

AN IDIOT'S INCOMPLETE GUIDE TO THERMAL DESKTOP

Small Satellite Research lab
The University of Georgia

Version 2.2

Casper Versteeg

June 17, 2018

Contents

1	Introduction	3
2	Creating Models	3
3	Defining Material Properties	4
4	Radiation Analysis Groups	6
5	Meshing	6
6	Model Checks	9
7	Conduction Between Parts	10
8	Orbit Creation	12
9	Case Set Manager	13
10	Results	16
11	If TD Breaks	18

List of Figures

1	Initial CAD Drawing Setup	3
2	Thermophysical Properties Menu	4
3	Optical Properties Menu	5
4	Radiation Analysis Groups Dialog	6
5	Mesh Controller and Parameters Menu	7
6	Thin Shell Data	8
7	Solid Element Attributes Menu	8
8	Finalized Mesh	9
9	Calculate Mass Results	10
10	Contactors Properties	11
11	Orbit Manager and Properties Windows	12
12	Case Set Manager	14

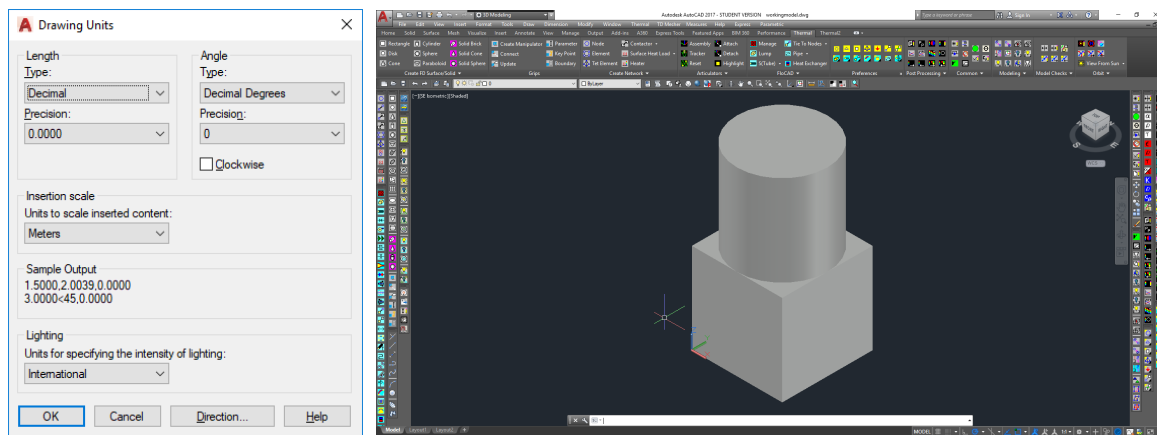
13	Case Set Editor	14
14	Output Tab	15
15	Run Status Completion	16
16	Final Results	16
17	EZXY Plotter Window	17

1 Introduction

This guide is meant to help those completely new to Thermal Desktop (TD) get acquainted with its major features and get started on building a project. By no means is this guide complete (hence the "incomplete guide") but although it skips over a lot of features, it should be comprehensive enough to give you the knowledge required to figure it out.

2 Creating Models

Because TD is an extension of AutoCAD, it allows you to work with existing models and drawings to create temperature profiles. Important to remember is that to run models in TD, a finite element mesh has to be created from your model down the line. In order to keep the size of this mesh manageable and speed up simulations, it's worth considering removing certain details from models that have little impact on the temperature distribution (i.e. small holes, extensions, etc.). For the purpose of this guide, we will be building a working model to help illustrate some of the features you will likely be using in TD.



(a) Unit Setup Definition

(b) Working Model

Figure 1: Initial CAD Drawing Setup

Before we get started, some unit definitions have to be specified in AutoCAD. As it turns out, TD has a tendency to ignore the units set in AutoCAD, so it pays off to create any drawings with the correct units in mind. In the TD preferences, the

units default to SI, meaning length units are meters. To avoid any headache down the line, it is recommended to make all drawings in terms of meters. This can be done by typing `units` in the command line. Figure 1a shows the units dialog.

Figure 1b above shows a simple $5 \times 5 \times 5$ cm cube, with a 5 cm high cylinder on top of it. Note the orientation of the coordinate system, this will be important in defining the orientation in orbit.

3 Defining Material Properties

Before any of the really important stuff happens, thermophysical and optical properties have to be defined as part of the setup. To define thermophysical data, go to `Thermal > Common > Thermo Property Manager`, or to the drop-down menu on top `Thermal > Thermophysical Properties > Edit Property Data...`

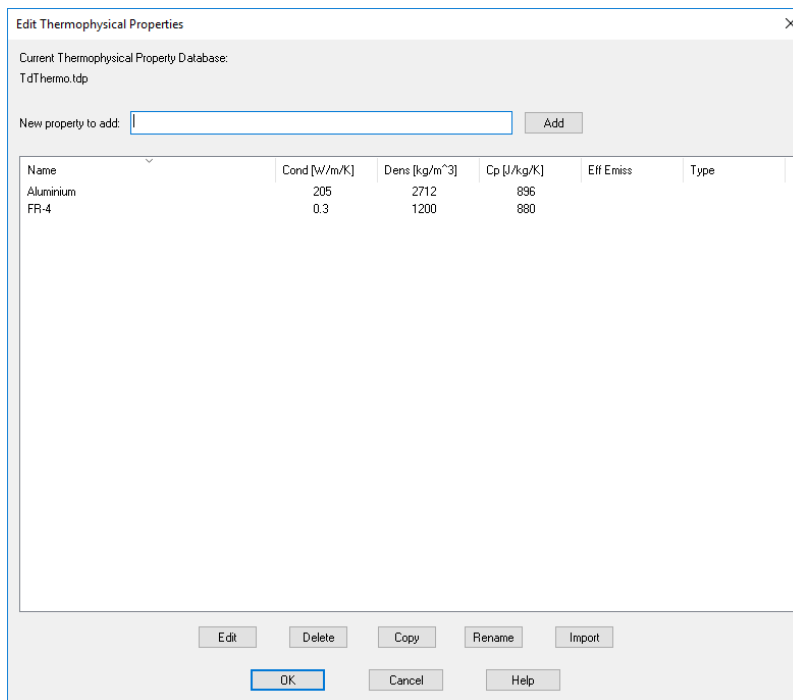


Figure 2: Thermophysical Properties Menu

This menu, as shown in Figure 2, displays all of the materials for use in the model. As many materials as needed can be created by typing a desired name in the "New property to add:" field, and hitting "add." This will open a new dialogue

where all the thermophysical characteristics of the material can be defined. For the purpose of this demonstration, we will define a material named Aluminium, with thermal conductivity $k = 205 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, specific heat $C_p = 896 \text{ J} \cdot \text{kg}^{-1} \cdot \text{K}^{-1}$, and density $\rho = 2712 \text{ kg} \cdot \text{m}^{-3}$. The remaining fields will be left as is. We will also define the common PCB material FR-4, whose properties are $k = 0.3 \text{ W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$, $C_p = 880 \text{ J} \cdot \text{kg}^{-1} \cdot \text{K}^{-1}$, and $\rho = 1200 \text{ kg} \cdot \text{m}^{-3}$.

Alternatively, if desired, an existing database can be imported by hitting the "Import" button. A database, in this case, is file created by TD when thermophysical properties are added. This file is named `TdThermo.tdp`, by default, and is editable with any simple text-editor like notepad.

Up next we have to define the optical properties for the radiation calculations. The menu for this can be found under the tab `Thermal > Common > Optical Property Manager`, or under the drop-down `Thermal > Optical Properties > Edit Property Data...`

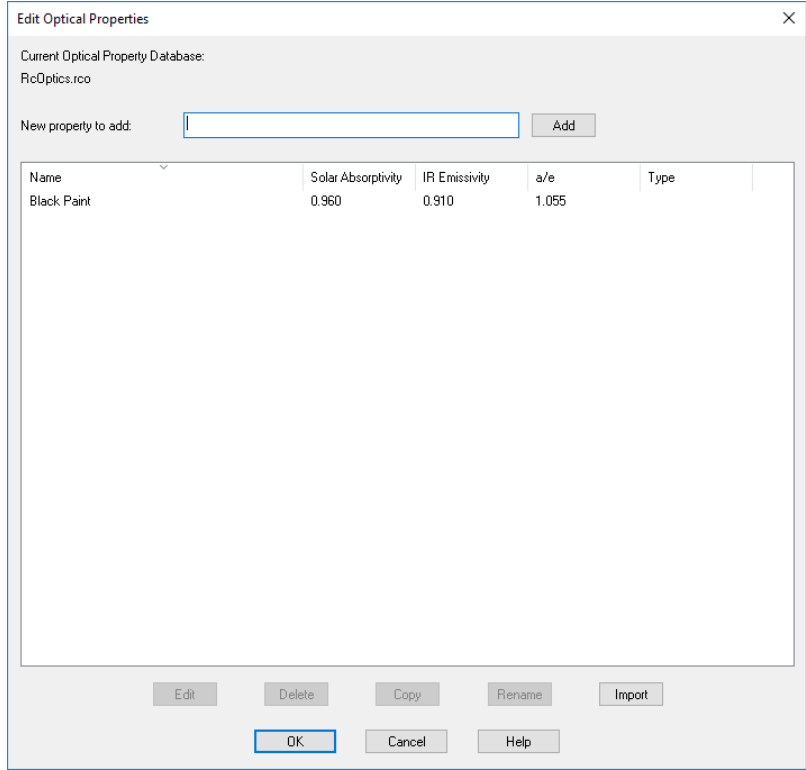


Figure 3: Optical Properties Menu

The general gist of adding surface finishes works the same as for thermophysical

properties, as does importing existing databases, which are called `RcOptics.rco`. For the purpose of this guide, we will define a single surface finish called "black paint," but as many finishes can be defined as necessary. We will define this finish to have an absorptivity $\alpha = 0.96$, and emissivity $\varepsilon = 0.91$.

4 Radiation Analysis Groups

To be able to adequately define the calculation tasks later on, we will define Radiation Analysis Groups (RAG), which will also need to be applied to the mesh. The RAG is under `Thermal > Common > Radiation Analysis Groups` or in the drop-down from `Thermal > Radiation Analysis Groups...`, which will open the dialog as shown in Figure 4.

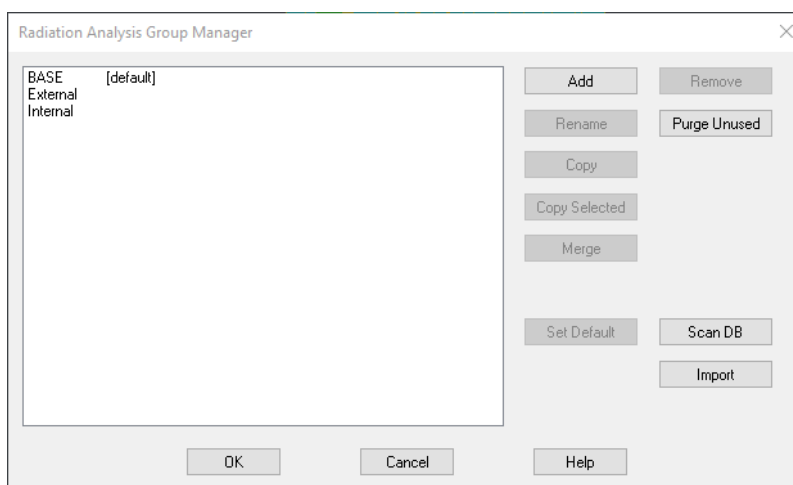


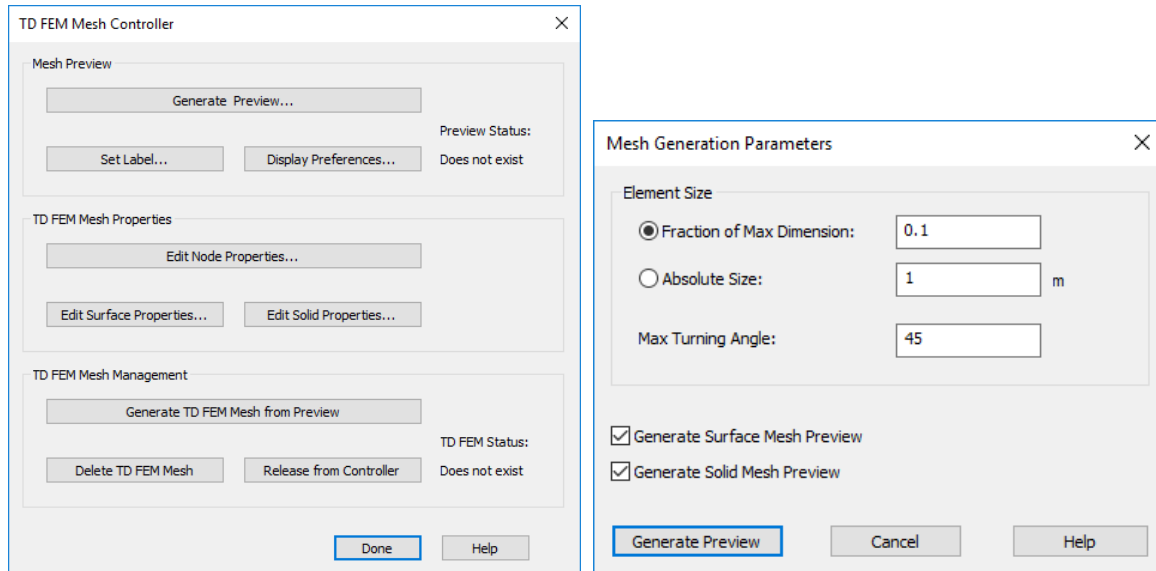
Figure 4: Radiation Analysis Groups Dialog

We are simply going to add two analysis groups, named "Internal" and "External" which are going to serve as groupings of surface directions (defined by the positive and negative normal).

5 Meshing

TD uses Finite Element Methods (FEM) to solve the node temperatures of the model. Therefore, the next step in the modeling process is to create the FEM mesh. The meshing tool can be accessed from the tab `Thermal2 > Mesher > Mesh`, or from

the drop-down **TD Mesher > Mesh Part**. Using the meshing tool, left-click on any continuous, solid object to bring up the Mesh Controller as shown in Figure 5a.



(a) TD Mesh Controller

(b) Mesh Generation Parameters Menu

Figure 5: Mesh Controller and Parameters Menu

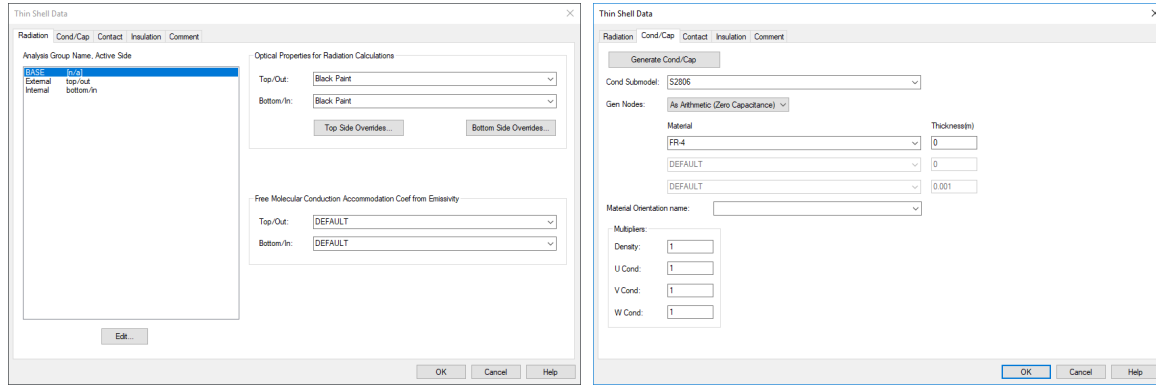
The first button to hit is "Generate Preview..." which will bring up the Mesh Generation Parameters menu (Figure 5b). This menu allows you to control how fine the mesh is. As a rule of thumb, the larger the fraction of max dimension, and the larger the turning angle, the more crude the mesh will be (it should be noted that the fraction of max dimension should be between 0 and 1). It is encouraged to play around with this menu to get a feel for the way TD creates meshes of existing geometry.

Once a preview has been generated, this should be visible on the model as yellow mesh lines, and it should say "valid" under Preview Status on the Mesh Controller.

Up next, we can use the material properties previously defined and apply them to the mesh by hitting "Edit Surface Properties..." which will bring up the Thin Shell Data menu as shown in Figure 6b. In this menu, we will apply the surface property "black paint" we created earlier to all the surfaces under Optical Properties for Radiation Calculations. In most cases we can leave the Free Molecular Conduction Accommodation Coeff from Emissivity as "DEFAULT," which has values $\varepsilon = \alpha = 1$.

Also, on the left side of Figure 6a, make sure that for every submodel the analysis groups are set properly. For consistency in the model, remove the BASE analysis

group from the calculations model, set "External" to Top/Out, and "Internal" to Bottom/In.



(a) Radiation Data

(b) Conduction Data

Figure 6: Thin Shell Data

Next, in the Cond/Cap menu (Figure 6b), we have to change the material to whichever part is being modeled. For this demonstration, the cube block is made of FR-4, and the cylinder is made of aluminium, so we can define those properties accordingly. It is important to also set the thickness of the material to zero, so that no additional material is being added to the model to interfere with the radiation and conduction models. Lastly, set the "Gen Nodes" fields to As Arithmetic. This makes it so that none of the 2D surface nodes participate in the conduction model, and only serve as a surface shell for incident radiation purposes. The rest of the properties in this menu we will leave as default.

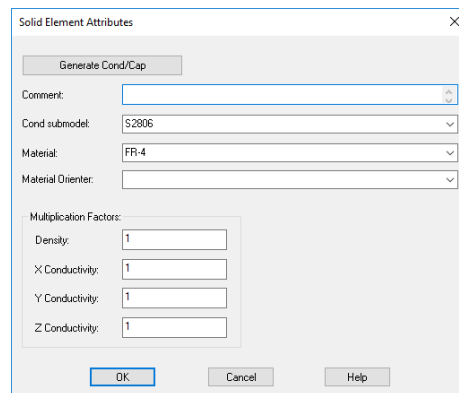


Figure 7: Solid Element Attributes Menu

Closing out of the Thin Shell Data menu, we go to "Edit Solid Properties..." in the Mesh Controller, which brings up the Solid Element Attributes menu in Figure 7. In here we set the Material field to whatever we set the material in Figure 6b, so because these windows show the properties for the cube block they are set to FR-4.

After all this, the only thing that is left to do is generate the actual mesh with the properties to be used by TD. To do this, simply left-click the "Generate TD FEM Mesh from Preview" on the Mesh Controller. The model in the background should come out looking something like what is shown in Figure 8, and the "TD FEM Status:" on the Mesh Controller should turn from "Does not exist" to "valid," much like when we created the preview.

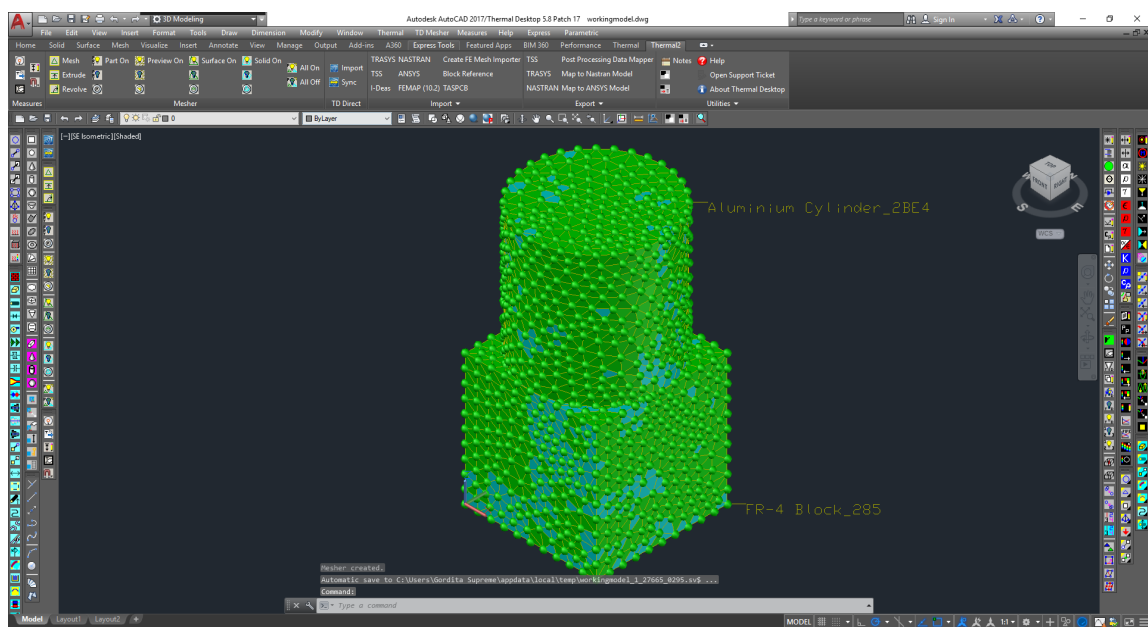


Figure 8: Finalized Mesh

6 Model Checks

Now that the mesh is created and materials properties are set up, this is an excellent time to start talking about model checks. TD has a few options for model checks that can be found both in the ribbon and the drop-down under **Thermal > Model Checks**. For reasons mentioned at the start of this guide, concerning units, the most powerful immediate tool is the mass calculator.

If the units are improperly defined, this will result in a model mass that you would not expect. If, for example, the units on the drawing were set to millimeters, and the dimensions of the block are $50 \times 50 \times 50$ mm, TD is going to completely ignore the drawing units, and assign the dimensions units of meters, thus resulting in a $50 \times 50 \times 50$ m block of FR-4, which will have a mass of several million kilograms. Not great.

This is why the mass calculator model check is such a great tool for this stage. By simply calculating the mass, and looking at the order of magnitude, this can be a great error catching mechanism and save you a lot of scaling-headaches down the line. For this guide, go ahead and click on the mass calculator under **Thermal > Model Checks > Calculate Mass**. Now you can select either a submodel, a grouped layer, or just hit **Enter** to select the entire model. TD will take the geometry and the density of the material assigned to it and calculate the physical mass of the model.

```
Command: _RcCalcMass
Select Objects for Mass Calc <No Objects Does Entire Model> or [MB/GRP]:
Mass for surfaces and solids based on nodal submodel names
Submodel, Surface/SolidMass, Insulation Mass, kg
S2806 0.16800 0.
S296 0.29692 0.
Total Mass For Surfaces and Solids = 0.464917 kg
Total Insulation Mass For Surfaces and Solids = 0.000000 kg
Mass for userDefinedNodes
Total Sum for Nodes is 0.000000 kg
Total Sum of Surfaces/Nodes/Insulation is 0.464917 kg
All nodes written to NodeSummary.xls
Mass Per Conductor Submodel written to MassPerCondSubmodel.xls
```

Figure 9: Calculate Mass Results

As soon as you have run the mass calculator, TD will expand the AutoCAD command terminal, which should look something like Figure 9. If the units are off, a good place to start debugging should be the drawing units, or the material properties.

7 Conduction Between Parts

TD is a pretty powerful piece of software, but unfortunately it is too stupid to realize what surfaces are touching each other for conduction purposes. To tell TD what surfaces are actually touching for conduction purposes, we have to define a Contactor in TD. We can create a contactor under the tab **Thermal > Create Network > Contactor** or under the drop-down **Thermal > TD/FEM Network > Contactor**.

From here, you can select every surface that touches every other surface manually, which is a royal pain and not recommended, or simply type `GRP` in AutoCAD's command line. This will bring up all the layers that currently exist in the model. When you create a mesh, TD creates a bunch of new layers in the model, one of which is the 2D surface of the mesh model. All we have to specify here, is the "from" and "to" contactor surfaces in the model. In this menu, select a 2D surface, hit "enter," and then again to be able to select the second surface. Again type `GRP`, and follow the above instructions. In this process you can select as many submodels as you want at a time.

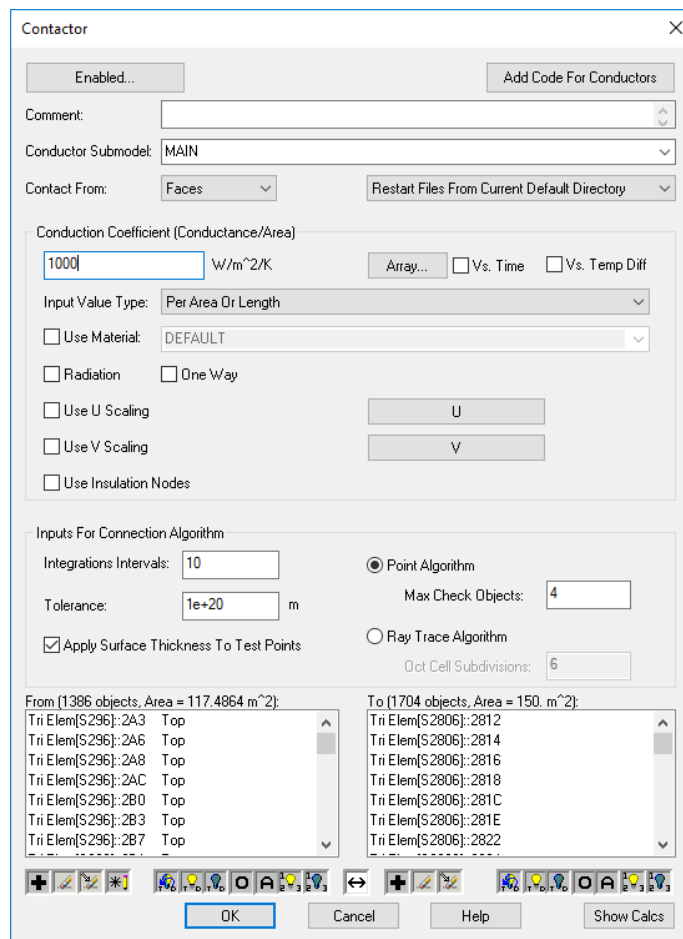


Figure 10: Contactor Properties

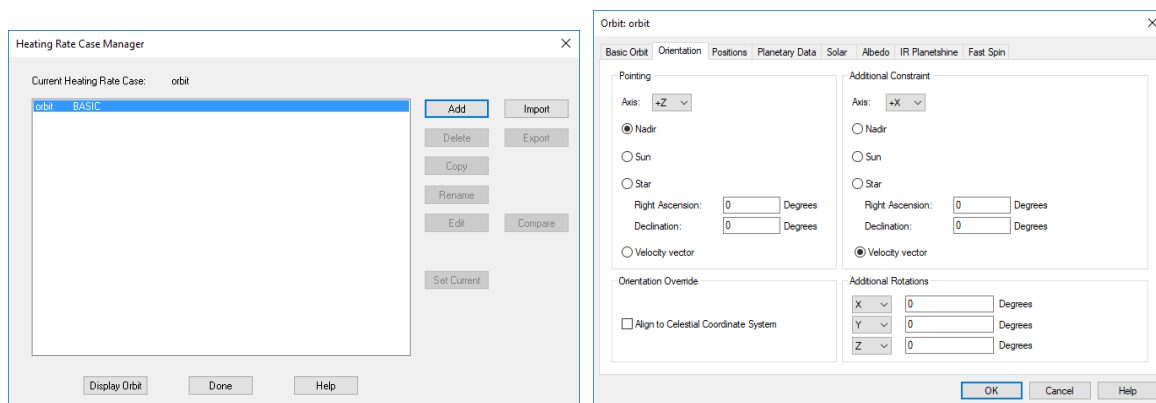
The final step, after all surfaces have been selected, is to set the properties in the

Contactor properties menu as shown in Figure 10. We set the conduction coefficient here to $1000 \text{ W} \cdot \text{m}^{-2} \cdot \text{K}^{-1}$. The conduction coefficient here is not the same as for the material properties (which has units of length), but is rather a function of the contact pressure between parts, and their surface finishes (and has units of area).

A final note on contactors: they are directional. This means that when you specify a surface to contact "from" and a surface to contact "to", TD will only allow heat to "flow" in the from-to direction. If you want a contactor to behave like a proper conduction interface, you have to specify two contactors for each connection. Say object O1 is touching object O2, you must specify a contactor with a conduction coefficient from O1 to O2, but also from O2 to O1. Otherwise heat will only flow from O1 to O2 (or whichever way is specified) but never the other way, which might not be realistic.

8 Orbit Creation

The next step in setup is to create an orbit which defines the heating rates for the model. The Orbit Manager can be found under the tab **Thermal > Orbit > Manage Orbits**, or under the same path in the drop-down menu. Going to the Orbit Manager opens up a window as shown in Figure 11a. Upon hitting "Add" and selecting a basic orbit (which is a circular orbit), the Orbit properties window opens up, where we can edit the physical characteristics of the orbit, the parent body, and the orientation of the model. It is worth taking a minute and perusing all the tabs in this window, and seeing what sort of parameters we can define; it all pretty much speaks for itself.



(a) Orbit Manager

(b) Orbit Properties

Figure 11: Orbit Manager and Properties Windows

The one thing we will be confirming for this demonstration, is the orientation of the model, for which we go to the (surprise, surprise!) Orientation menu. Also looking back to Figure 8 to note the coordinate system orientation, we will make sure that we have the $+Z$ axis pointing Nadir (as to make the aluminium cylinder pointing towards Earth at all times, and making the FR-4 block pointing Zenith), and the $+X$ axis pointing in the direction of the velocity vector. Due to the symmetry of the model, we could also have chosen $+Y$ as pointing with the velocity vector, this would just require adjusting the view angle in the viewport later to have a result that looks the same. Lastly, in the Positions tab, set "increments" to 60, to increase the time-resolution of the simulation. It is worth noting that TD will automatically add extra increments to accommodate for eclipse entering and leaving times.

Once the orbit is created, you can hit the "Display Orbit" button in the Orbit Manager if you wish. This will create a (not to scale) view of the model in the orbit we have just created. Alternatively, press "Done" to proceed.

9 Case Set Manager

The final step in the setup is the Case Set Manager (CSM). The CSM, in a nutshell, initializes the calculations TD will be solving for the model, and how you wish the results to be stored for post-processing. The CSM can be found under the tab **Thermal > Common > Case Set Manager**, or under the drop-down **Thermal > Case Set Manager**.

By default, there will be a "Case Set 0" already predefined by TD (see Figure 12), which we will be editing for this demonstration. In practice, you can add as many different cases as you want, for different kinds of orbits, beta angles, hot and cold cases, you name it.

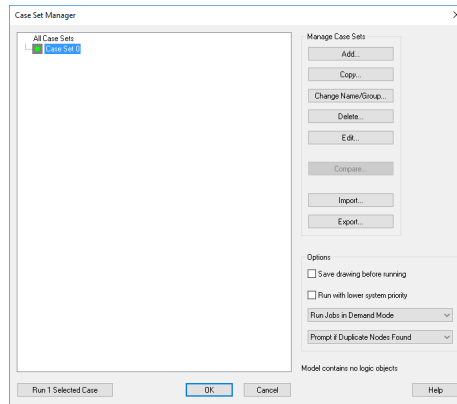
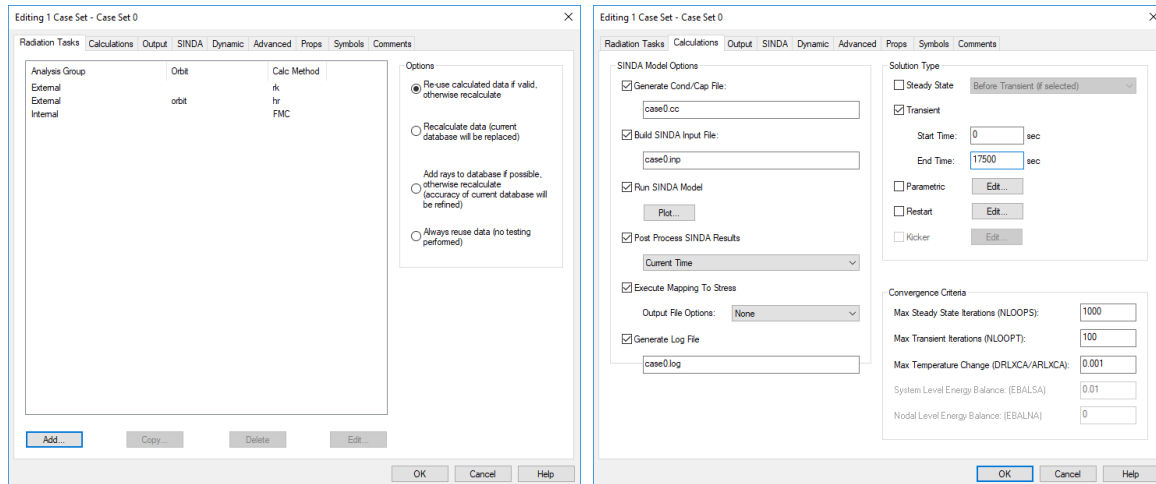


Figure 12: Case Set Manager

Double-clicking, or hitting "Edit..." with Case Set 0 highlighted, will bring up the Case Set Editor, as shown in Figure 13a. This first tab is a bit of a misnomer, as you can also have TD calculate view factors and conduction.



(a) Radiation Tasks

(b) Calculations

Figure 13: Case Set Editor

In the Radiation Tasks tab we are going to be adding a couple of things. Hit "Add..." and select the calculation type "Radks," and set it to analysis group "External." Radk is short for radiation conductance. Leave all the other fields as default, although the same disclaimer applies as before, that it is worth investigating the other tabs and the different options available in TD. In the same way, also add Heating

Rates, and Free Molecular Conduction to "External" and "Internal" analysis groups respectively. The final product should look exactly like Figure 13a.

Because we would like to see the temperature distribution as a function of time in orbit, we are going to un-check the "Steady State" field in "Solution Type" under the Calculations tab, and check "Transient" (Figure 13b). We can leave the start time at 0 (zero) sec, but we will set the end time to 17500 sec, so that we will see behavior for at least three orbits.

Lastly we will be changing one last parameter, which is the solution time-step in the Output tab (Thermal Output Increment). By default, TD will determine this time-step by itself if this parameter is left to zero. However, we will be setting this to 10, as this has a tendency to speed up the solver, although it is a good exercise to experiment with this somewhat to get a feel for the processing time of your model. Now, for the remainder of this demonstration we will not be adjusting any of the other parameters in the Case Set Editor, but do make sure that in the Output tab, under "Output for Color Postprocessing and XY Plots" the "Temperatures" box is ticked, as that is parameter we are most interested in.

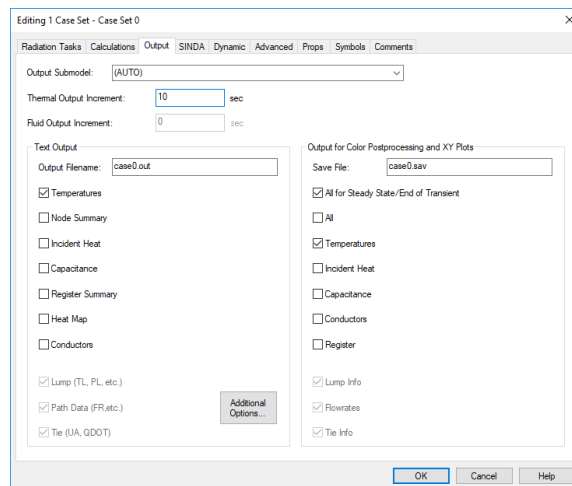


Figure 14: Output Tab

In truth, the CSM is far more extensive than this demonstration will expose you to, and it allows you a lot of control over the simulation. It will allow you to make your simulation incredibly specific for whatever mission you are performing. As such it is definitely worth spending some time with the CSM to get an appreciation for the kinds of things it will allow you to do.

10 Results

Once you have entered all the desired settings in the CSM, press the button in the bottom-left that says "Run x Selected Case(s)." TD will then go and run all of its calculations, take your entire life, and then hope you get a dialog like the one in Figure 15. If you get a different window, good luck fixing that (the last section in this guide is a good place to start, but you might just be S.O.L.).

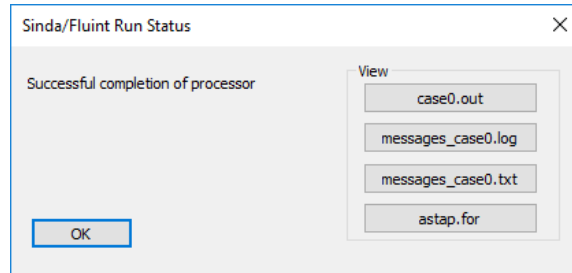


Figure 15: Run Status Completion

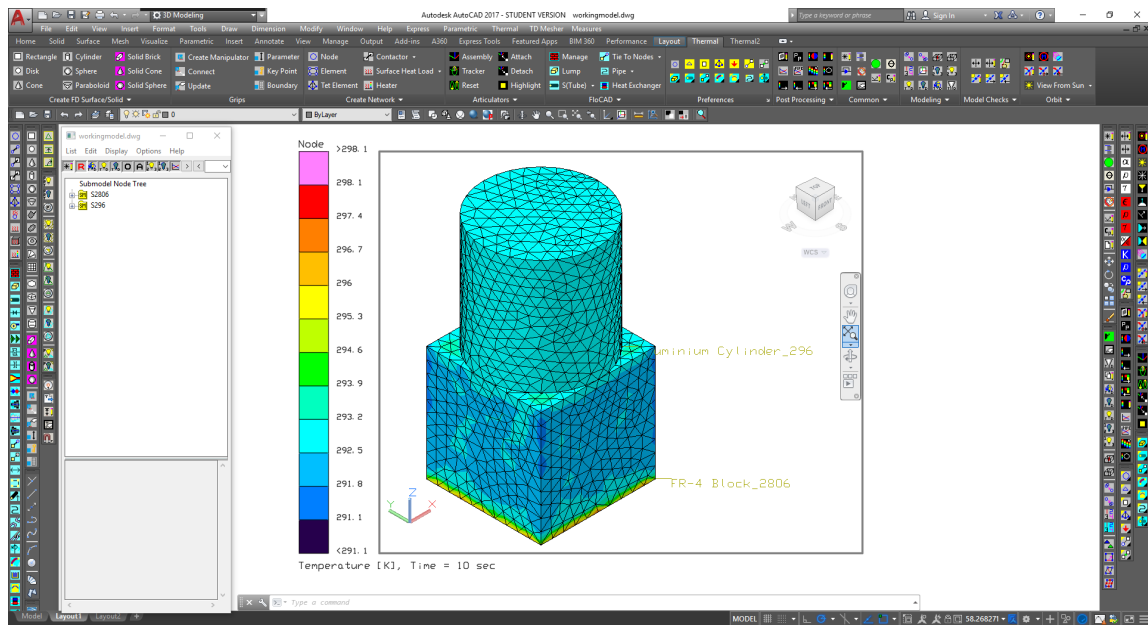


Figure 16: Final Results

Upon completion, you can go to Layout1, and TD should have created a layout

similar to what is shown in Figure 16. Here, you can use the cycle through time buttons to show the temperature profile at each of the time increments defined in the orbit.

Alternatively, in the Model Browser, you can right click on any subassembly, and then click "XY Plot," to have TD create a temperature Vs. time graph of each node. Figure 17 shows the temperature plots of the aluminium cylinder (left) and the FR-4 block (right), and perfectly illustrates the difference in behavior we expect to see based on a material's conductivity. Because aluminium is very thermally conductive, the entire subassembly reaches "uniform" temperature relatively quickly, which can be observed from all the plot lines being so close together. On the other hand, FR-4 is thermally insulating, so it takes a lot longer for the temperature profile to become uniform as the heating rates change, resulting in a wide band of temperature ranges for the submodel.

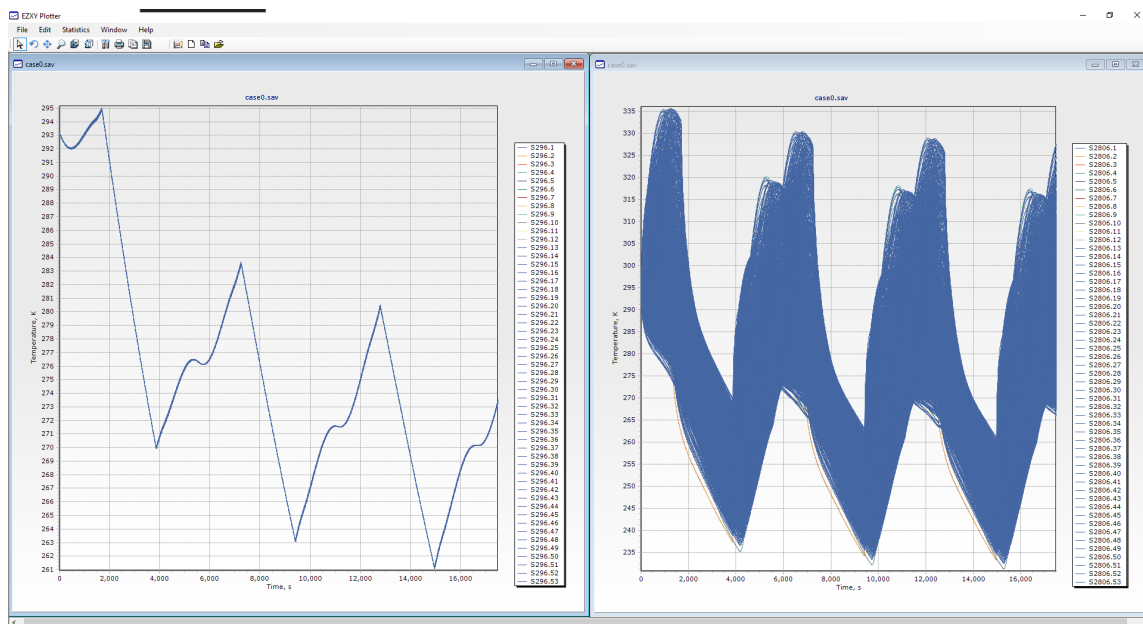


Figure 17: EZXY Plotter Window

Some final words on this guide, it is titled "Incomplete" for a reason. Thermal Desktop has a wide variety of options and settings available to make the thermal model as exact to your specifications and mission parameters as possible. By no means did this guide touch upon all of them, it is merely to serve as a tool for new users to learn how obtain correct results, and troubleshoot and diagnose problems. There is no substitute for experience and a sense of intuition when it comes to thermal

models, and as such it may be a good idea to have your thermal results checked by someone who has more experience in the field of thermal engineering.

Obviously, for more advanced features and issues that this guide does not touch on, the Thermal Desktop Manual should be consulted first, and as a last resort their tech support can be useful.

11 If TD Breaks

If you have worked with Thermal Desktop for a while, you may have noticed it has a tendency to break. Whether it crashes during the modeling phase, or gives errors when the model has finished running, something will go wrong in your time using TD. While the points of failure may be numerous, here are a few things to try when it does break.

Firstly, an explanation on how TD works. Thermal Desktop is merely a plugin to AutoCAD. It serves as a graphical user interface (GUI) to the finite element solver SINDA. You could program SINDA do solve your models by hand, but manually inserting thousands upon thousands of nodes will make you hate your life pretty quickly. TD basically automates this process, so whenever you mesh a part, it dynamically writes a set of instructions to the SINDA solver that allows it to use the model you developed. SINDA is incredibly powerful, but unfortunately runs on a very old programming language called Fortran. Fortran is incredibly fast at doing matrix operations (which is essentially what finite element models are), but can be a pain in the neck to get running on your PC.

Before you do anything—and I cannot believe I have to say this—try restarting the computer. During the time a computer is on and used by various people, background processes accumulate and take up valuable space you need to run your models. By restarting the computer you will clean up the background tasks to some extent. You can also try going into the task manager and manually closing some things that are taking up RAM or disk-read/write space.

The software you need to run the SSRL's version of SINDA successfully, are Microsoft Visual Studio 2016 and a compatible Fortran environment. Unfortunately, MS Visual Studio 2016 is completely deprecated, and you cannot download this anymore. So instead, you will need the 2017 version. However, the Intel Visual Fortran (IVF) (the Fortran environment you will need) version compatibility becomes an issue here. IVF 17.0 is made to run with Visual Studio 2016. The earliest version of IVF 17.0 that is compatible with Visual Studio 2017 is part of the **Intel Parallel Studio XE Version 2017 Update 5**, so you will have to install this version, and

make sure it never updates or things will break.

Having these pieces in place should make it so SINDA functions correctly, but nothing is ever guaranteed. For example, the issue that caused me to write Version 2.2 of this guide is truly a silly one. Unfortunately, all of the software mentioned above has some license associated with it somewhere. If any license stops working or expires, the entire thing will come crumbling down. So before you freak out and re-install everything (this will probably reveal the issue that's causing TD to crash, but is a huge time-sink, so consider this the nuclear option), make sure you check all the licenses of involved software. This means opening Microsoft Visual Studio, the Intel license manager, and AutoCAD, to make sure all licenses are up-to-date and working.

In some instances, AutoCAD or TD will tell you in the AutoCAD command line that there is an issue with some registry keys that will need to be updated. In the start menu, type `regedit`, and navigate to whatever registry key TD is telling you will need to be changed. In most cases changing the registry is harmless, but make sure you do not break any internal Windows processes by changing the wrong one.

If all else fails, you can try re-installing TD. It might be worth completely re-installing the computer on which TD is located to make sure all of the variables changed since the last fresh re-install are also reset. In the event this happens, consider making an image of the computer with TD in a working state so it can be reset the next time it breaks, and potentially saving you time in having to completely re-install everything again in the future.

Good luck. You're going to need it.