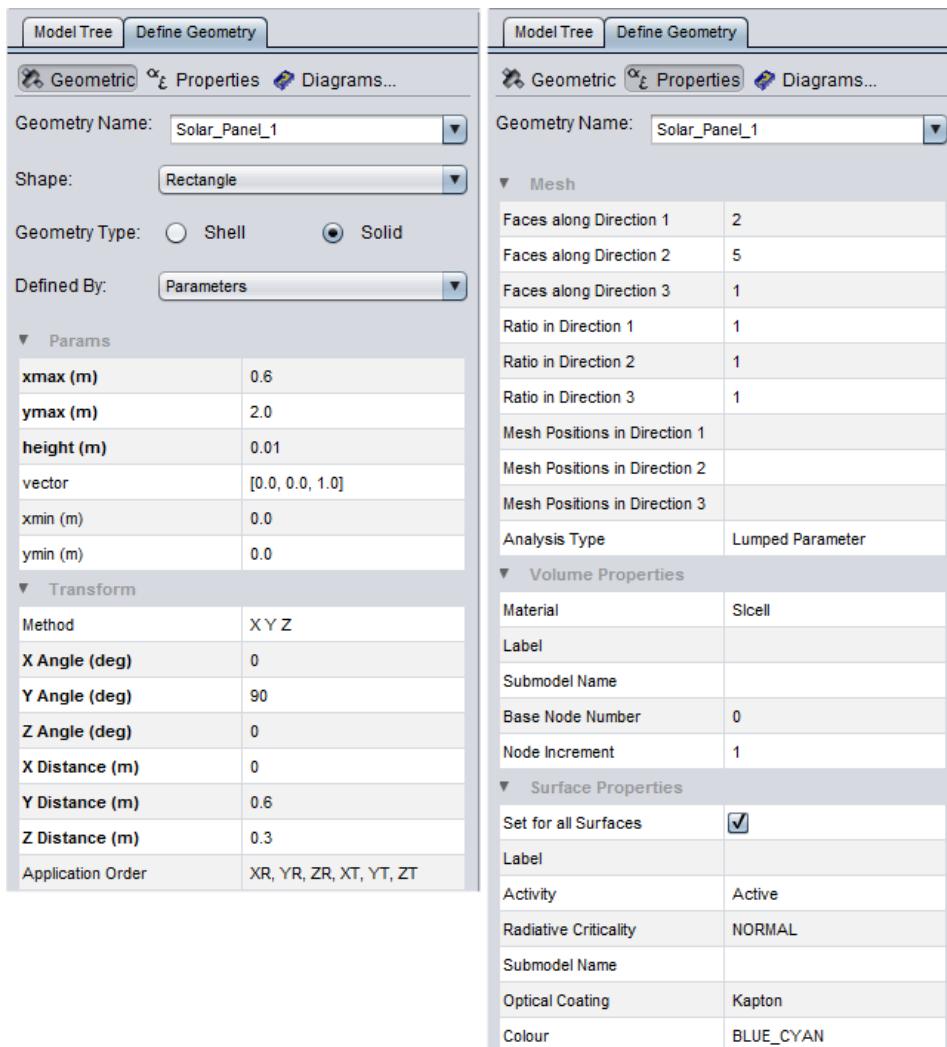


Figure 3-13 First solar panel definition

A regular mesh has been used for the Solar Panels (SP) definition, antennas, and the rest of the geometry

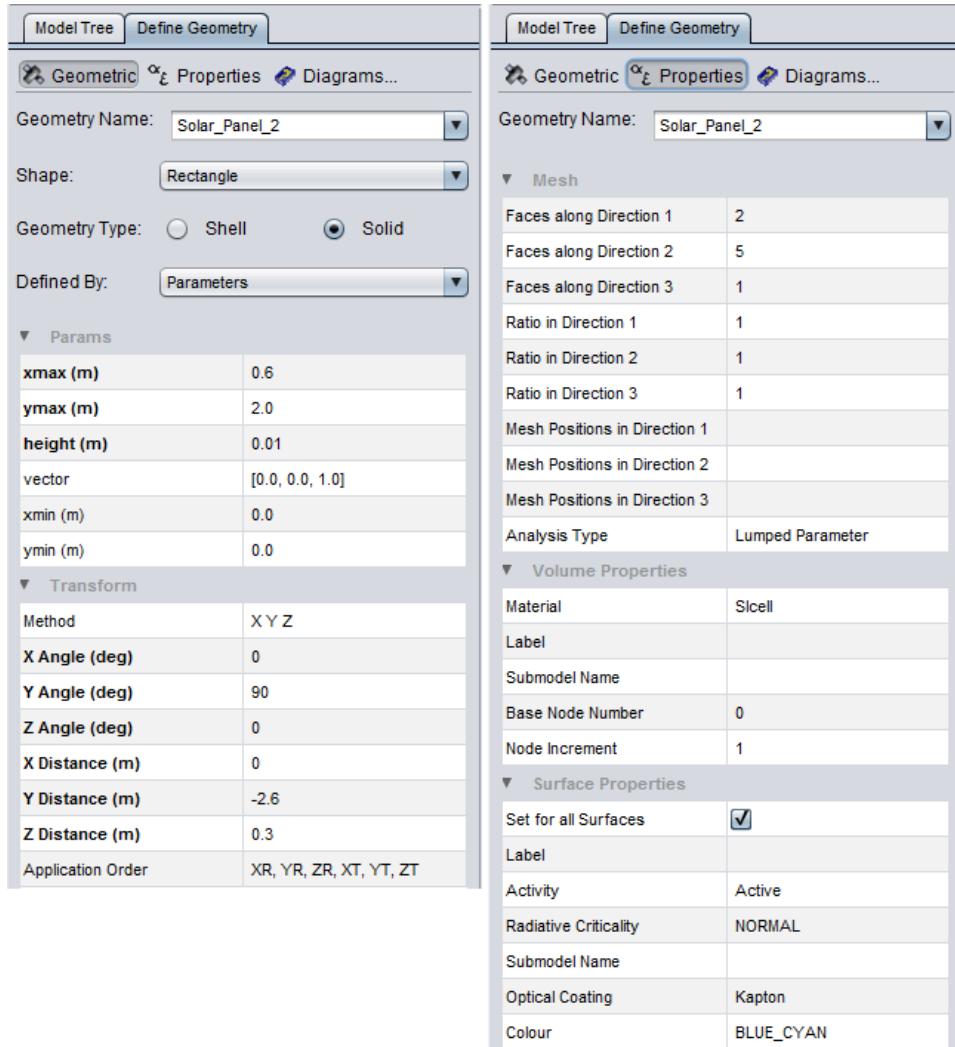


Figure 3-14 Second solar panel definition

It is then recommended that you use the **Display Data** dialog option to display and verify the bulk material and the optical properties.

Please refer to Subsection 3.1.1: “The Geometry” for the OPs and Figure 3-12 for more information and update the geometry definition as required. Select **Default** in the visualisation window to return the model to normal state.

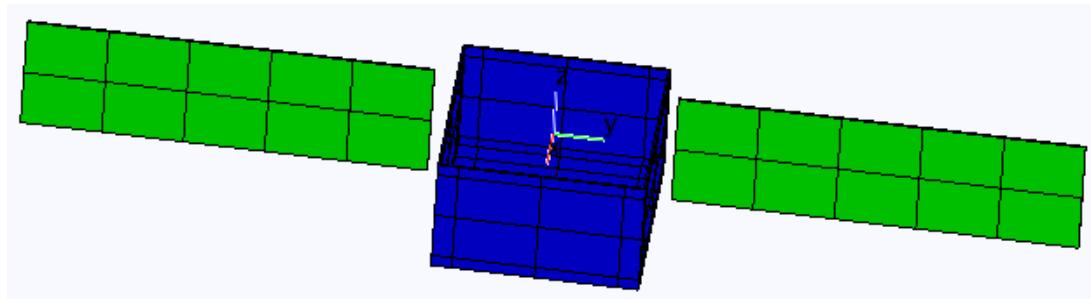


Figure 3-15 Model with SP

The process will be completely different for the definition of the geometry representing the units; the contents of Figure 3-17, Figure 3-18 and Figure 3-19 should be copied into the Geometry Command window. Press Enter after each copy/paste.

Select File → Preferences and select Show the Command Entry Panel box, then click OK to view the Command History panel under the model tree. Click History... to open the Command History pop up window for the next definition.

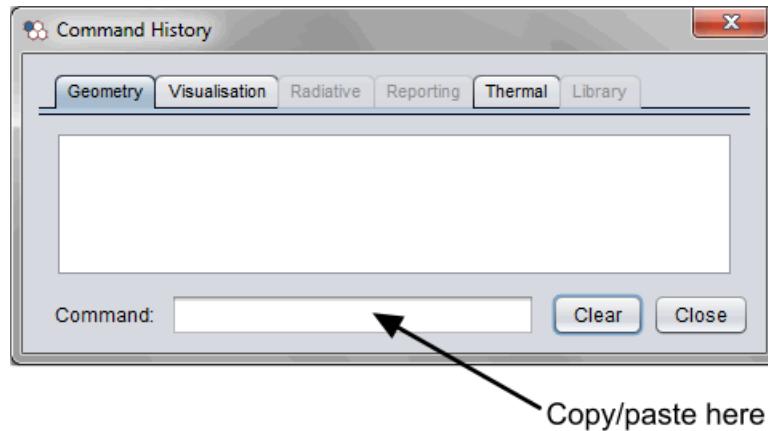


Figure 3-16 Geometry Command Window

```
GEOMETRY Unit_1;
Unit_1 = SOLID_RECTANGLE_POINTS(
point1 = [-0.10000000, -0.50000000, -0.49000000],
point2 = [0.20000000, -0.50000000, -0.49000000],
point4 = [-0.10000000, -0.40000000, -0.49000000],
point5 = [-0.10000000, -0.50000000, -0.44000000],
analysis_type = "Lumped Parameter",
bulk = Aluminium,
meshType1 = "regular",
nodes1 = 1,
ratio1 = 1.00000000,
meshType2= "regular",
nodes2 = 1,
ratio2 = 1.00000000,
meshType3 = "regular",
nodes3 = 1,
ratio3 = 1.00000000,
nbaseVolume = 0,
ndeltaVolume = 1,
optical = al,
activity= "Active",
criticality = "NORMAL",
colour = "CYAN");
```

Figure 3-17 Geometry unit 1

```
GEOMETRY Unit_2;
Unit_2 = SOLID_RECTANGLE_POINTS(
point1 = [-0.30000000, 0.50000000, -0.49000000],
point2 = [-0.40000000, 0.50000000, -0.49000000],
point4 = [-0.30000000, 0.40800000, -0.49000000],
point5 = [-0.30000000, 0.50000000, -0.44000000],
analysis_type = "Lumped Parameter",
bulk = Aluminium,
meshType1 = "regular",
nodes1 = 1,
ratio1 = 1.00000000,
meshType2= "regular",
nodes2 = 1,
ratio2 = 1.00000000,
meshType3 = "regular",
nodes3 =1,
ratio3 = 1.00000000,
nbaseVolume = 0,
ndeltaVolume = 1,
optical = al,
activity= "Active",
criticality = "NORMAL",
colour = "CYAN");
```

Figure 3-18 Geometry unit 3

```
GEOMETRY Unit_3;
Unit_3 = SOLID_RECTANGLE_POINTS(
point1 = [-0.10000000, -0.50000000, -0.49000000],
point2 = [0.20000000, -0.50000000, -0.49000000],
point4 = [-0.10000000, -0.40000000, -0.49000000],
point5 = [-0.10000000, -0.50000000, -0.44000000],
analysis_type = "Lumped Parameter",
bulk = Aluminium,
meshType1 = "regular",
nodes1 = 1,
ratio1 = 1.00000000,
meshType2 = "regular",
nodes2 = 1,
ratio2 = 1.00000000,
meshType3 = "regular",
nodes3 = 1,
ratio3 = 1.00000000,
nbaseVolume = 0,
ndeltaVolume = 1,
optical = al,
activity = "Active",
criticality = "CRITICAL",
colour = "CYAN");
Unit_3 = ROTATE (
object_name = Unit_3,
x_ang = 0.00000000,
y_ang = 0.00000000,
z_ang = 90.00000000,
clear = TRUE);
Unit_3 = TRANSLATE (
object_name = Unit_3,
x_dist = -0.90000000,
y_dist = 0.00000000,
z_dist = 0.00000000,
clear = FALSE);
```

Figure 3-19 Geometry unit 3

The code in Figure 3-17, Figure 3-18 and Figure 3-19 is purely the language generated by the software when you click **Apply** in the **Define Geometry** dialog. It can be used to create a geometry file “.erg” or define the entire geometry of the model.

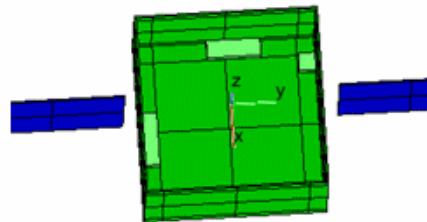


Figure 3-20 Internal Units

It is now time to define a simple cut of the model (a cylinder through a plate). It is recommended you design the plate and cylinder through this plate for the correct diameter of the hole. The plate (rectangle) will be defined by points, therefore it is preferable to display the points before defining the plate as shown in Figure 3-21 below.

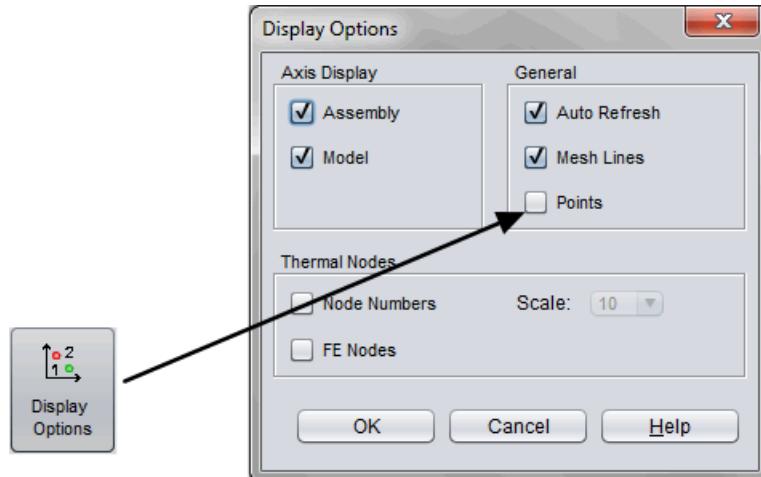


Figure 3-21 Display options

Select Define → Geometry and define a shell rectangle by points named `Upper_Plate`. Ensure to pick the internal points of each corner shown in Figure 3-22

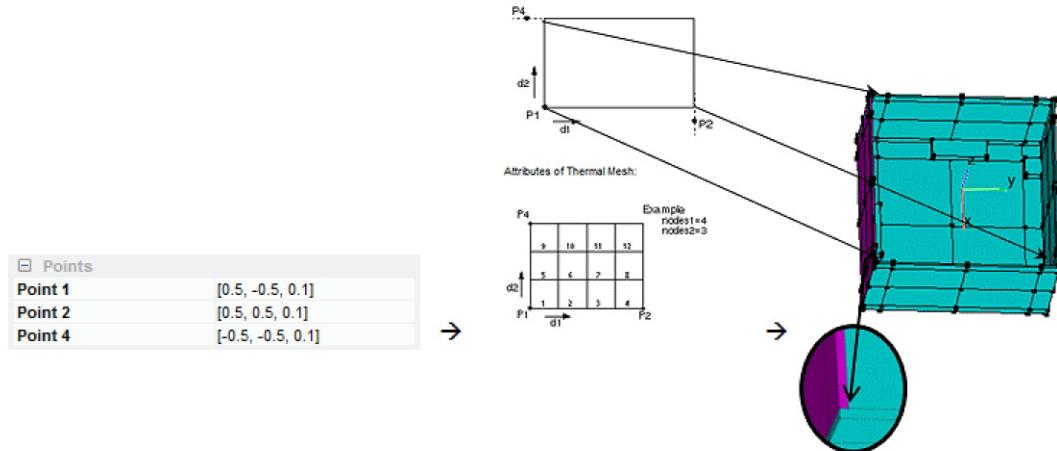


Figure 3-22 Rectangle definition by point

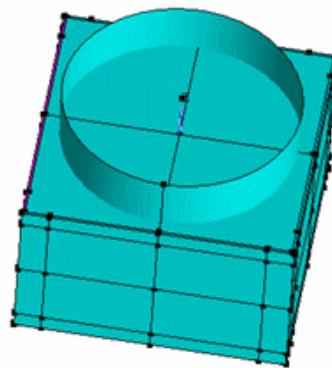


Figure 3-23 Cut preparation

The properties of the rectangle are a 2x2 mesh, set Faces along Direction 1 to 2 and Faces along Direction 2 to 2. Select DUAL for Bulk Materials, set Thickness to 1cm and define the material as set in Figure 3-5. Next define a cylinder by Parameters through this plate with a radius of 0.448 and hmax 0.5. Set the Faces along Direction 1 and 2 to 1. This cylinder must be centred to the upper plate and named Cylinder_for_cut.

3.3.4.1 Performing the cut

Before performing a cut, you must know which shell is the cutter and which is the shell to cut. Select **Geometric** → **Combine** or **Cut** which displays the **Define Combine or Cut** dialog. Define the cut name in the **Target Geometry** field, Add the shell to be cut (**Upper_Plate**) to the **Geometry** tab and Add the cutting tool (**Cylinder_for_cut**) to the **Cutting Tools** tab as shown in Figure 3-24 and click **OK**.

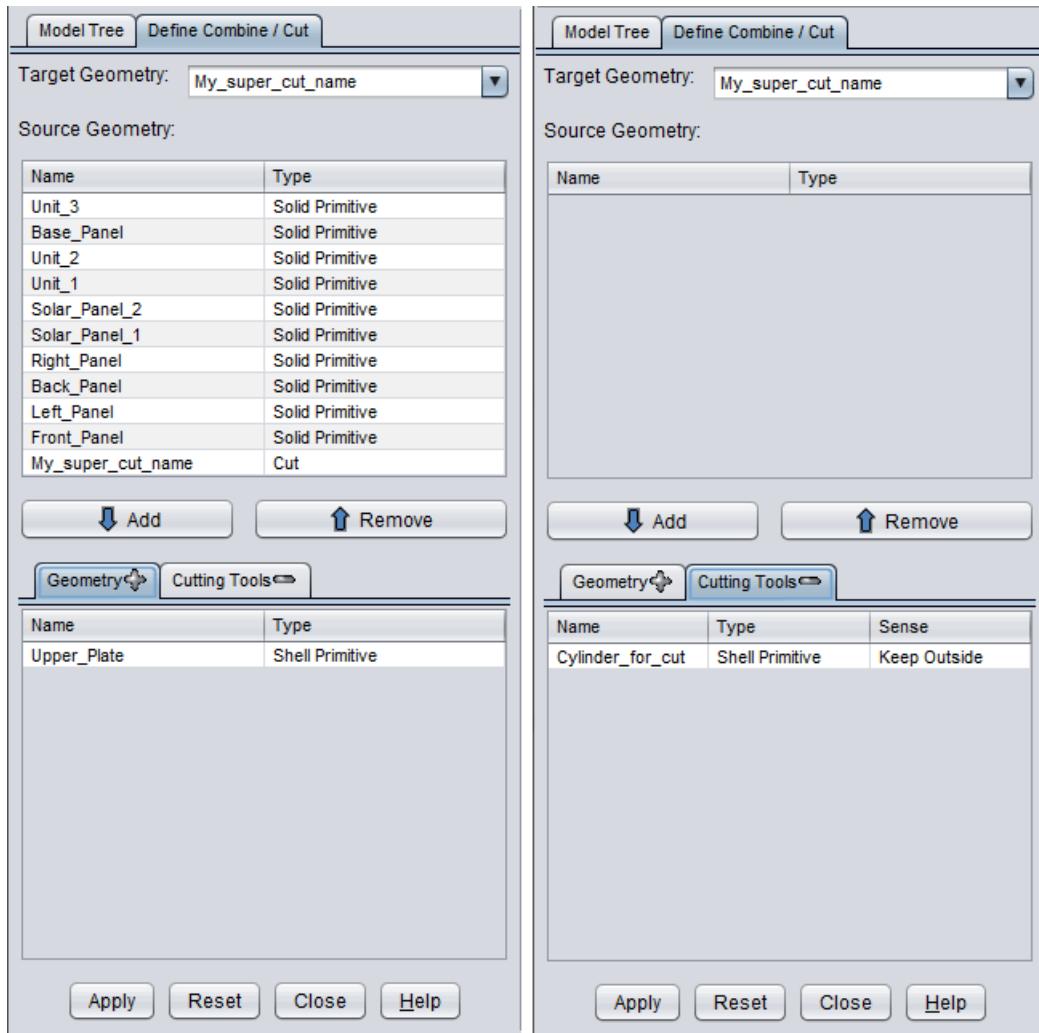


Figure 3-24 Define Combine or Cut Dialog

In the source geometry field, select the shell you want to cut (which is **Upper_Plate** in this model), and the cutter (which is **Cylinder_for_cut**), ensure to select the correct **Sense** in the Cutting Tools tab; then click **Apply**.

If the cut is not defined in the right sense, you can change the sense by right-clicking on the cutting tool in the model tree and selecting **Switch Sense** as shown in Figure 3-25

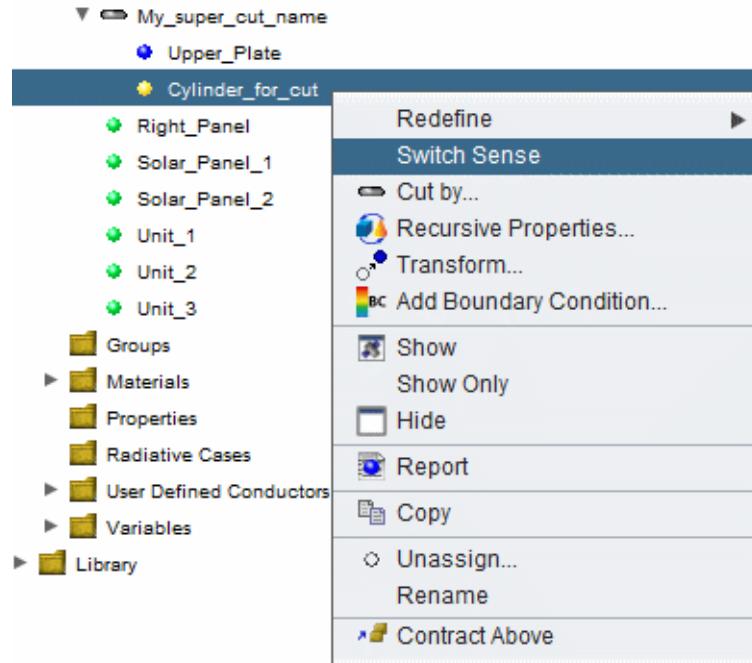


Figure 3-25 Cutting Sense

The first antenna now needs to be defined. This antenna must be located in the hole of the **Upper_Plate** created using the cut operation; this ensures there is no connection between internal nodes and environment node.

The gap could be checked by defining a cavity as the internal surface of the model (not shown here). If the cavity is calculated as an internal cavity, it implies there is no radiation leakage from the cavity to the environment.

The antenna is created as shown in Figure 3-26

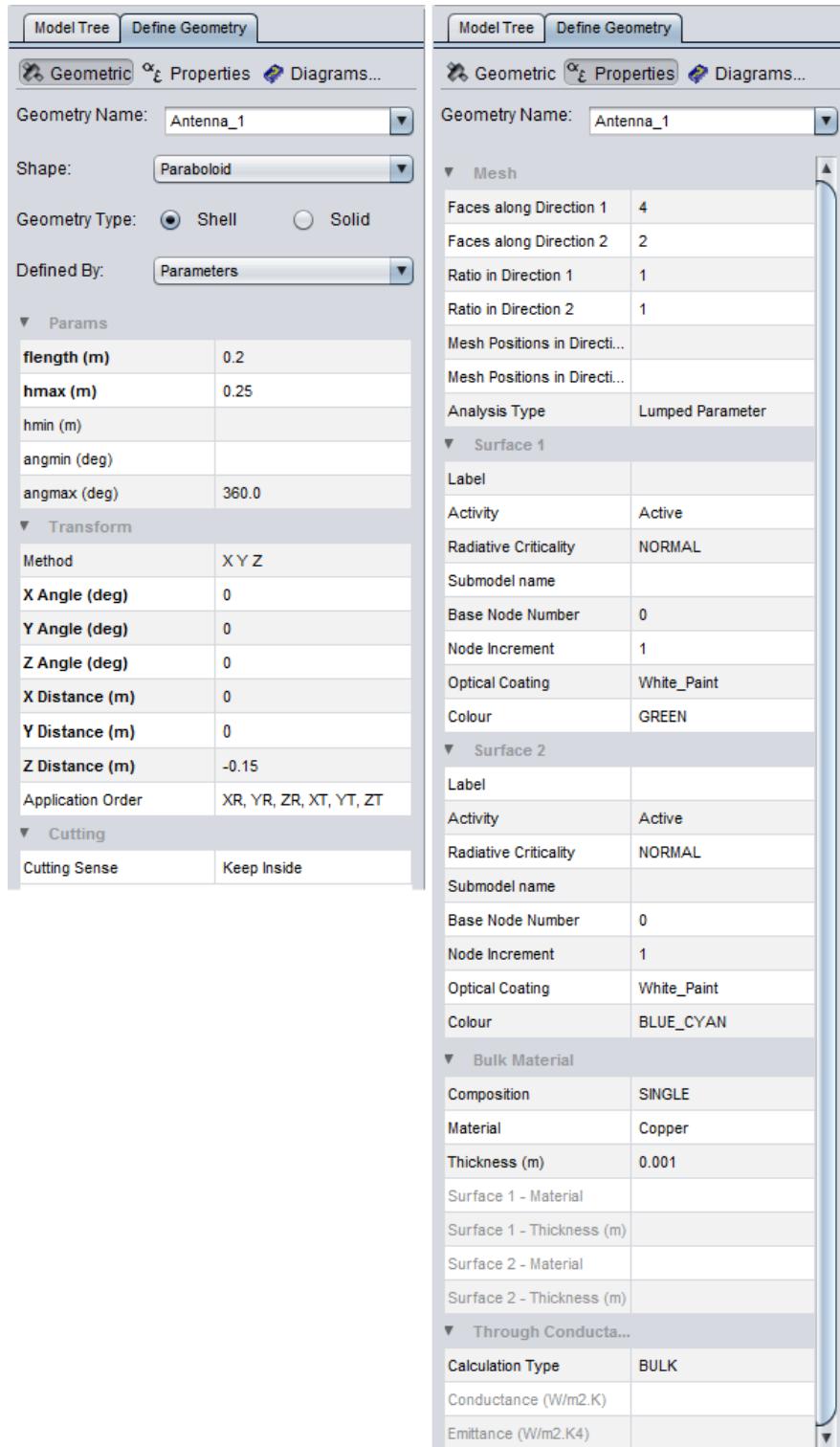


Figure 3-26 Antenna definition

A previous analysis showed that the first antenna (shell) needs to be colder than its current temperature. Therefore, define a cold finger is in order to reduce the temperature of this antenna.

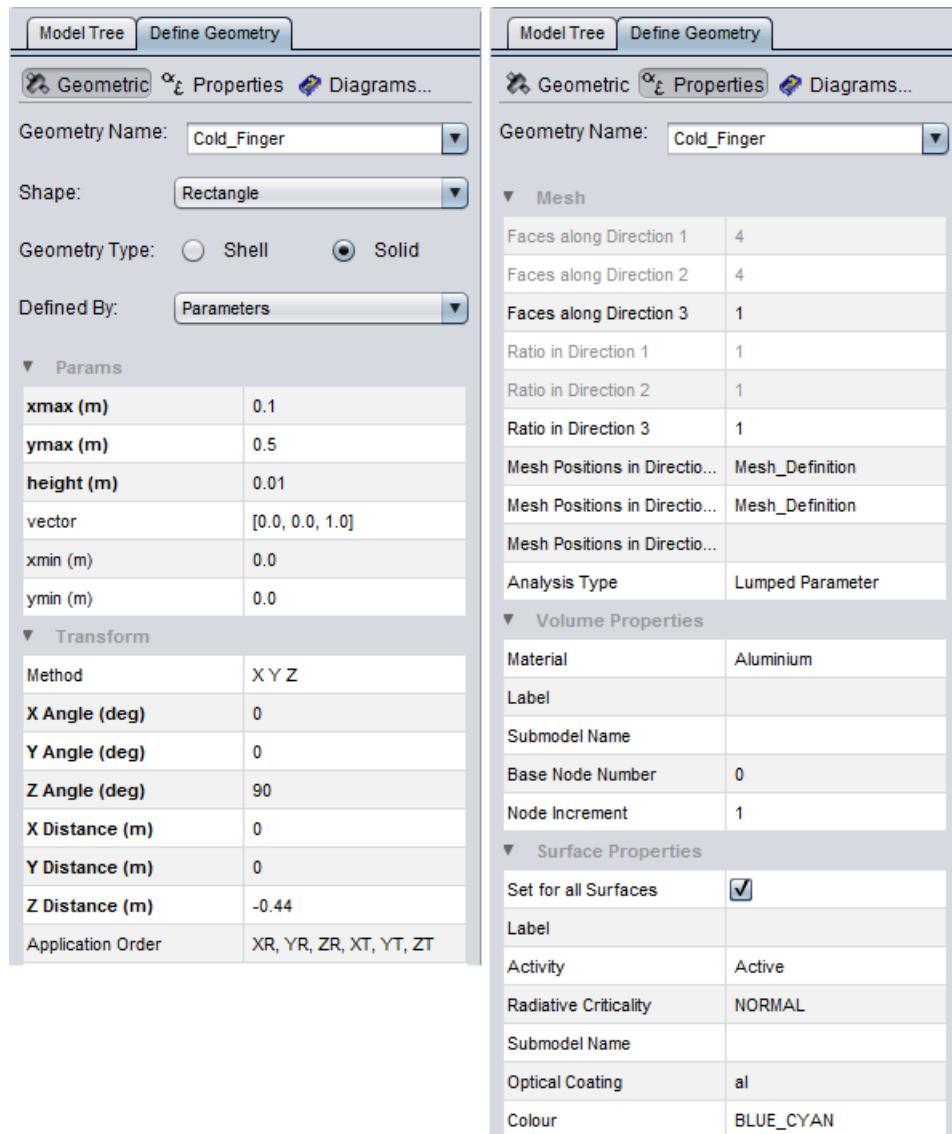


Figure 3-27 Cold finger definition

```
GEOMETRY Antenna_2;
Antenna_2 =
SOLID_PARABOLOID_PARAMS(
flength = 0.2,
hmax = 0.25,
thickness = 0.01,
hmin= 0.0,
nbaseVolume= 0,
ndeltaVolume = 1,
bulk = Copper,
criticality = "NORMAL",
optical = cu,
activity = "Active",
colour = "CYAN",
meshType1 = "regular",
nodes1 = 1,
meshType2 = "regular",
nodes2 = 1,
meshType3 = "regular",
nodes3 = 1);
Antenna_2 = ROTATE(
object_name = Antenna_2,
y_ang = 90.0,
clear =TRUE);
Antenna_2 = TRANSLATE(
object_name = Antenna_2,
x_dist = 0.01,
z_dist = -1.0);
```

Figure 3-28 Geometry unit 1

The Antenna_2 geometry uses the same process as the internal units of the model. Copy and paste the contents of Figure 3-28 above to define another antenna in the model.

Define a Real Vector called coldVect in order to increase the connection between the antenna and the body as shown in Figure 3-29.

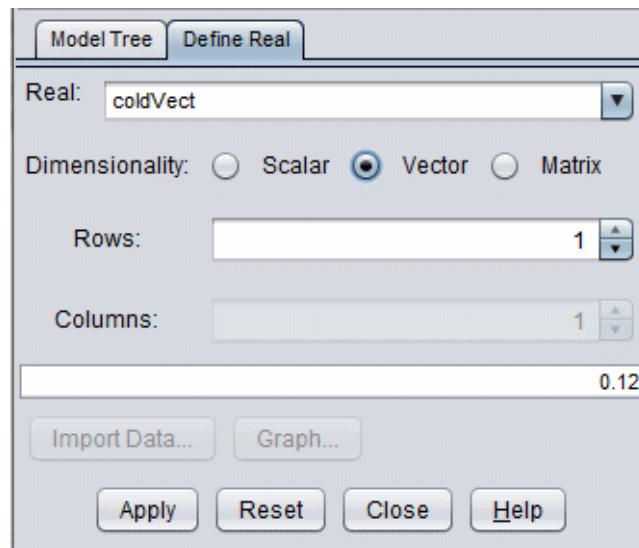


Figure 3-29 Vector definition

Then connect the antenna to the rest of the model by the defining the geometry in Figure 3-30.

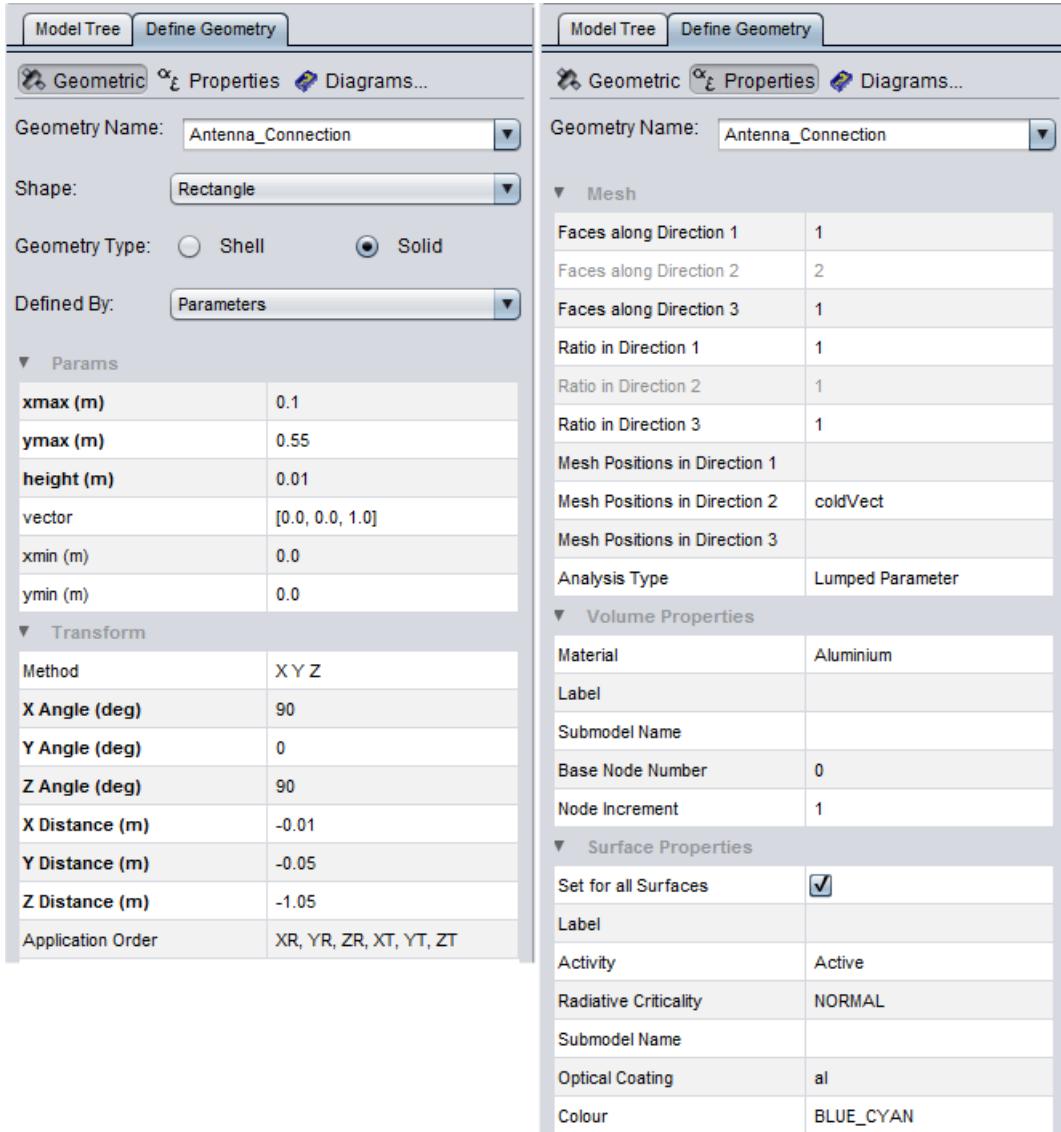


Figure 3-30 Antenna connection definition

The NGNs definition is performed in order to apply BCs on the internal units. Therefore, the size and location of these NGNs must be logical (the size should be smaller than the Body and the location close to the unit. This is why NGNs are defined as Figure 3-31.

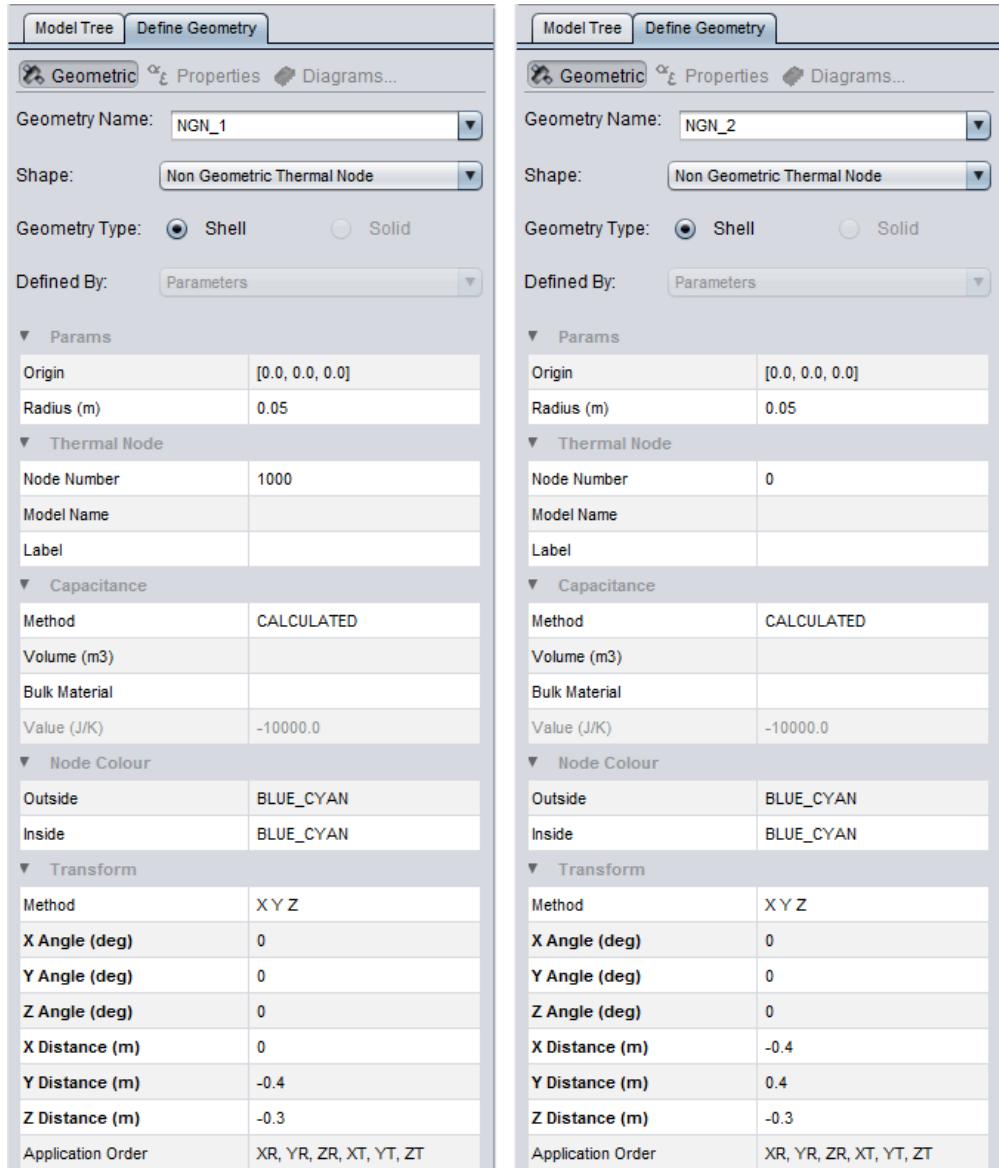


Figure 3-31 NGN definitions

3.3.4.2 Assigning the model

Before performing any radiative calculations and defining boundary conditions, the model must be assigned in order to inform the software which part of the model to be used for the calculation. You can also create combinations from here to create an assembly.

The first combination must be with the solar panels which is the moving part. Then, the fixed part is composed of the rest of the geometry:

- Structure
- Internal units

- Antenna(s)
- Cut

You should only have two main combinations under Model_2. Should you need to add combinations to Model_2, select the geometries required in the model tree (Combine, my_super_cut_name, Antenna_Connection and Antenna_2) and drag them in the Body combine. Your model tree should have combinations the same as Figure 3-32 below, then moving on to Subsection 3.3.4.3: “Define an Assembly”.

Note that under Model_2, you should include two combinations, Solar_pan_moving (including the solar panels) and Body (including all other geometry and combines created earlier).

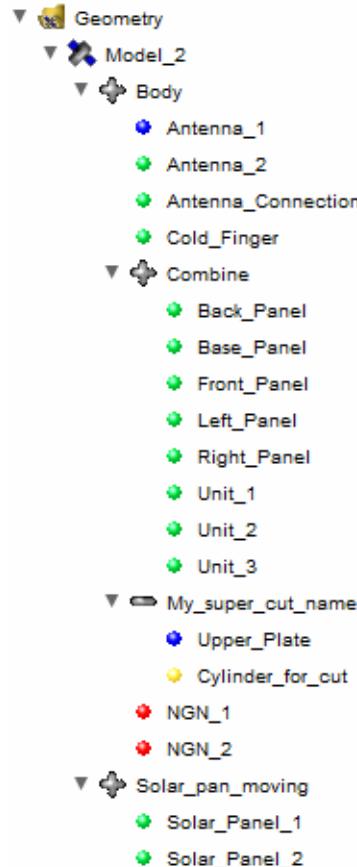


Figure 3-32 Model tree for the assembly

3.3.4.3 Define an Assembly

To define a moving part, right-click in the model in the model tree → Assign (or Reassign if assigning the first time) → Assemble...

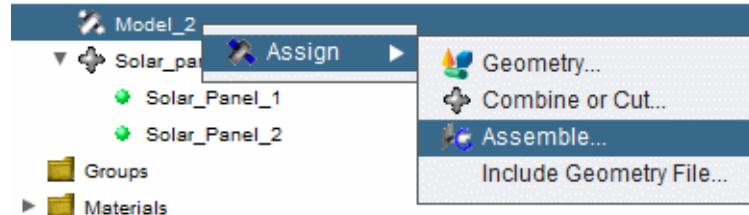


Figure 3-33 Assemble definition

For this moving part, the reference component is Body and the moving component is Solar_pan_moving.

Select and enter definitions in Figure 3-34 and select True Sun as the Preferred Orientation. Select the pointing vector for this case as +X axis and the solar panels will rotate around the +Y axis.



Figure 3-34 Define assembly dialog

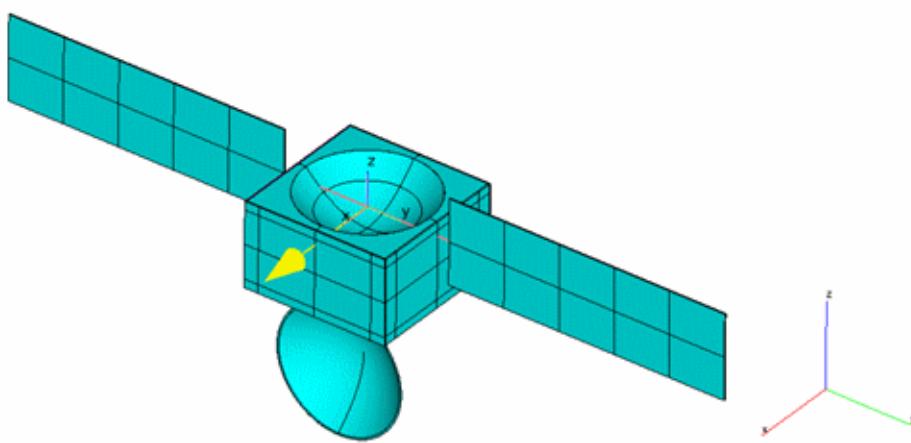


Figure 3-35 Pointing vector and rotation axis definition

As you can see in Figure 3-35, the pointing vector is in yellow and the rotation axis is in pink.

This configuration is basic for the assembly definition and the geometry has been defined in a way to avoid defining a coordinate shift. Indeed, the rotation axis and the pointing vector are located by default in the centre of the coordinates system.

3.3.5 Boundary condition

There are three BCs on this model, two of them have been added to the NGN and then with the conductive link to units, the last one is applied directly on the unit.

Define BC_50deg and BC_20W by selecting Define → Boundary Condition to open the Define Boundary Condition dialog for the NGNs in Figure 3-36

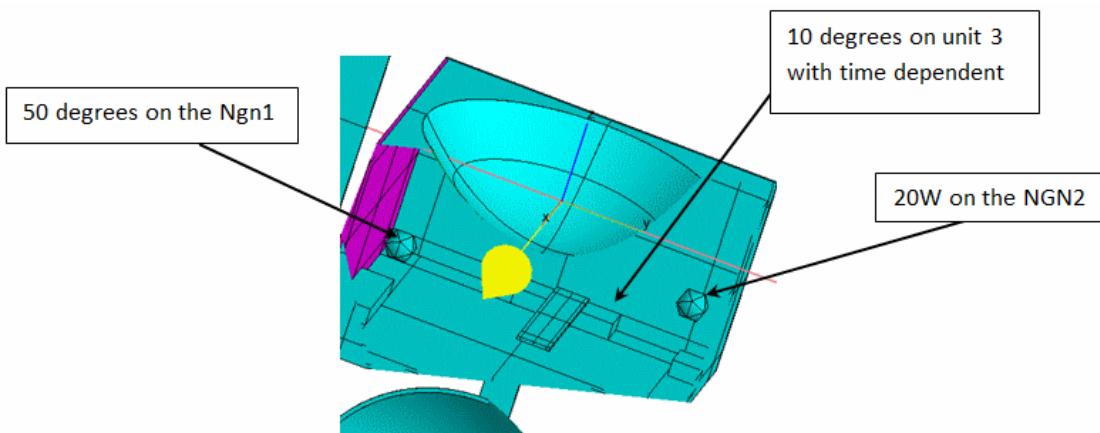


Figure 3-36 BCs of the model

Define the first BC which is applied on the NGN1 as an Initial Temperature. This temperature will change during the calculation process. The second BC on the NGN2 is a Total Area Heat Load. This is a constant dissipation of a unit. The third and last boundary condition is a fixed Temperature applied on the Unit_3 with time dependency in order to reduce the temperature of the antenna by conduction.

Create the Property by copying and pasting the first section of Figure 3-37 into the Command History, then define BC_CF by copying and pasting the second section of Figure 3-37 into the Thermal Command History.

```
REAL prop_T_data[7, 2] = {0.0, 10.0, 999.9, 5.0, 1000.0, 5.0, 4999.0, 3.0,
5000.0, 3.0, 20999.0, 2.0, 21000.0, 2.0};
PROPERTY prop_T;
prop_T = DEFINE_PROPERTY (dependence = "TIME", data = prop_T_data);
```

Copy and paste the following in the Thermal Command History to define BC_CF:

```
BOUNDARY_CONDITION BC_CF;
BC_CF = TEMPERATURE(reference = "Unit_3:volume1", value = prop_T);
```

Figure 3-37 Create property

Define boundary conditions BC_50deg and BC_20W in Figure 3-36 by selecting Define → Boundary Condition in the ribbon bar.

3.3.6 Conductive links (User Defined Conductors)

A cut has been defined for the antenna and there is no automatic conduction generated between the upper plate and the antenna structure. Therefore, Groups have been created from the body in order to be connected to the upper plate.

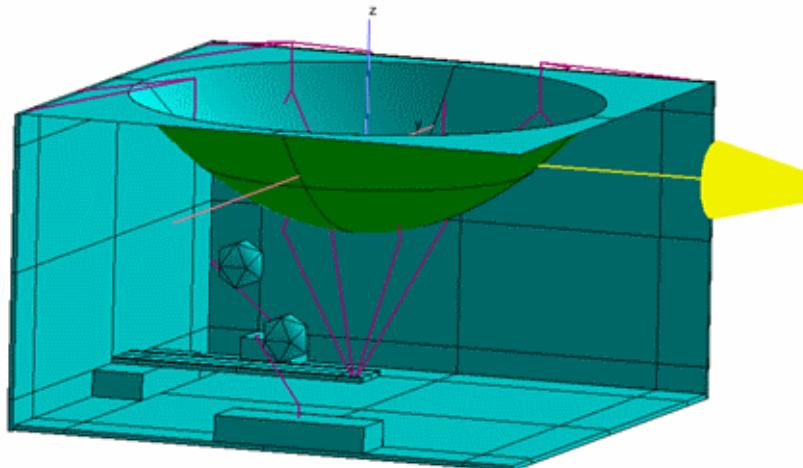


Figure 3-38 Conductive links

Create User-Defined Conductors for the BCs between the NGN and the units, as shown above in Figure 3-38.

To finish the conductors, a User-Defined Conductor needs to be created between the cold finger and the antenna. The temperature of the antenna might be hot compared to the rest of the model, therefore, create groups for the body has shown in Figure 3-39.

To group the antenna and body, set the Picking Mode to Face, and click on the faces to select them (hold Ctrl to select more than one face). Then, right-click and select Group as shown in Figure 3-39, name the antenna connection group grp1 and the body connection group g1, g2, g3 and g4 for each corner (see Figure 3-39).

When a User-Defined Conductor needs to be defined, the Group will be available for reference, as shown in Figure 3-40.

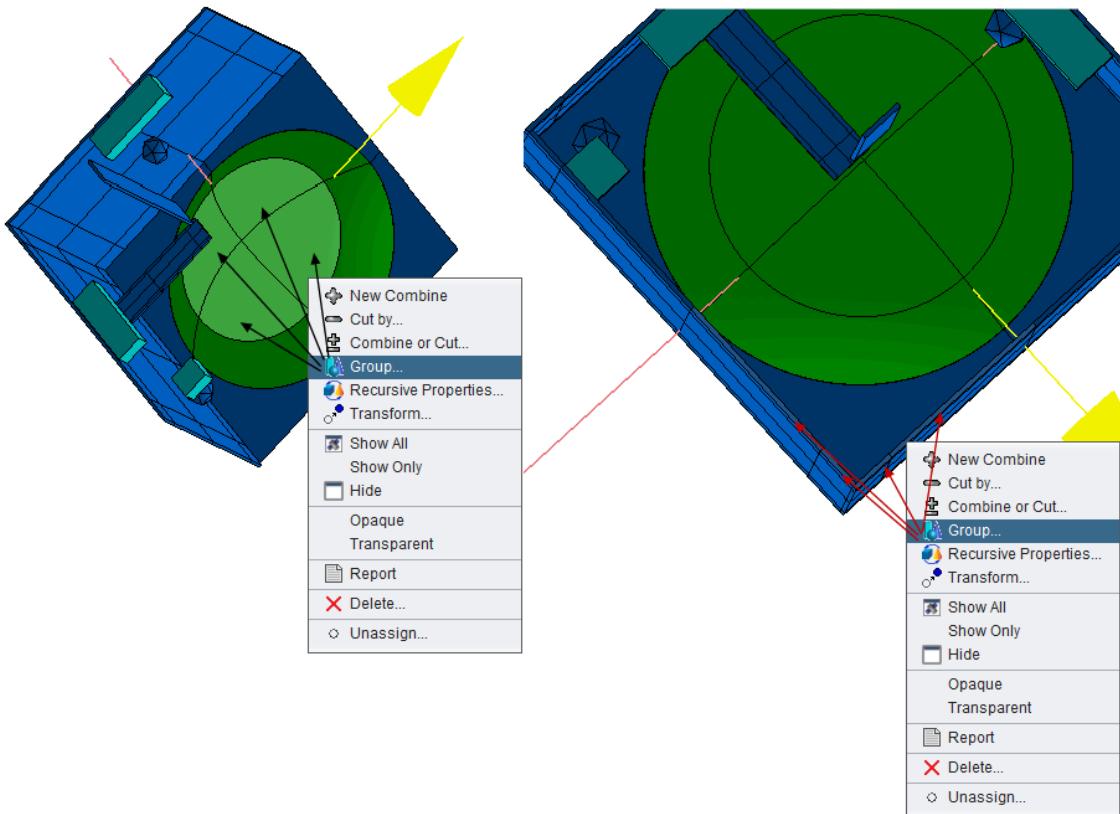


Figure 3-39 Group for antenna connection (left) and body connection (right)

These Groups (grp1 and g1 to g4) must be connected to each corner face of their **Upper_Plate**, as shown in Figure 3-40.

For the destination of the conductive User-Defined Conductors, it is preferable to choose the Groups created and the face of the **Upper_Plate** as a destination. Select **Define** → **User Defined Conductor** and complete as Figure 3-40 for each group. The face can be picked for the **Destination Reference** on the model itself, then define **Method** → **Value** and **Value** → 800W/K. Name the next conductors Cond_2 for g2, Cond_3 for g3 and Cond_4 for g4.

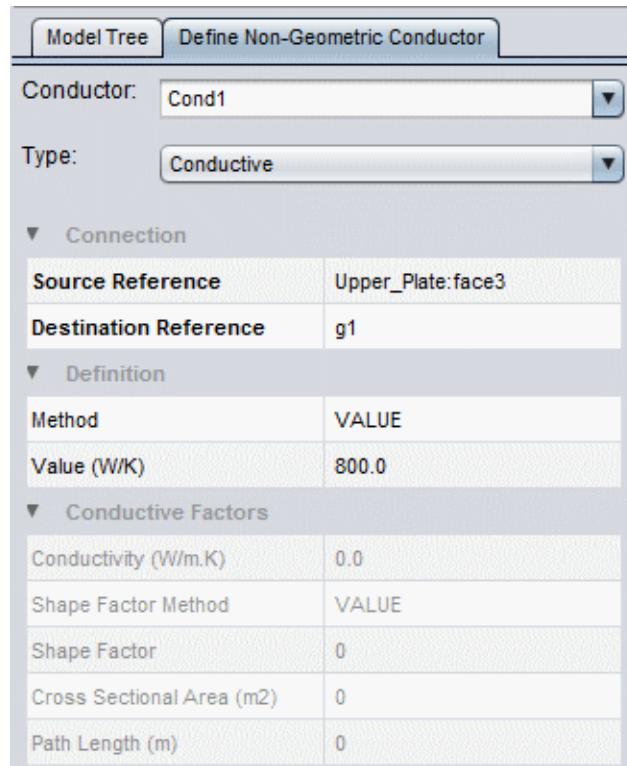


Figure 3-40 Conductor link definition

The Group for the faces under the antenna is named C1 and connected to the Cold_finger:face2 (define Method → Value, Value → 800W/K.).

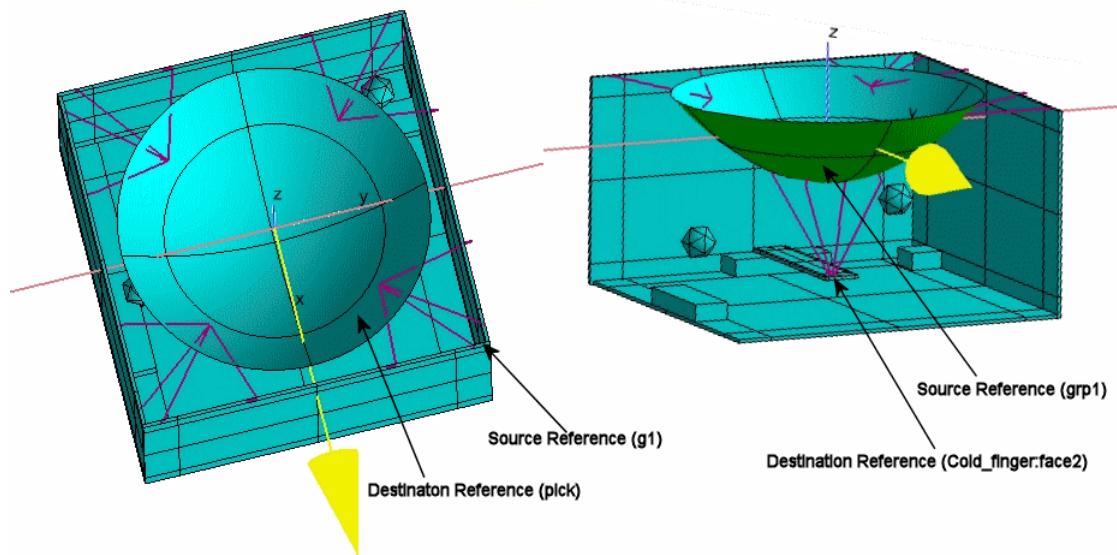


Figure 3-41 Conductor connections

To finish with the conductors, you will define four User-Defined Conductors into the antenna (replace the intra conductor). For these conductors, the reference and the destination does not matter. You can choose the reference and destination component (value for the conductance is 800 W/K).

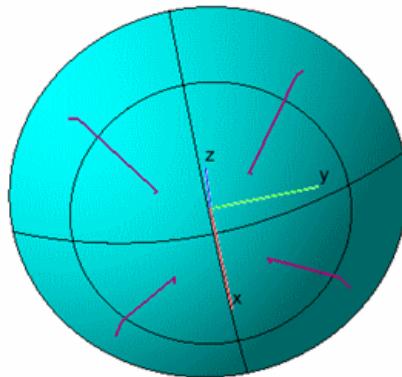


Figure 3-42 Intra conductor

3.3.7 Contact Zone

Only one Contact Zone is defined in this model between Antenna_2 and the Antenna_Connection, therefore, if the criticality of the source reference geometry (surface) is defined as normal (default value), 10,000 rays will be fired perpendicularly from this surface and the number of rays that hit the destination reference will be recorded. This calculates the actual faces in contact and the associated contact area. Due to the large difference in surface areas of these surfaces, if the source reference is the antenna connection, rays will probably find fewer surfaces and be more accurate than if the source reference is defined as the antenna.

Select Define → Contact Zone and name it cii. The Source and Destination References are shown in Figure 3-43. Note that the Source Reference is Antenna_Connection:Face3. Select Surface from the Destination Reference drop-down menu before selecting the Antenna_2 face. Set the Contact Conductance to 1500 W/m²K, and the Maximum Gap to 0.1m.

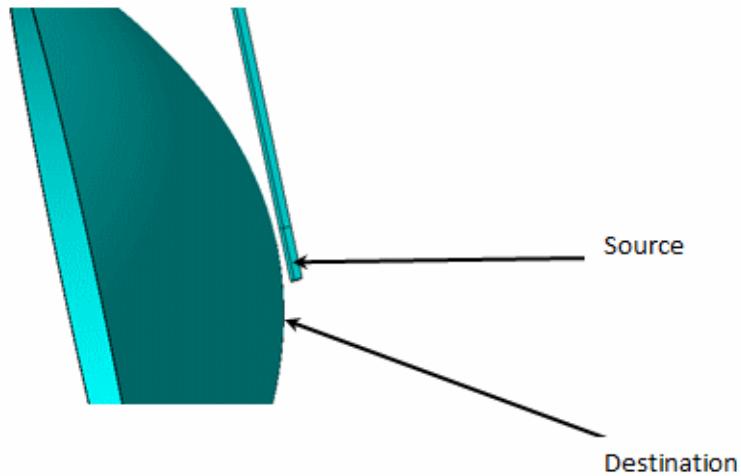


Figure 3-43 Contact surface definition

To check if the Contact Zone has been created correctly, it is recommended you right-click on the Contact Zone (cii) in the model tree and select Report.

```
*****
Model name = Model_2
Report produced by REPORT_SYMBOL procedure at 15:11 Thu 24 Jul 2014
*****
```

Contact Zone	cii
Source Type	FACE
Source Reference	Antenna_connection:face3
Destination Type	SURFACE
Destination Reference	Antenna_2:surface1
Contact Conductance (W/m2K)	1500.
Maximum Gap (m)	0.1
Number of Sample Points	10000
Node Pair Areas (m2)	
(416, 423)	0.0066
Total Contact Area (m2)	0.0066

Figure 3-44 Contact zone report

If a surface value appears in the command history, the Contact Zone has been performed correctly.

3.4 Running radiative cases

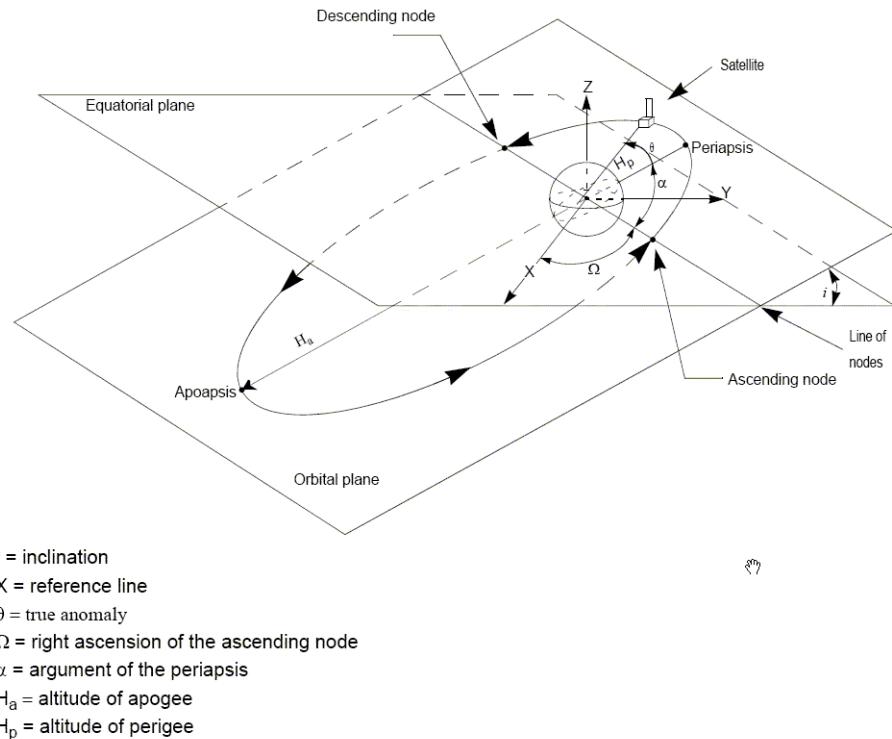


Figure 3-45 Orbit parameters

Once you are happy with the geometry definition of your model, you can use the Radiative module in Workbench to define the orbit parameters and run the radiative calculations. This section assumes that you are familiar with Radiative Case definition and orbital elements.

Each Radiative Case will relate to a single radiative analysis for a single spacecraft with a single behaviour definition in a single orbit. Any changes to the spacecraft, its behaviour, the orbit or the analysis run, will require an additional Radiative Case.

This part is special because three Radiative Cases will be performed in only one Analysis Case. Remember that each Radiative Case needs to be executed before performing a thermal analysis through an Analysis Case.

The Radiative Cases will be defined as a Hohmman orbit, which is one circular orbit, one transfer orbit, and another circular. This example model is trying to give you another view of a Radiative Case and an option has been added for each Radiative Case.

3.4.1 Radiative Cases definition

Double-click on the Radiative Case folder in the model tree to open the Radiative Case Dialog, name the Radiative Case RadCase01_circular and select Orbital modelling as the Type.

You do not need to make any changes for the Environment tab for this Radiative Case. Nevertheless, the orbit parameters for the satellite should be defined below.

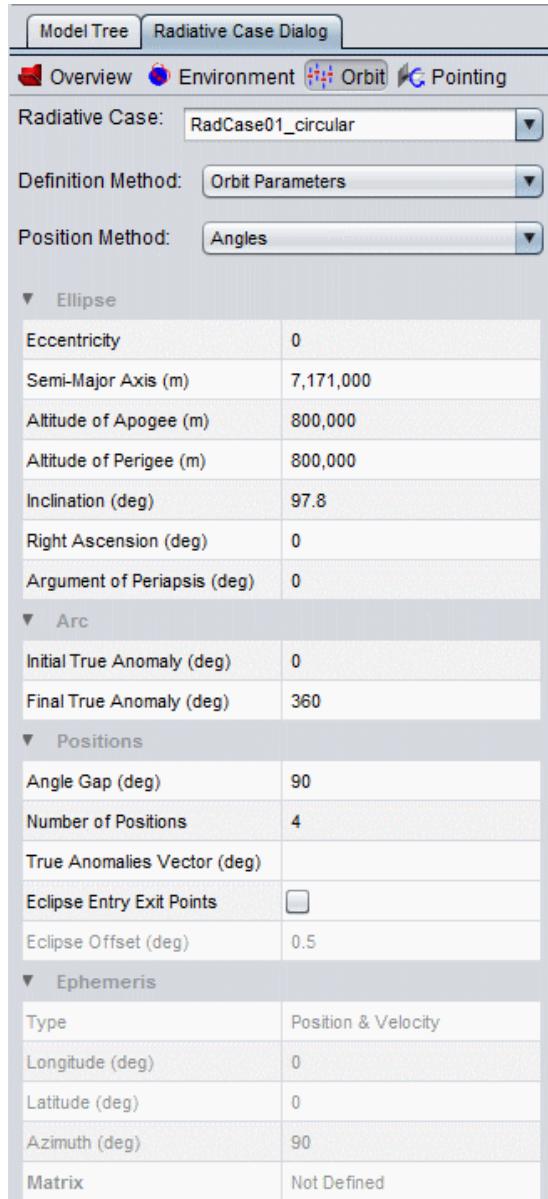


Figure 3-46 Orbit for the first radiative case

This is a typical polar, low orbit for a Hohmann transfer. The orbit is circular because the altitude of apogee and perigee is the same.

The attitude of the spacecraft has been defined with respect to the moving part. Indeed, if the solar panels need to get power the faces need to be in front of the Sun. Hence the attitude for the first orbit is defined in Figure 3-47 and Figure 3-48.

Select Define → Property to define Prop1 detailed in Figure 3-47 below to specify the movement in order to get the antenna as much as possible in front of the Sun during the sunlight area.

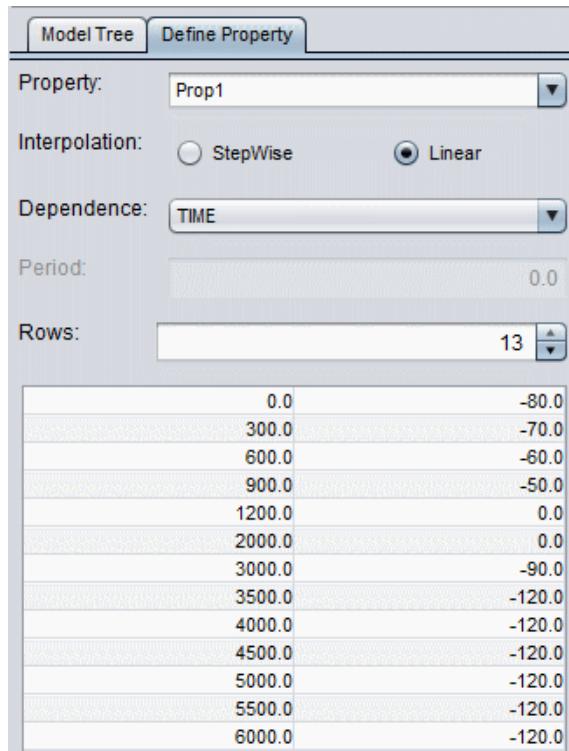


Figure 3-47 Property definition for PSY Euler angle

Define a movement around Psi in Figure 3-48 by selecting Pointing in the Radiative Case Dialog.



Figure 3-48 Attitude of the spacecraft for the first orbit

It is possible to check exactly what you have done in the GUI if the number of positions is modified (without running the Radiative Case). In this case 30 positions have been set in the Orbit tab in order to display the orbit with the evolution of the spacecraft around it.

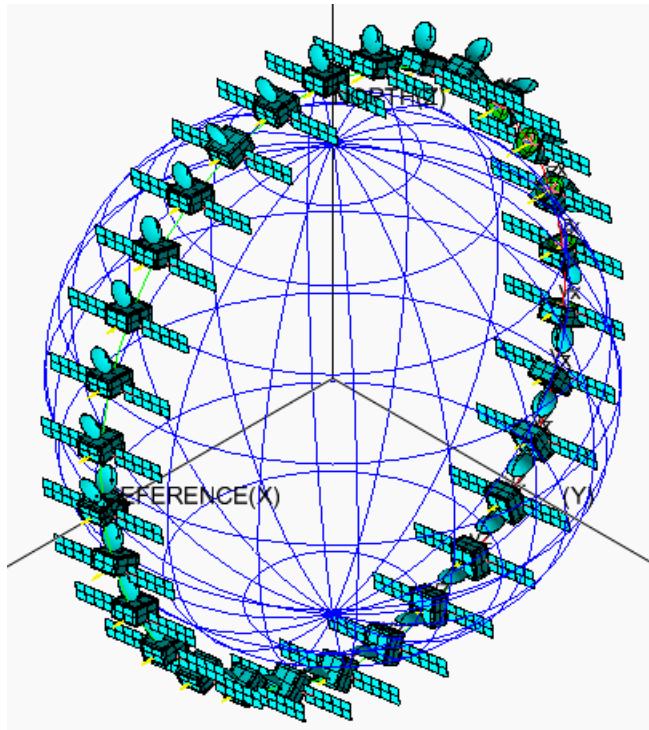


Figure 3-49 Orbit display with 30 positions

Once your Radiative Case is defined, Execute it by ensuring the default Calculations are set to: REF MCRT, Solar Absorbed MCRT, Planet and Albedo Abs MCRT.

Note, if the Radiative Case does not execute, one or more optical properties may not be set, check that each side and edge of the model has assigned optical properties as per Figure 3-2.

The second Radiative Case is a transfer orbit. We know that to get the transfer orbit, the satellite needs to be fast enough to be extracted from the Earth. This velocity is named liberation velocity. In this case, the satellite will need to cycle 3 times to achieve this velocity.

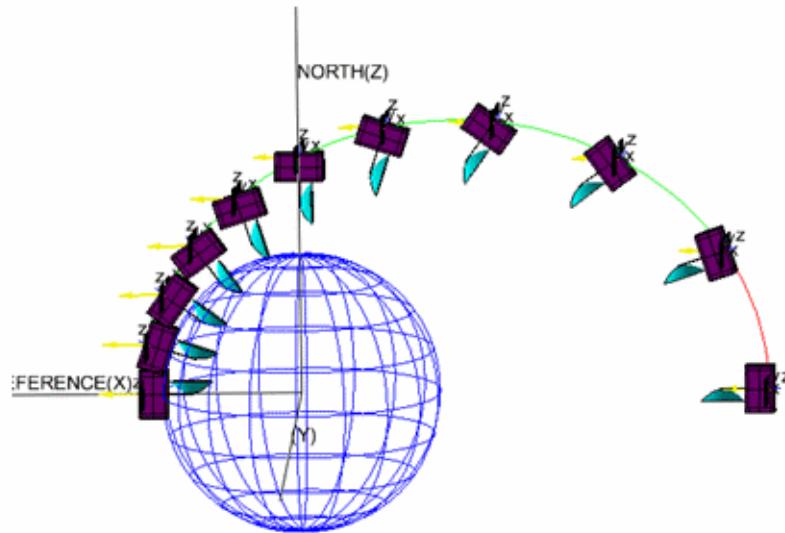


Figure 3-50 Second orbit (transfer orbit)

As we can see, a transfer orbit is not a complete orbit. Hence, the true anomaly for this orbit must be from 0 to 180 degrees. The Altitude of Apogee is 15000000m, it is also possible to define the eccentricity of this orbit by this following formula.

$$e = \frac{((A_p + R_{Earth}) - (A_a + R_{Earth}))}{((A_p + R_{Earth}) + (A_a + R_{Earth}))}$$

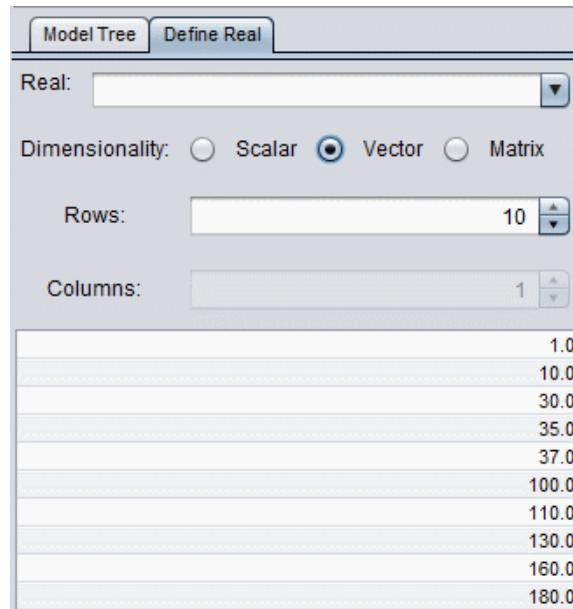


Figure 3-51 Vector defined for the second radiative case

For the transfer orbit, you will need to define a Real Vector in order to decide the number of positions required for this orbit. As there is a lot of information in this orbit, 10 orbit positions have been chosen detailed in the Real Vector in Figure 3-51.

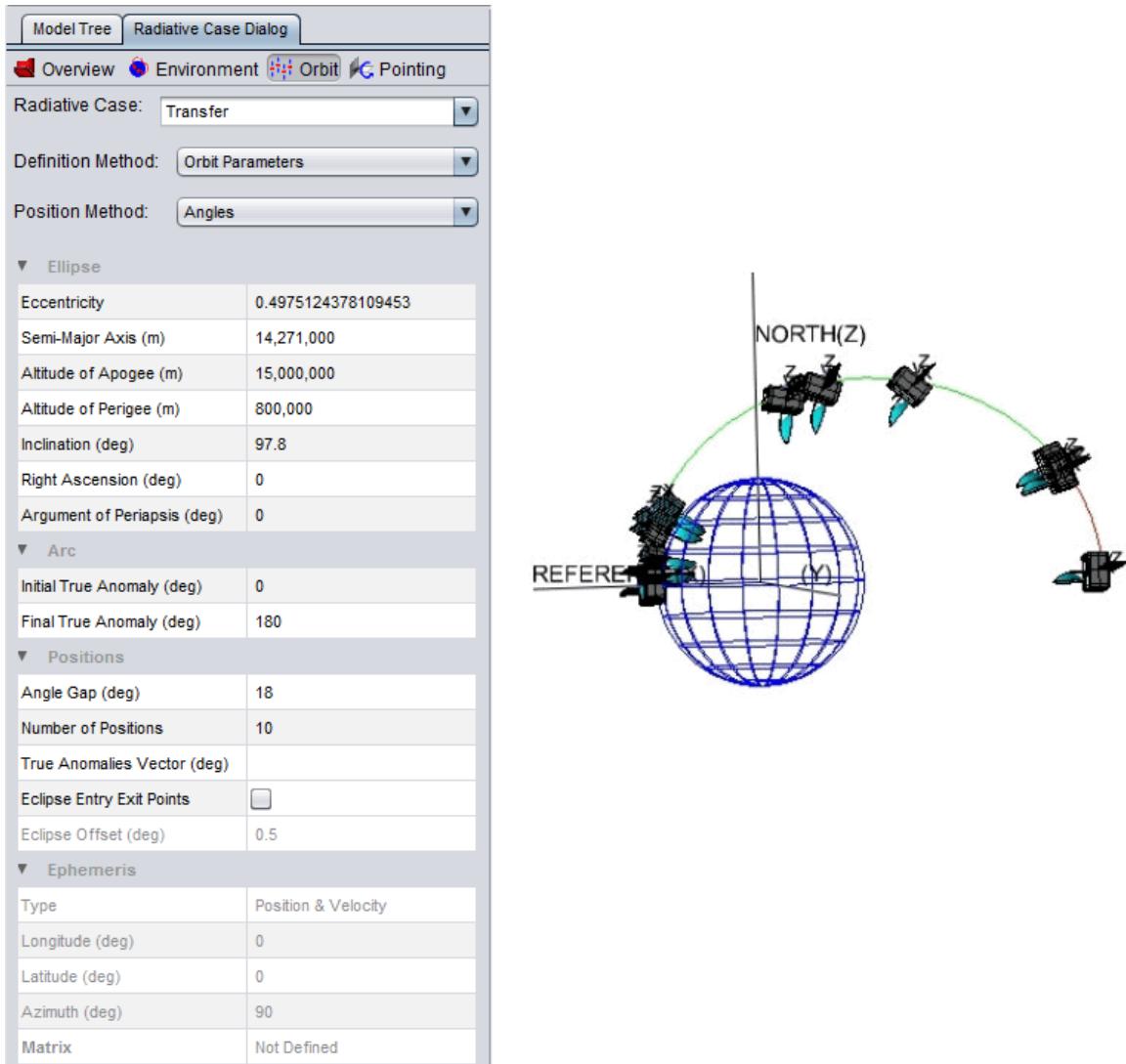


Figure 3-52 Orbit for the transfer

According to Figure 3-51, the orbit displayed in Figure 3-52 shows the impact of the Real Vector defined. You will need to define a vector (in our case, the one shown in Figure 3-51), with this Real Vector referenced within the Radiative Case as the true anomalies vector.

The execution of this Radiative Case is the same as the first one. All default values set have not been changed.

For the last orbit, because the transfer orbit is not a complete orbit, the right ascension and the true anomaly need to be modified. The true anomaly should start from 0 to 360 degree for a complete orbit, but the argument of Periapsis should start from 180 degree. The last

orbit is defined in Figure 3-53:

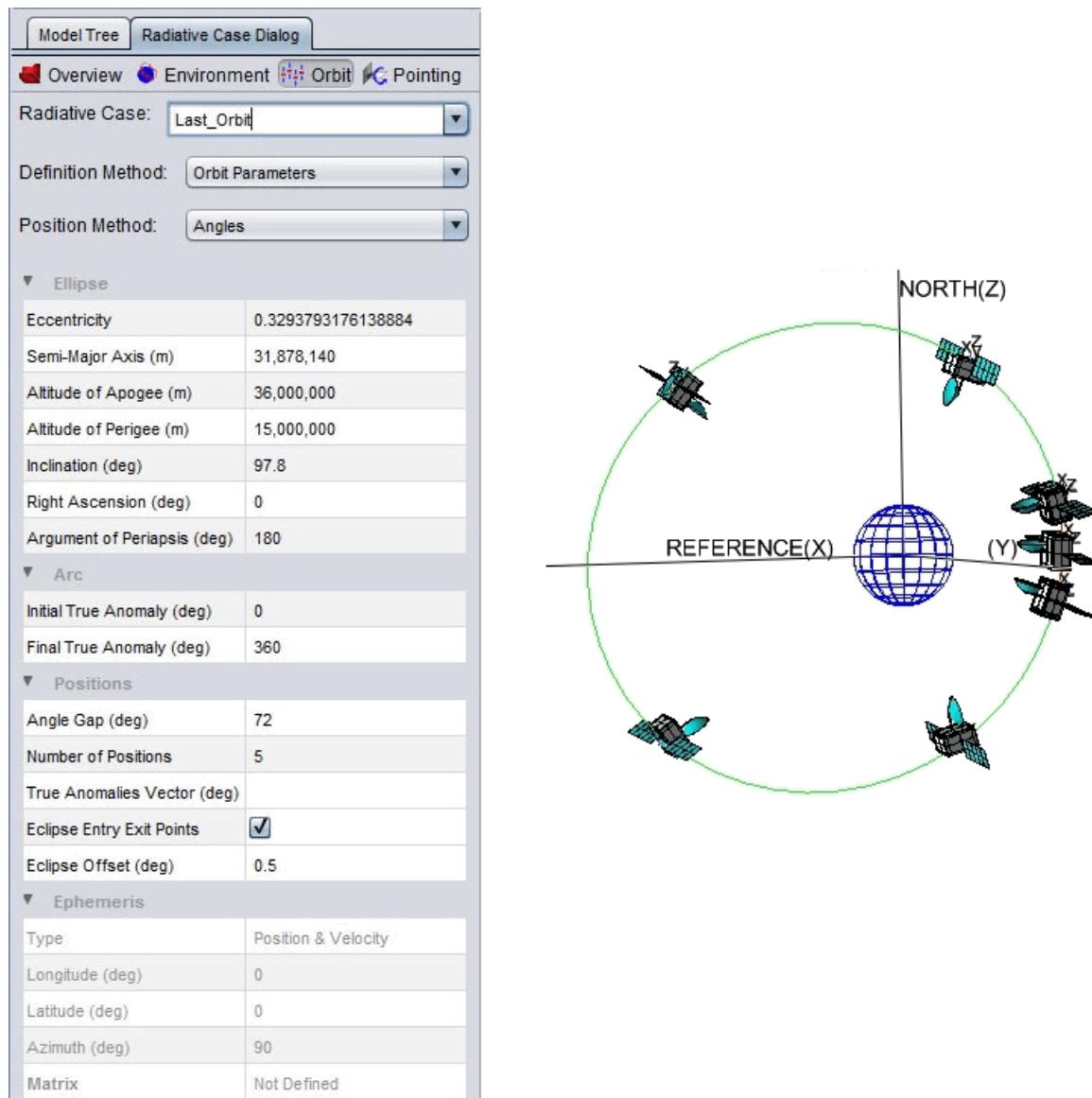


Figure 3-53 Last Orbit

As you can see in Figure 3-53 above, the Eclipse Entry Exit Points box has been checked. It means that when the satellite goes in eclipse area, the software will automatically add two more positions (one just before the eclipse entry point, one just after and the same for the eclipse exit point). These positions correspond to the penumbra.

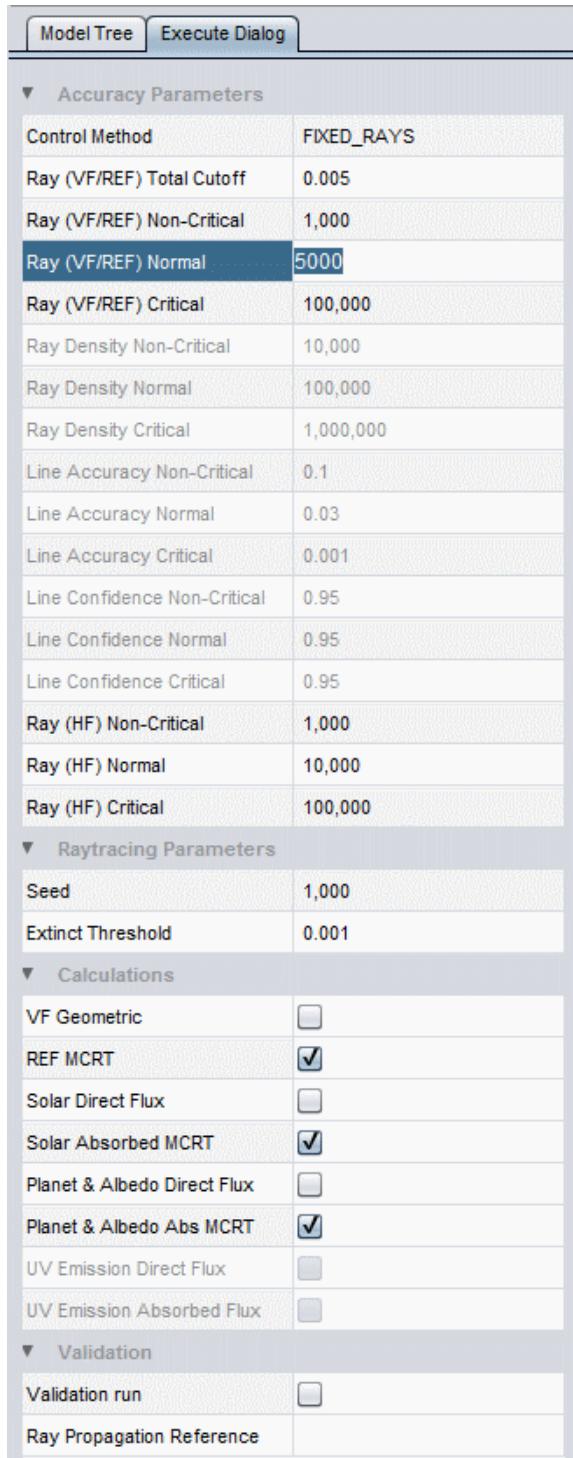


Figure 3-54 Execution of the last radiative case

For the last orbit, it is not required to be accurate. Therefore, the value for the number of rays fired per (Normal) face has been changed to 5000 rays. The criticality of most of the solid is set to normal, hence only this value needs to be changed.

Ensure all Radiative Cases have been executed by right-clicking on the Radiative Case, Select Open, then right-click on the Radiative Case again and select Execute.

3.4.2 ACG (Auto-generate Conductive Generation)

Once the orbit has been defined and executed, select Thermal → Auto Generate Conductive Interfaces... This option detects all contacts between the geometry.

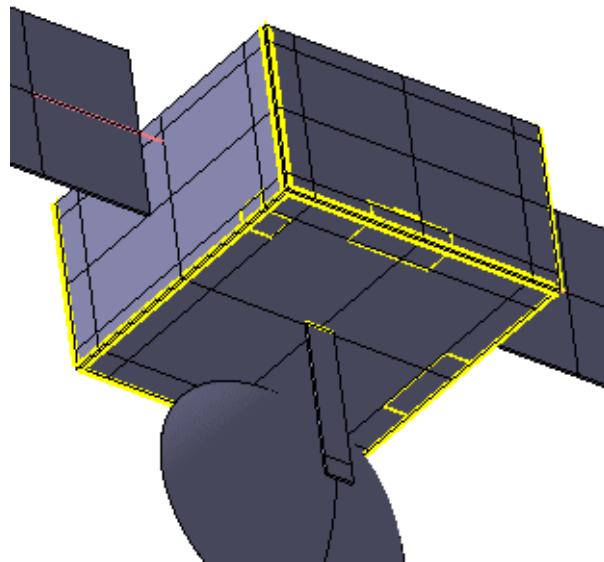


Figure 3-55 Contacts between face to face solids

Make sure that all contacts found by ACG are Fused. To ensure that the contact between solids are valid, right-click on the Conductive Interfaces folder → Validate Conductive Interface. Then check in the report to see if there are errors on some interfaces.

3.5 Solving with ESATAN-TMS Thermal

In this section, you will solve the thermal model and generate a TMM file and execute it in order to get the temperature evolution around all orbits.

Define an Analysis Case by double-clicking on acase01 in the in the Analysis Case folder from the model tree to open the Define Analysis Case dialog. Name the analysis case transfer and select Chained Radiative Cases as the Analysis Case Type from as shown in Figure 3-56 to select the output of your analysis case.

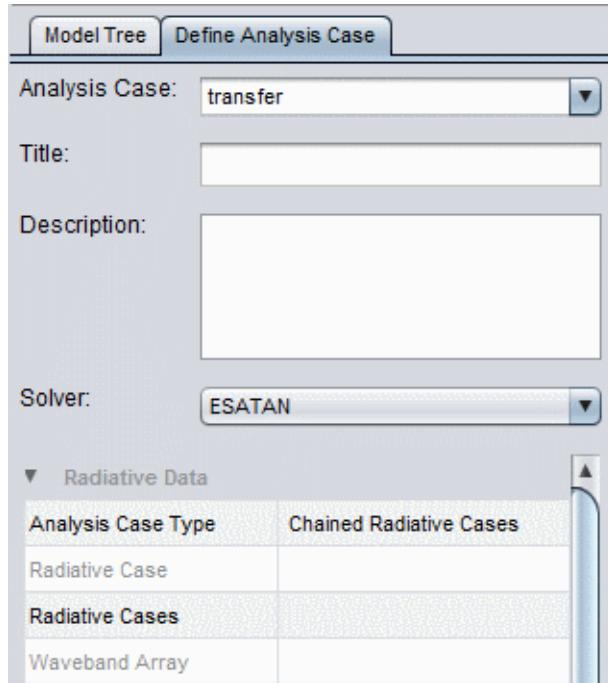


Figure 3-56 Analysis case definition

At this stage, you have to select Radiative Cases with respect to the order definition, which is circular orbit, transfer orbit, and eccentric orbit.

In the Radiative Cases option select the box to open the Chained Radiative Cases diaolg and include your Radiative Cases. Click on the Add button to add a row and select your Radiative Cases and select them with respect to the definition order. Once the three Radiative Cases in Figure 3-57 have been selected, click OK.

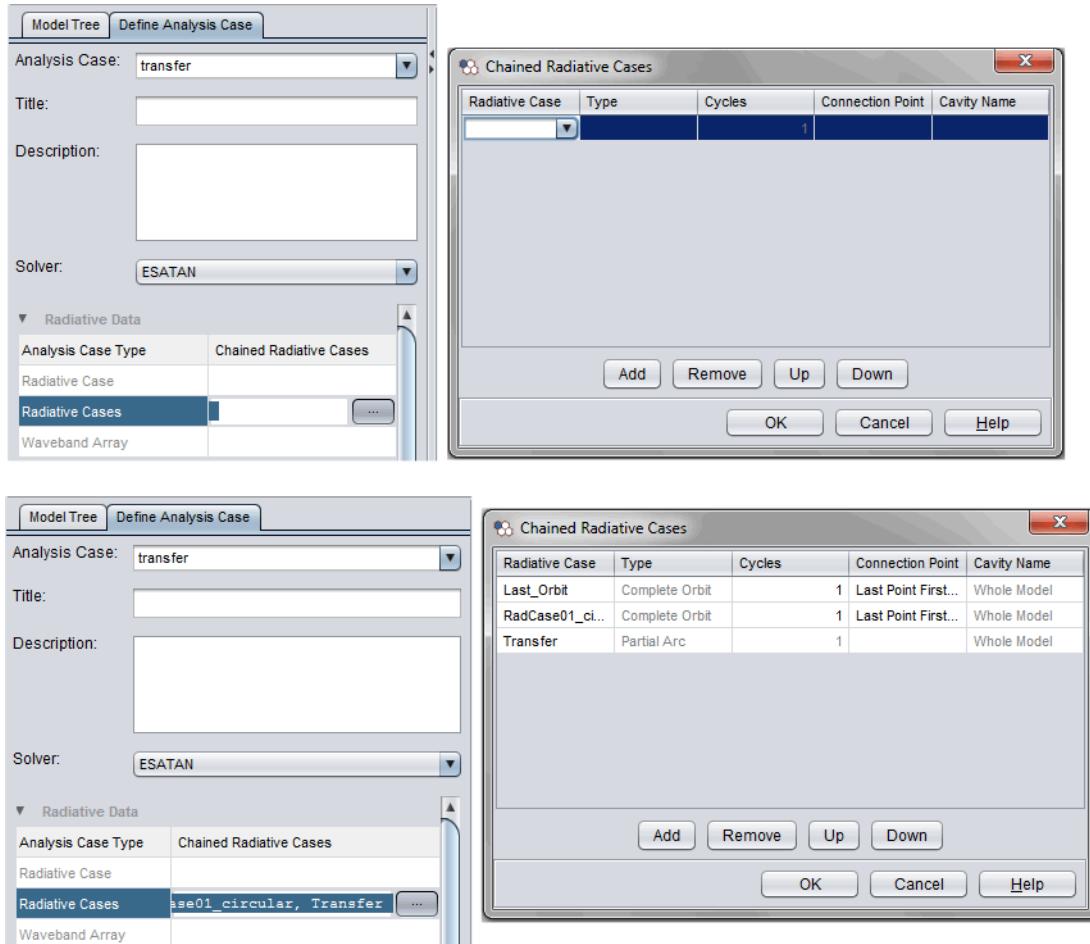


Figure 3-57 Chained Analysis Case Definition

In this case, you are not required to do anything else, nevertheless, it is possible to cycle the orbit. For instance, if the satellite needs to do more than one orbit around of the earth, the **Cycles** option could be 3,1,1: For three orbits regarding the circular orbit, one for the transfer and one for the eccentric orbit. The next step is to click on the Solution Control button  to display the Solution Control Dialog.

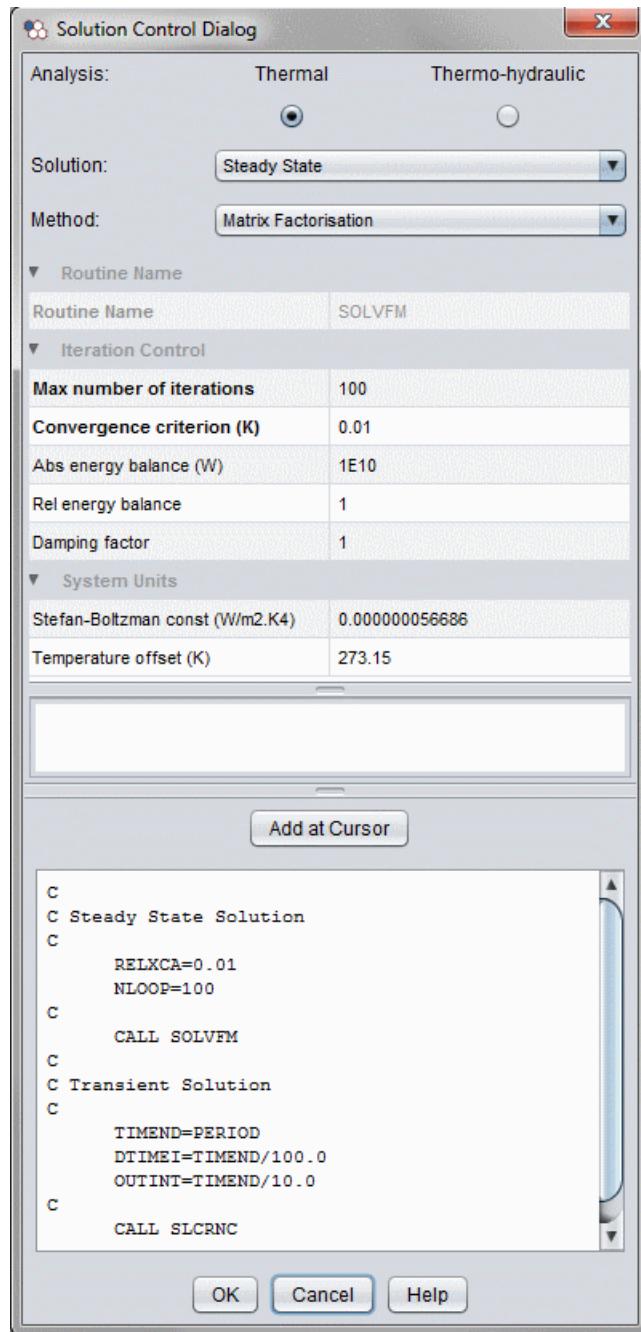


Figure 3-58 Template file creation

In the Solution Control Dialog, define a steady state calculation followed by a transient calculation. The steady state analysis is performed first in order to improve the convergence.

To define the steady state solution, keep the default values when you open the Solution Control Dialog and click on Add at Cursor. Then, for the transient analysis, the default value for TIMEND is the time for the complete orbit, with the three Radiative Cases.

The Max number of iterations (NLOOP) and Convergence criterion (K) (RELXCA) value are the same as the steady state solution, and the OUTINT is by default TIMEND/10. This value (OUTINT) will display the temperature of your model every second and you can modify this value for the calculation.

The time step for the calculation (time between each calculation), is defined by you (in our case 1s), and is the DTIMEI value. Every 1 second, the solver will generate a result for the temperature.

See the Thermal Engineering Manual and User Manual for more values.

NLOOP is the maximum number of loops defined, which ESATAN-TMS can use for the convergence. This value is mostly never exceeded, however it could appear in some cases where this value needs to be increased. Therefore, a clear warning message located in the output file (.out file) will warn that convergence has not been achieved and therefore this value may need to be increased to achieve convergence

RELXCA is the primary convergence criterion used by the thermal solvers is modulus of temperature change over iteration. The user specifies the required value with control constant RELXCA (typically 0.001 for a steady state solution), and the program stores the calculated maximum (over all nodes) in control constant RELXCC. The solution is assumed to be converged when $RELXCC < RELXCA$.

The TIMEO constant is the time at start of solution step.

Remember to click on the Add at cursor button in order to add the transient calculation to the analysis.

Once all variables are set as described in the Figure 3-59, click OK.

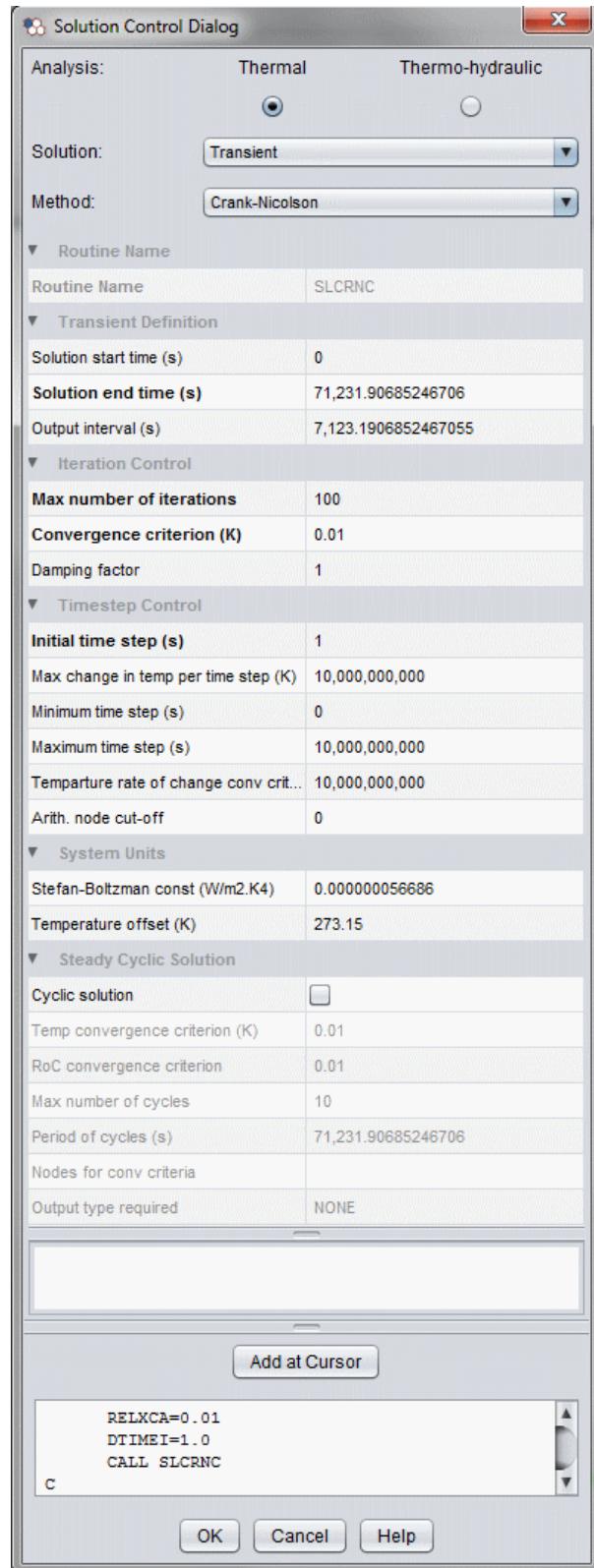


Figure 3-59 Transient analysis definition

This execution block is available in the TMM file and in the execution file located in the model tree, in your analysis case folder.

The next step is to update the **Define Analysis Case** dialog.

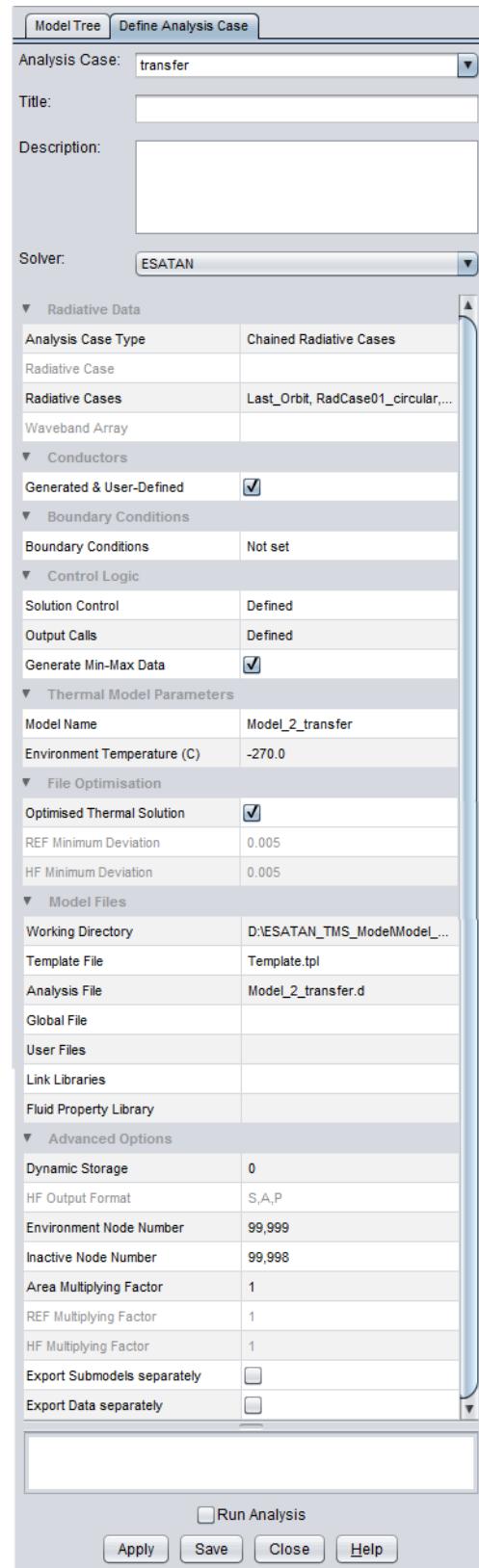


Figure 3-60 Analysis file definition

The last thing to do in the analysis case is, to check or define the Boundary Condition. Select the button  next to Boundary Conditions and select all the items. In this case, tick all of them, however, it is also possible to define another analysis case where only part of the BCs can be selected. To select all of them in one go, click on .

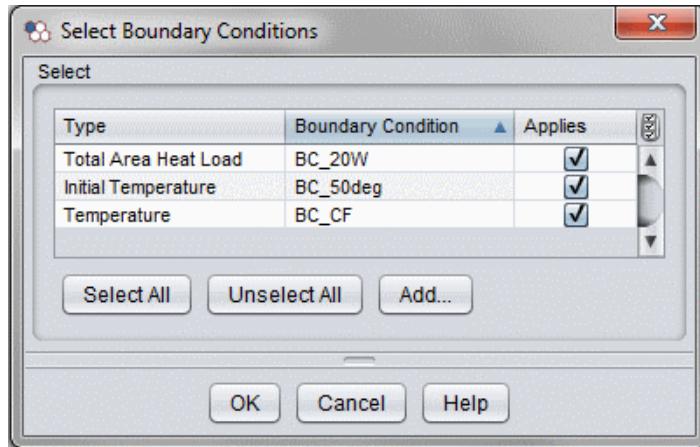


Figure 3-61 Boundary condition definition

If the BCs in Figure 3-61 are not available, it is possible that it was not saved and therefore you will need to define them as per Subsection 3.3.5. There are only three BCs in this model but it could be very different for a larger model (more than 100 BCs).

Once everything is defined correctly, please make sure that the Run Analysis check box at the end of the Define Analysis Case dialog is ticked.

The Analysis Monitor will display once you click Apply where you can check the calculations, and you can check them in the analysis case model tree.

Note that all optical properties need to be defined as Figure 3-2 should your Radiative Case fail to execute. To view optical coatings on your model, click on the Display Data icon in the ribbon bar, select Opticals from the Category menu, Thermo Optical Properties from the Display menu, then click Display. You will need to ensure that all sides and edges of each panel and antenna have optical coatings defined (zoom in and hide panels to check), the grey parts of your model will need to be assigned.

Radiative Cases must also be executed before you run this Analysis Case/

3.6 Post-Processing Thermal Analysis Results

For this model, we only need to display the temperature of the satellite around the orbit. Therefore, we need to use the TMD2 file which is found in the model tree by expanding

Analysis Cases folder, then the Results File folder. It will show as MODEL_2_TRANSFER.TMD.

By default, if a steady state calculation is performed, followed by a transient one, the steady state will have the TMD extension and the transient TMD2. Right-click on the TMD2 file and select Post-Process.

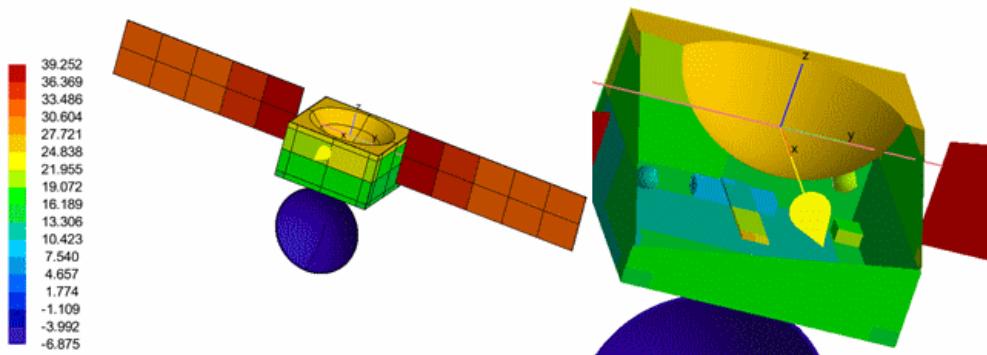


Figure 3-62 First result for the first orbit position

The result for the first orbit position is displayed. Use clipping plane option to check what is happening inside of the model by clicking on  in the ribbon bar.

3.6.1 Post-Process on geometry

To display the temperature for all orbit positions, you will need to select the Mode as Orbital Results in the Display Data dialog and tick the box Display all cycles as it is defines in Figure 3-63.

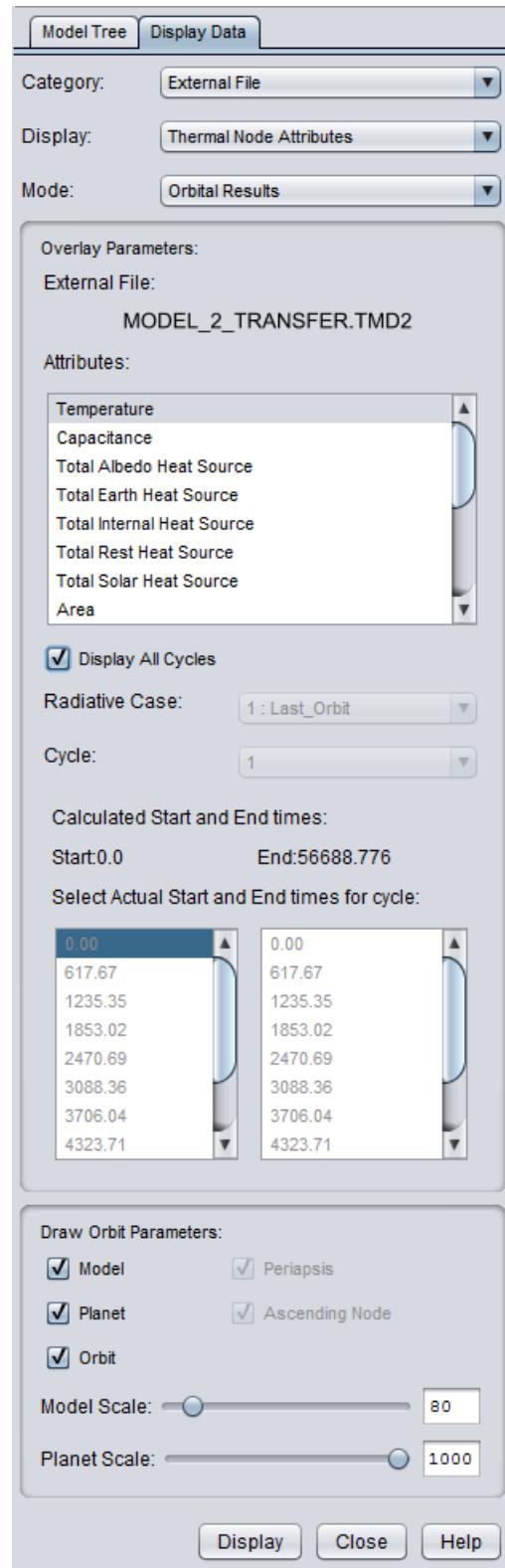


Figure 3-63 Display options

As it is possible to observe in Figure 3-64, the satellite is displayed only every 7115 seconds, which is too small to see any detail for the visualisation.

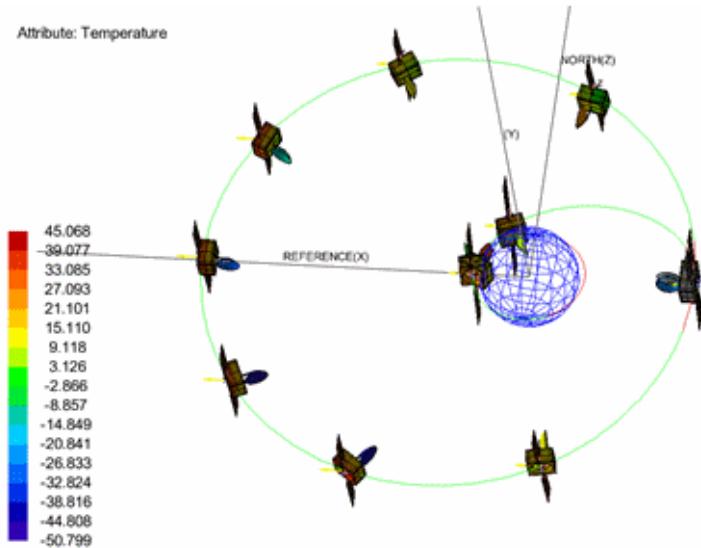


Figure 3-64 Temperature every 7115 s

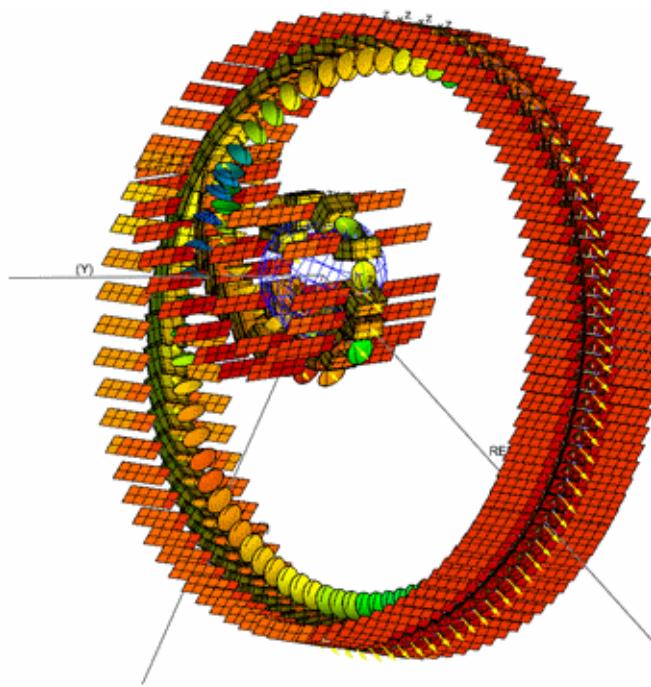


Figure 3-65 More orbit position displayed

In this case the visualisation display has been set every 900 seconds which is very small in comparison to the first analysis. To change and play with these values, open the transfer analysis file from the model tree, select the Solution Control button and change the value in the Execution Block for OUTINT as shown in Figure 3-66 and click OK then Apply.

```

C
NLOOP=100
RELXCA=0.01
CALL SOLVFM
C
C
NLOOP=100
RELXCA=0.01
STEFAN=5.6688E-8
TIME0=0.0
TIMEEND=71155.552202
OUTINT=900.5555220
DTIMEI=1
CALL SLCRNC
C

```

Figure 3-66 Template file modification

Well done! This is how to perform a thermal analysis with ESATAN-TMS. Do not hesitate to play with the DTIMEI value to assess the accuracy of your model.

Note: When the Analysis Case has been run at least one time, do not hesitate to open the thermal output file (.out file) in order to check the convergence (energy balance is very close to 0 and ENBALR smaller than ENBALA). Then, for the transient analysis, check the CSGMIN value. If this value is lower than the defined time step (DTIMEI), modify DTIMEI to be smaller than CSGMIN. CSGMIN is the minimum time constant of the model, therefore, if this value is lower than the imposed time step, the accuracy of the predicted temperature could be reduced.

3.6.2 Post-Process using charts

The thermal results can also be post-processed using charts. For this example we shall create the following 3 charts:

1. The temperature of the 3 units
2. The solar panel temperature
3. Heat flow from the antenna to the cold finger

3.6.2.1 Temperature of the three units (Attribute Chart - Line Chart)

- Open the Analysis Case transfer and create a new attribute chart using the results MODEL_2_TRANSFER.TMD2.

- From the model tree, select the geometry Unit_1, Unit_2 and Unit_3 (use Ctrl for multiple selection) and add to the existing chart using the right-click menu option Add To Existing Chart.
- Rename the chart Unit Temperature and set the range axis label Temperature[C]

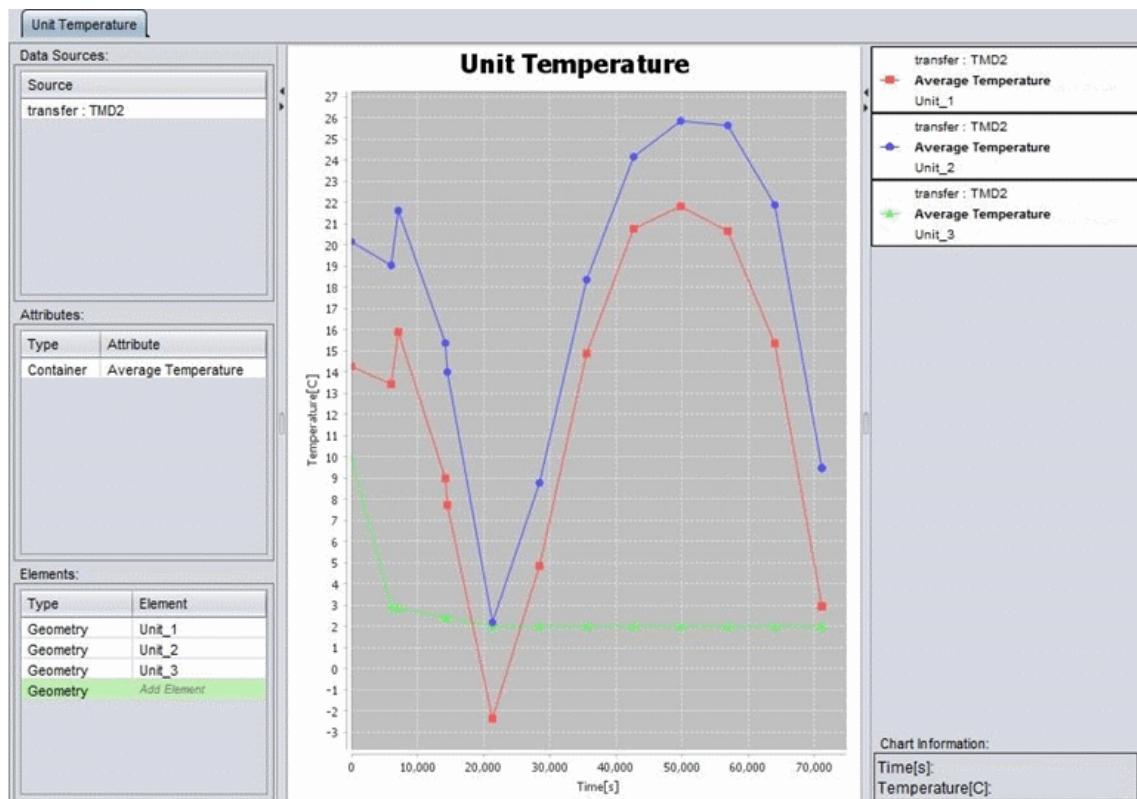


Figure 3-67 Unit temperature chart

3.6.2.2 Solar panel temperature (Attribute Chart - Line Chart)

- Create a new chart by selecting the Attribute Chart from the Post-Process ribbon bar, or by right-clicking on the Chart item in the model tree and selecting New Chart → Attribute Chart.
- Select the Data Source by clicking Data Source Selection from the Post-Process ribbon bar and selecting the Attribute Chart just created. Select the data source transfer: TMD2.
- In the model tree, select the two solar panels Solar_Panel_1 and Solar_Panel_2 and add to the new chart using the right-click menu option Add To Existing Chart.
- Rename the chart Solar Panel Temperature and the Range axis label Temperature[C].

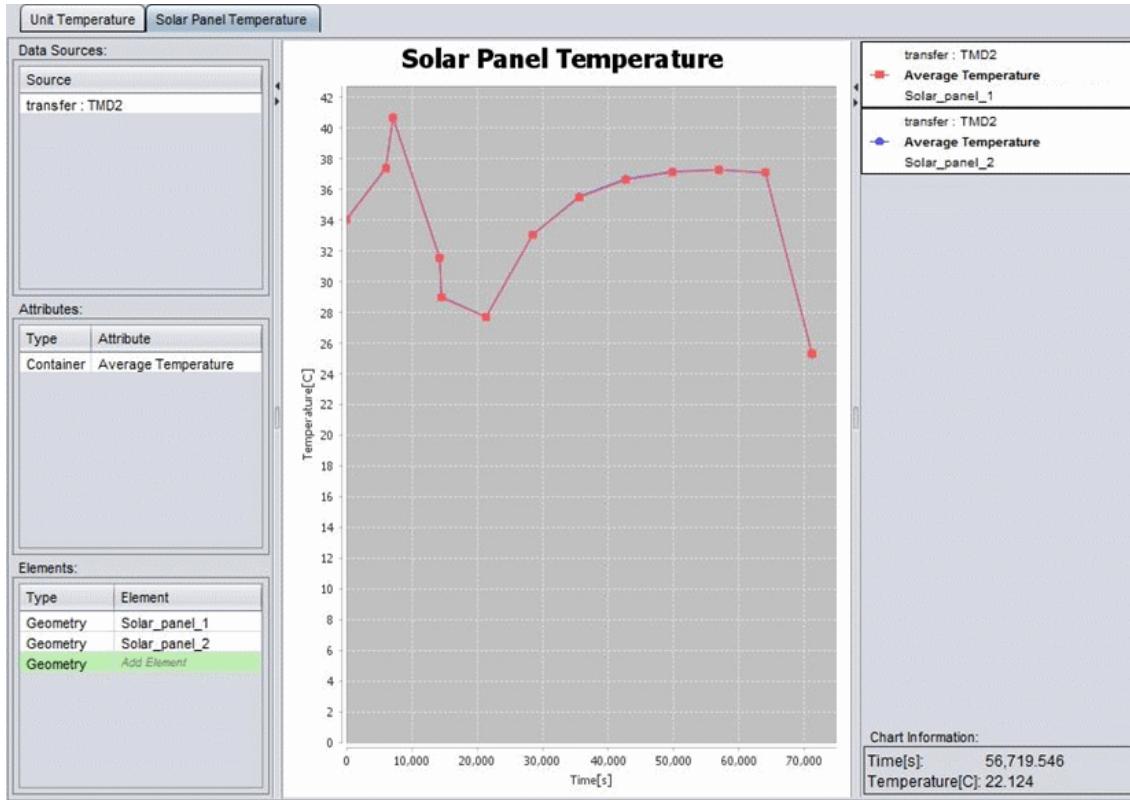


Figure 3-68 Solar panel temperatures chart

3.6.2.3 Heat flow from the antenna to the cold finger (Heat Chart)

- Open the Analysis Case transfer and create a new Heat Chart using the results MODEL_2_TRANSFER.TMD2.
- Select the Double Chart layout option from the Post-Process ribbon bar.
- Drag the new chart to the new chart area (lower right-hand side).
- Hide the geometry Antenna_2, Antenna_Connection and Base_Panel so you can access the Cold_Finger geometry in the visualisation.
- Ensure the visualisation picking mode is Face and select the face with the User Defined Conductor connecting the Antenna, then add the node representing the face by selecting the right-click option Add To Existing Chart.
- Set the picking mode to Surface, select and add the Antenna surface (surface1) to the Heat Chart.

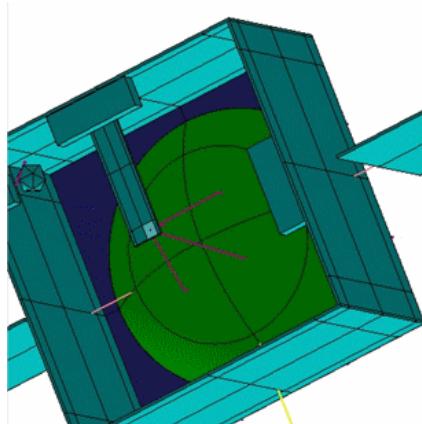


Figure 3-69 Heat flow from antenna to cold finger

- Ensure that only the attribute “Total Heat Flow” is selected (the default).
- Rename the chart Cold_finger. Heat Flow and set the Range axis label to Power[W].

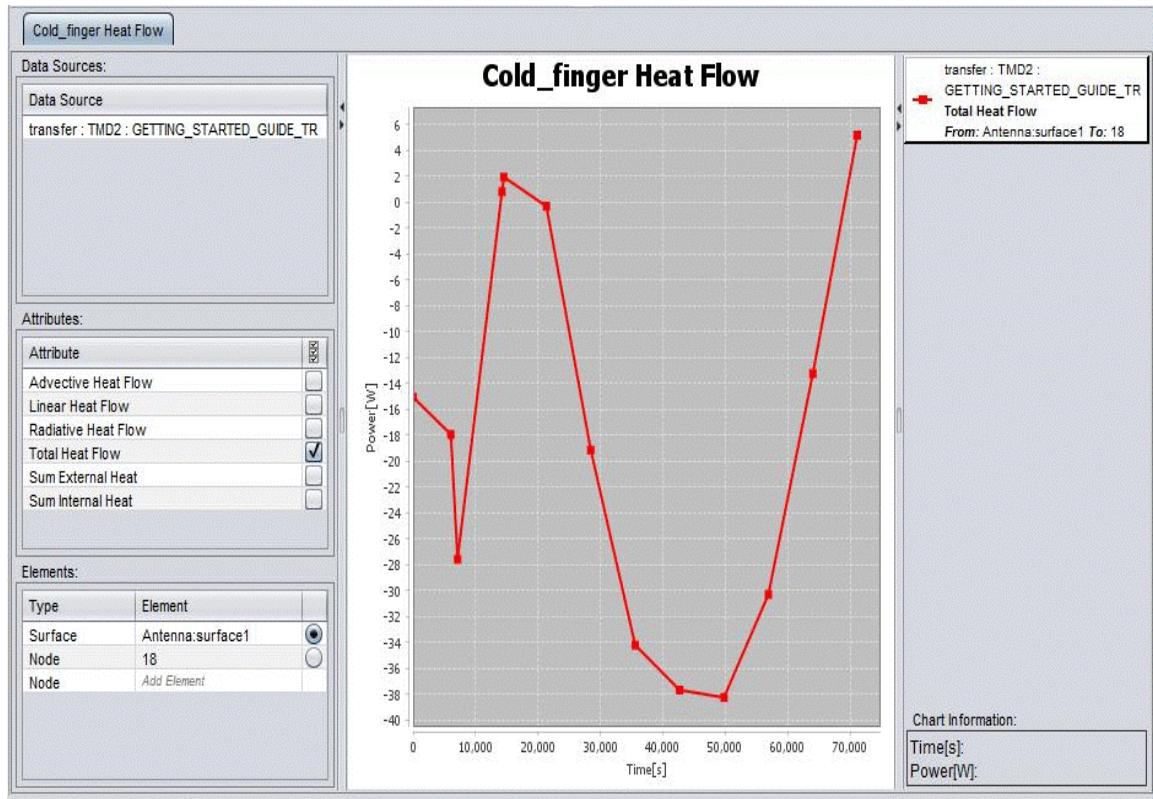


Figure 3-70 Cold_Finger heat flow chart