Simcenter Flotherm Transient Modeling Update Tutorial for v2019.2

Devi Prasad Gorrepati

Last Modified: 30 March 2020

CONTENTS

INTRODUCTION

This tutorial has been created to show the steps required to set up a transient model and then how results might be viewed in the Analyze Results.

The tutorial assumes that you fairly familiar with Simcenter Flotherm and its menus and know how to create simple geometry and apply attributes. The model consists of only two cuboids of different materials and with a periodic heat load to one of them. The model is not intended to represent any real world application but merely to explain the functionality of the software.

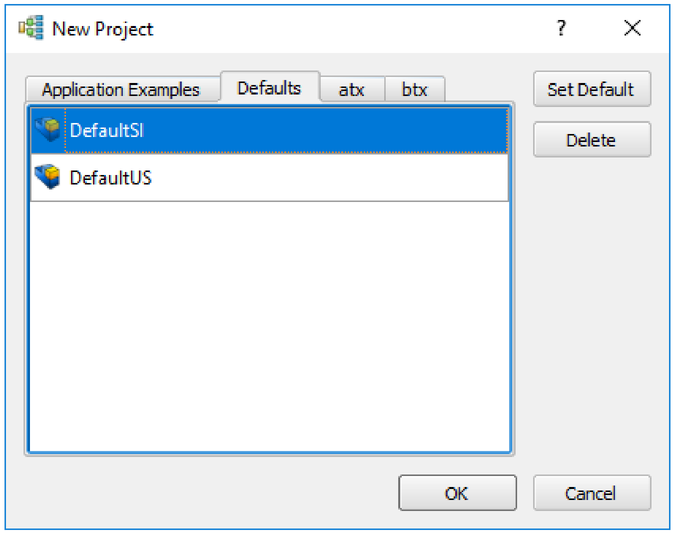
FUNCTION AND LIMITATION

Transient profiles can be applied to source attributes, thermal attributes and ambient attributes for temperature and radiant temperature.

Solar radiation cannot have a transient profile applied and remains a steady state calculation. Please contact your local support for advice on how to model a transient solar radiation application.

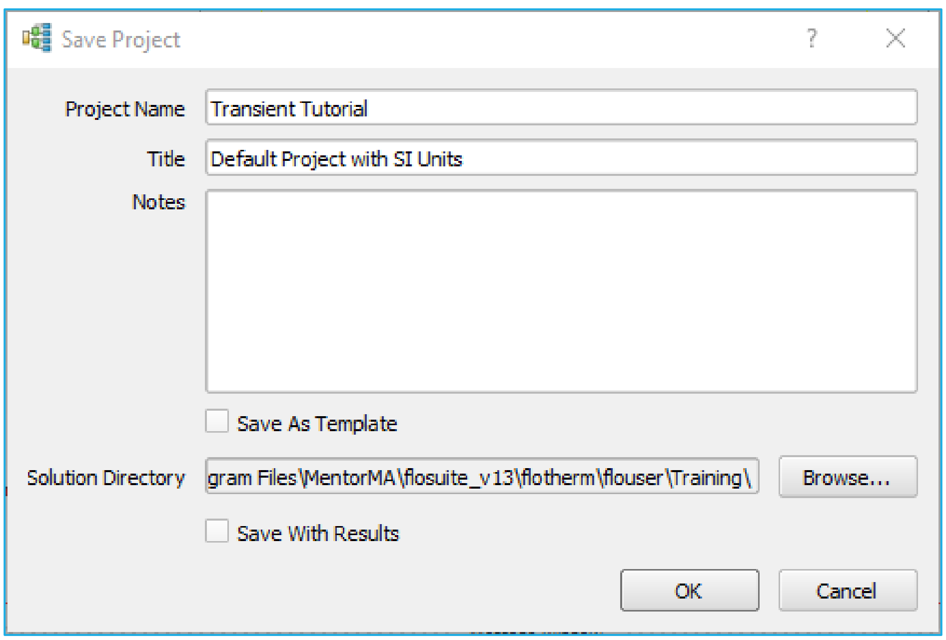
TUTORIAL EXAMPLE

Start Simcenter Flotherm. If Flotherm has already been started, **select** [Project/New]. **Load** the Default SI project.



New Project Dialogue

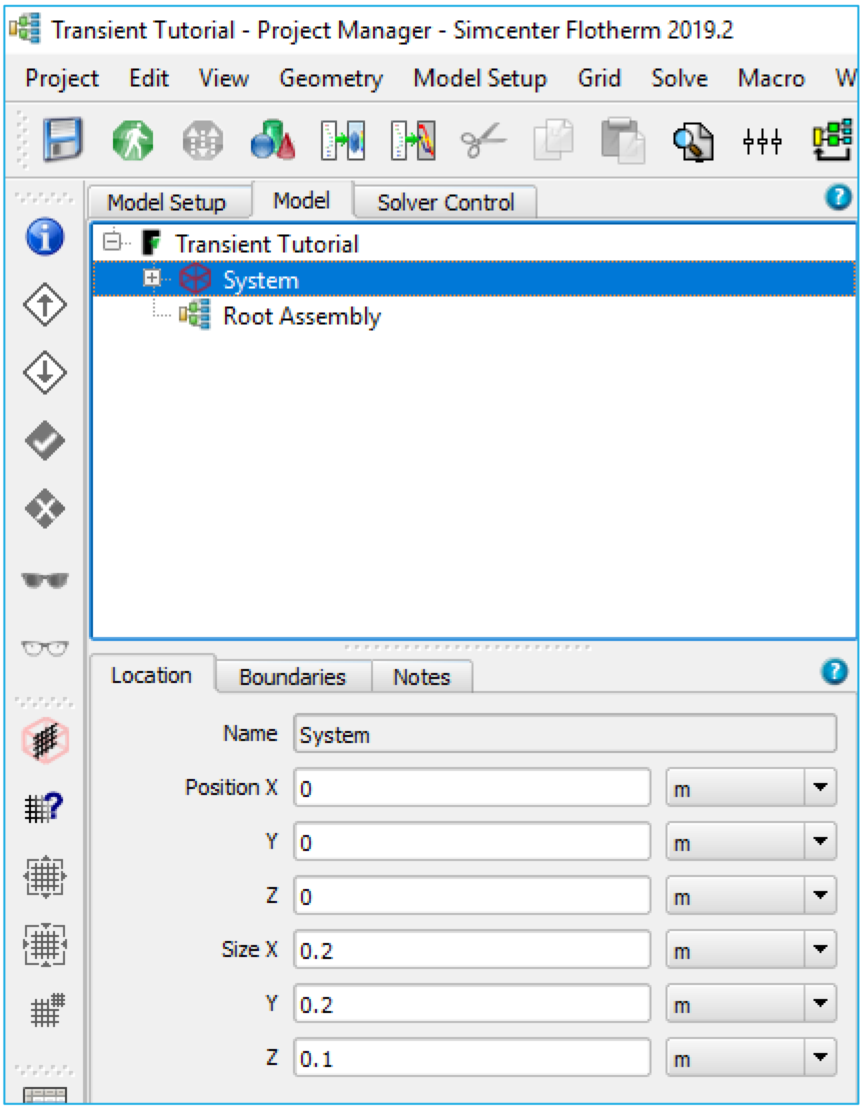
Use Save As to **save** the new project with the name "Transient Tutorial".



Save Project window

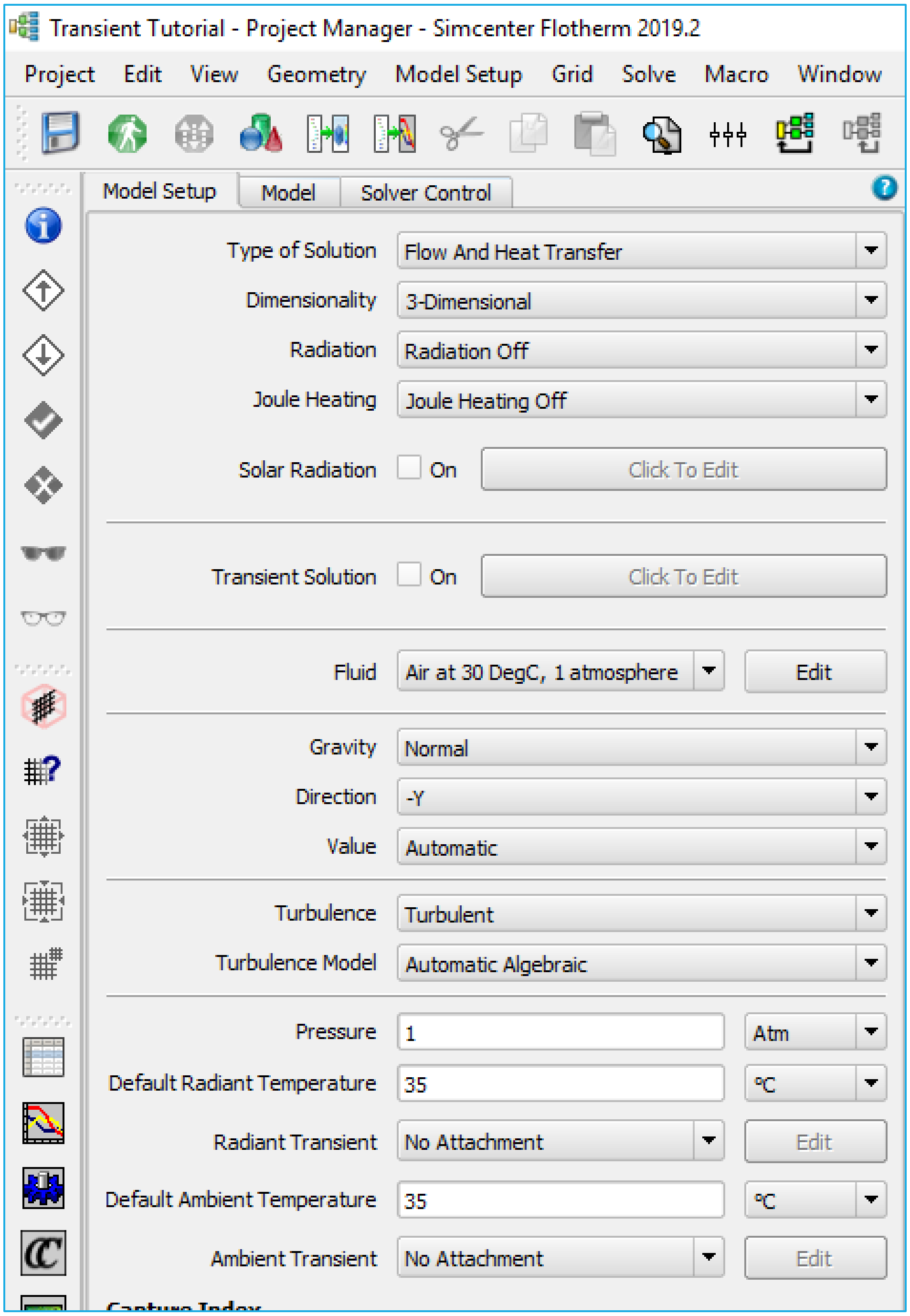
In the Project Manager window **click** on System in the Model Tab. In the property menu below **select** the Location tab and **change** the size of the overall solution domain to: X=0.2m, Y=0.2m, Z=0.1m

The position of the domain should remain as (0, 0, 0) m



System Domain

Attach an ambient attribute to the system. **Click** on the "Model Setup" tab; **Enter** 35C in the field for "Default Ambient Temperature"

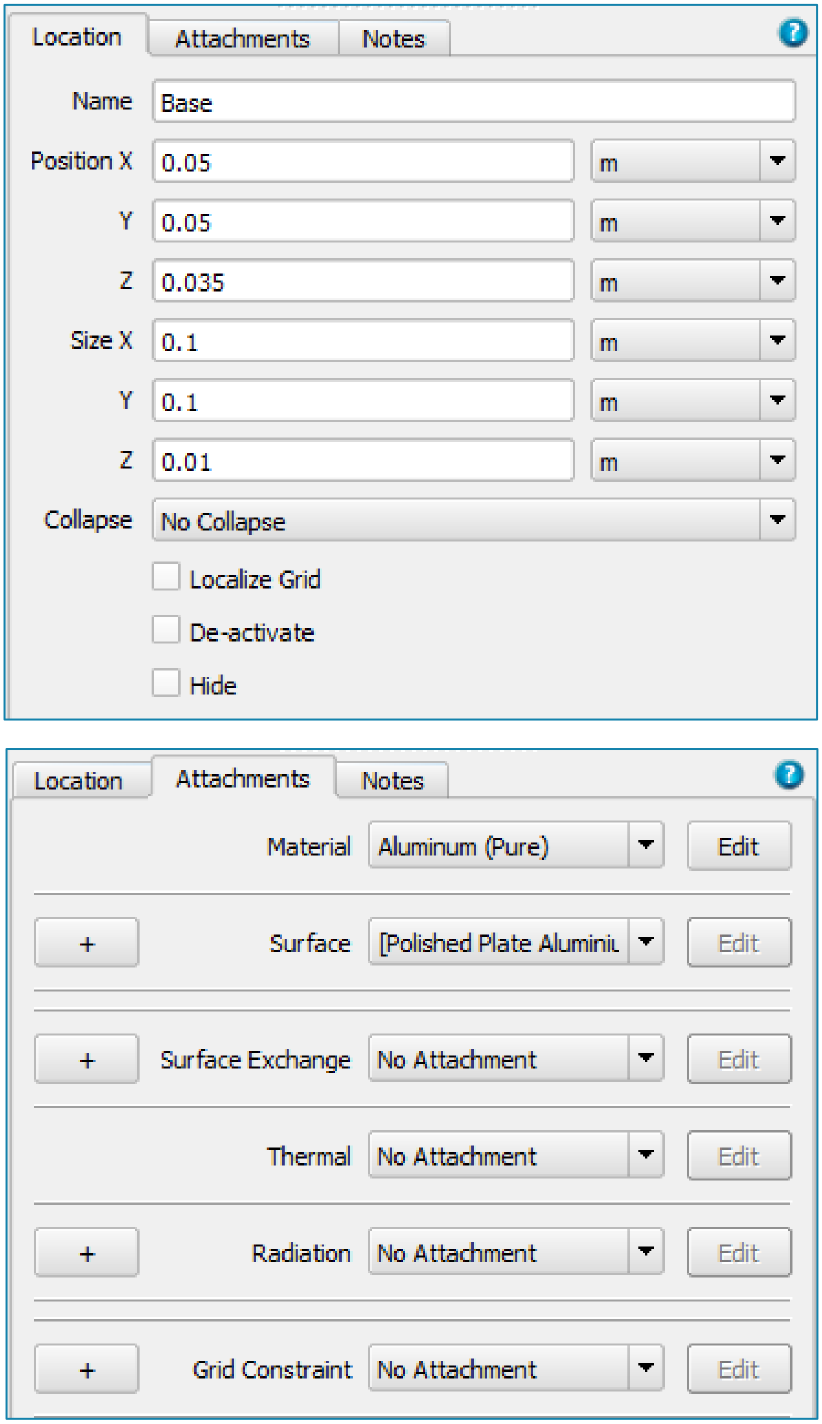


Ambient Setting

Using your preferred method **create** a cuboid with the following specifications:

Location (0.05, 0.05, 0.035) m; Size (0.1, 0.1, 0.01) m

Name the cuboid "Base" in the property menu.
Set the material of the cuboid to Aluminum (pure)

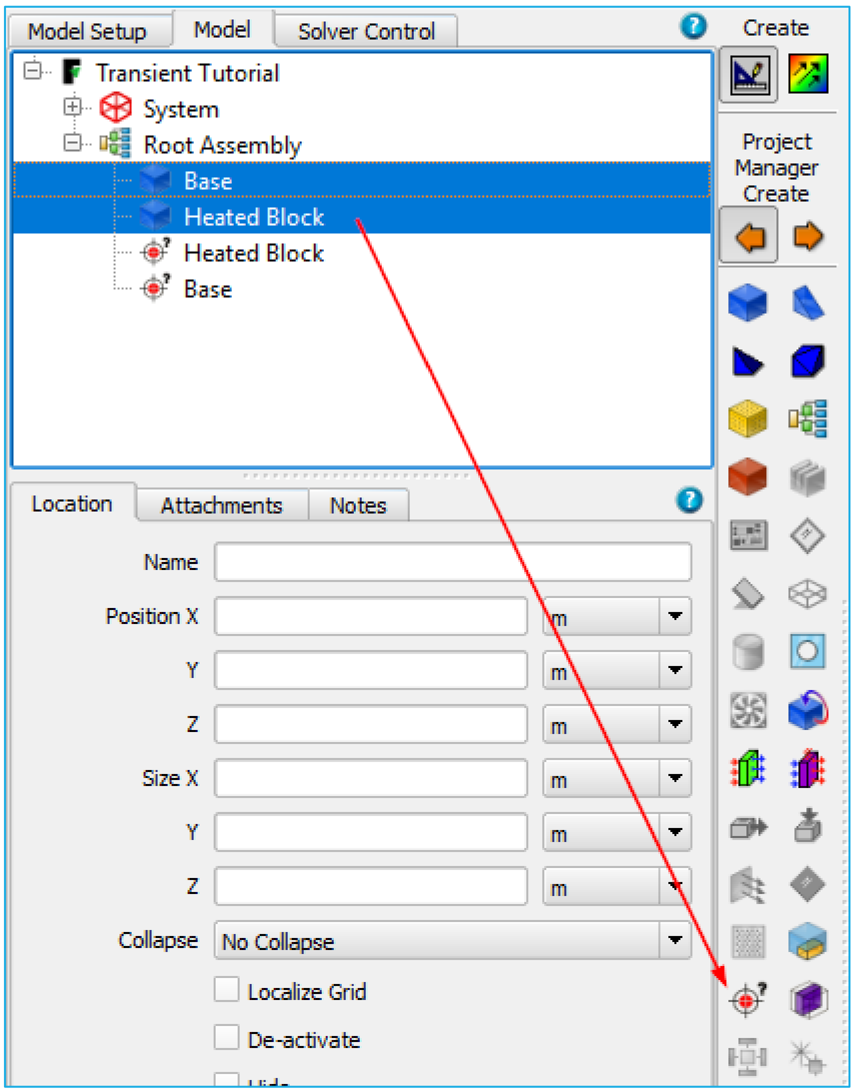


Base Cuboid

Then **create** a second cuboid with specifications:
Location (0.09, 0.09, 0.045) m; Size (0.02, 0.02, 0.02) m Material: Copper (pure)

Name the cuboid "Heated block".

Create monitor points for each cuboid by **highlighting** them in the Project Manager and **clicking** the monitor point icon.

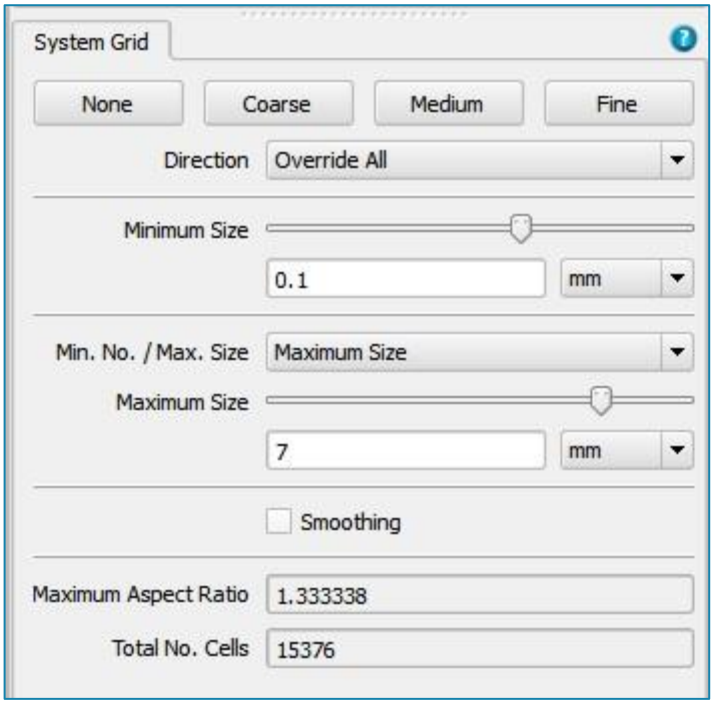


Monitor point

You can now check the grid settings.

In Project Manager **go to** [Grid/System Grid], or **click** the grid icon, .

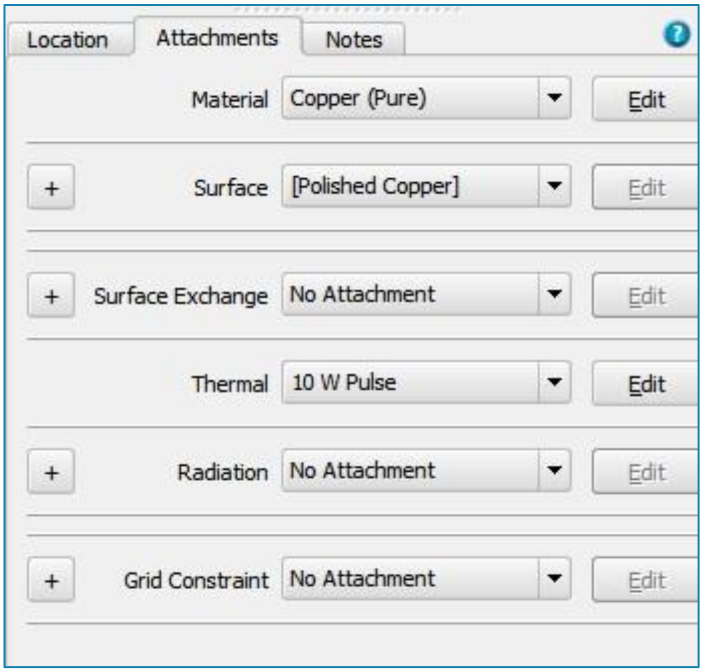
Make sure "Override All" in the drop down menu for "Direction" is selected and **set** the minimum size to 0.1mm and the maximum size to 7mm.

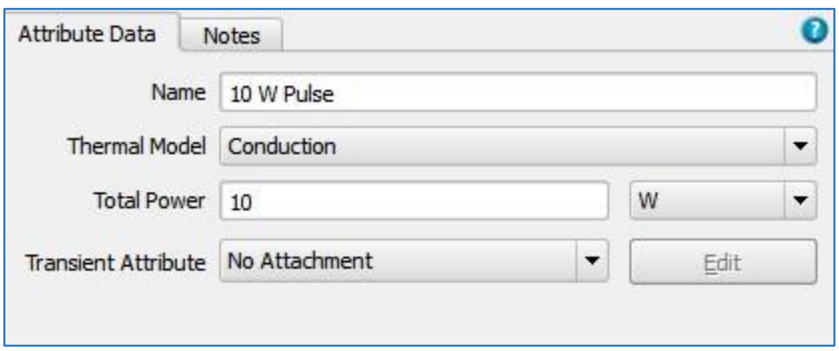


Grid Settings

The second cuboid will be heated via a pulse of 10W for 1s, and then no heat for 1s repeated over 30s for the transient model.

To set the pulsing heat function, **Select** the heated block in the project tree and **click** on "Attachment" tab in the property menu. **Create** a new thermal attribute named "10W pulse". Keep the default conduction thermal model and **set** the total power to 10W.





Thermal Attribute

This would give a steady input of 10W, to set up the pulsing behavior we use the transient profile.

In the “Transient Attribute” drop down list, shown in Figure 7, **create** a new transient profile called “pulse”. **Use** the function option and **create** a new sub-function called “pulse 0-2s”. **Choose** the pulse type from the drop down menu for Sub-Function Type. **Set** the “Start Time” to 0S and “End time” to 2S, “Amplitude” to 1. Leave “Rise Time” and “Fall Time” at 0s, and **change** the “High Time” to 1s. **Switch** on the Periodic function so the pulse is repeated over the whole transient time period.

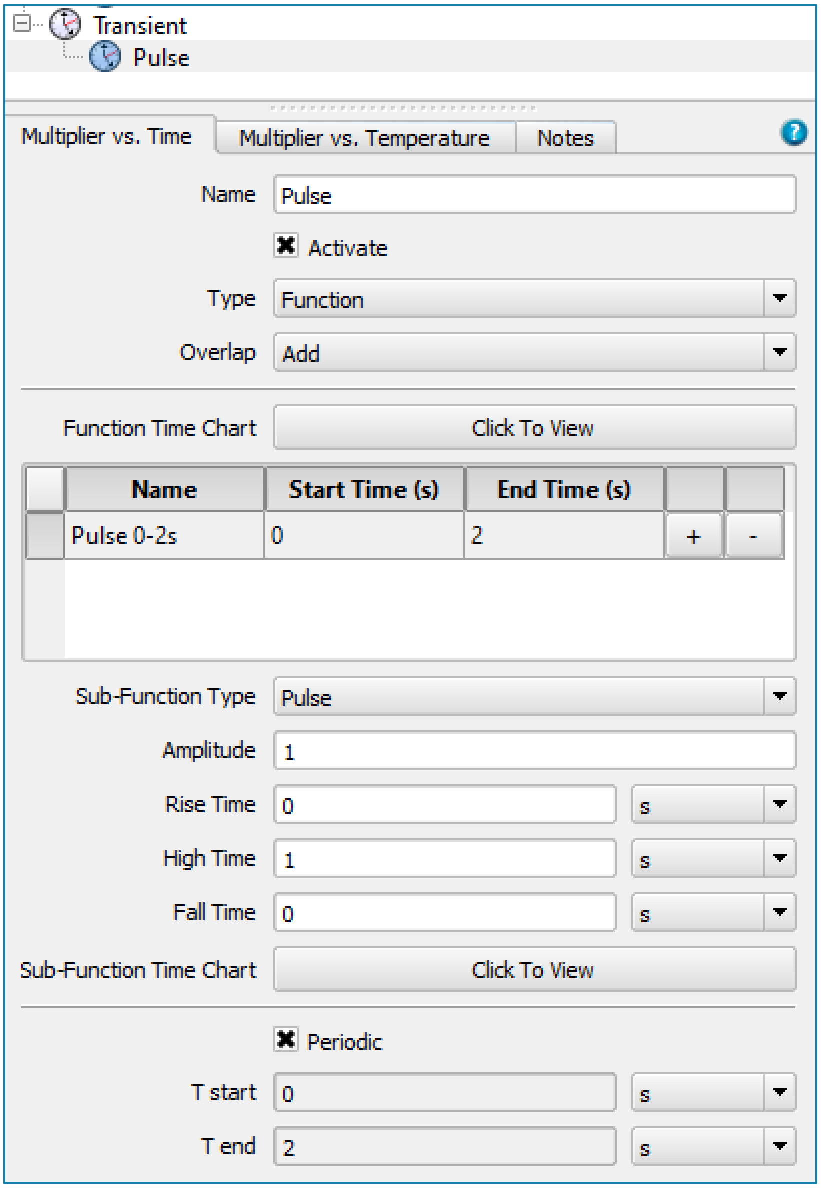
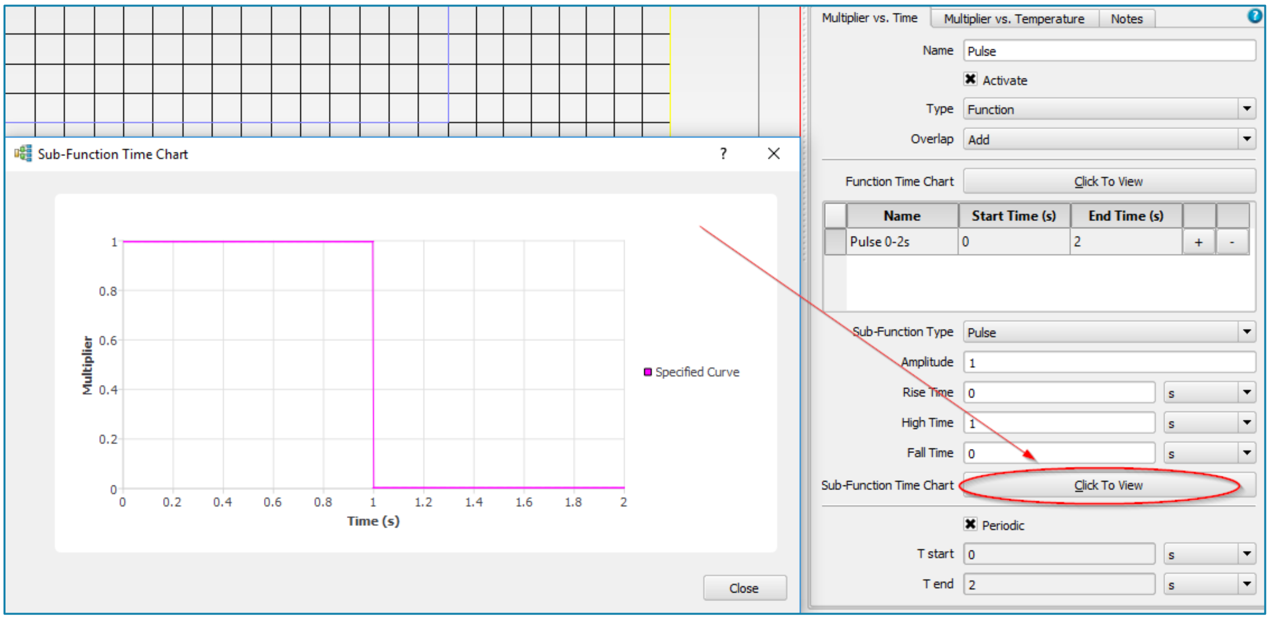


Figure 8 Transient Function

You could alternatively set the amplitude to 10 and set the thermal attribute with 1W rather than 10 – the profile amplitude will multiply the thermal setting. However it is best practice to vary the amplitude between -1 and 1.

You can view the sub-function by using the “Click To View” button – the pulse is shown in a new window, Sub-Function Tim Chart:

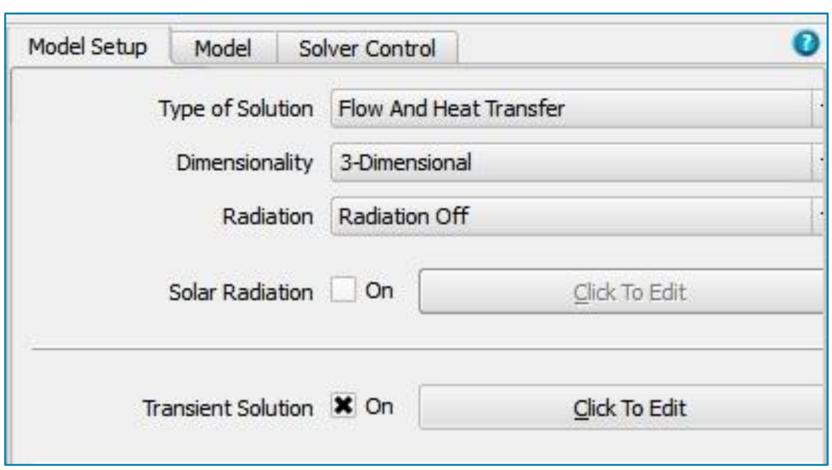


View Transient Function

Close the “Sub-Function time Chart”.

Now you have set up the transient function for the objects in the model. You need to set up the transient solution data.

In Project Manager **Click** on “Model Setup” tab. **Switch** On the “Transient Solution” and then “Click to Edit” to open the “Transient Solution” dialogue.

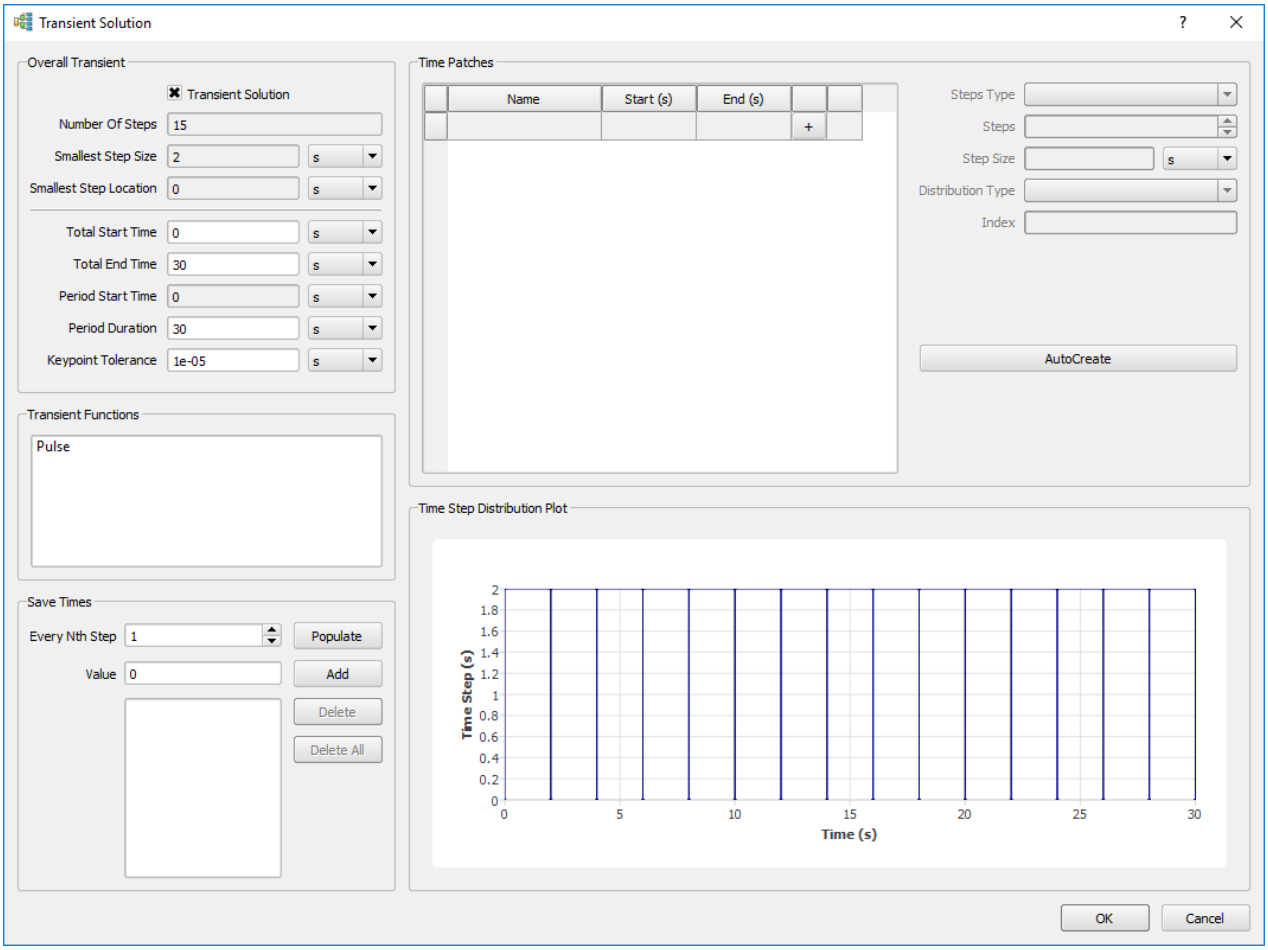


Overall Transient Controls

Under the "Overall Transient" controls: **enter** "Total End Time" of 30s, and "Period Duration" 30s.

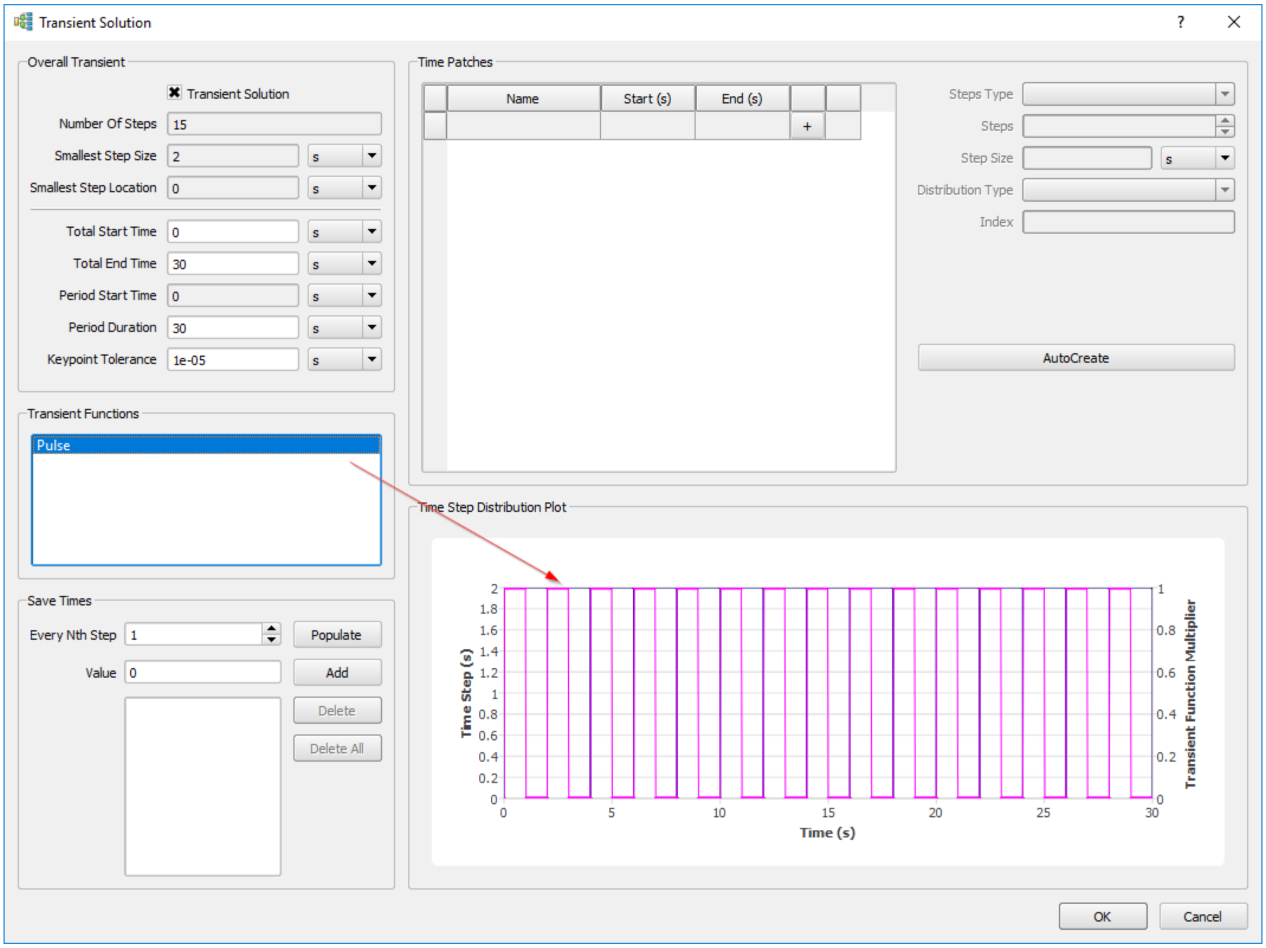
The "Time Step Distribution Plot" part of the dialogue shows in pink the times when any of the transient functions defined in the model (shown in the "Transient Functions" part of the dialogue) start and/or stop.

In this case a pink line is shown at 2s intervals since our pulse function is defined over a 2s period.



Time Step Distribution

When you highlight a transient function in the list the shape of the function is shown in the distribution plot.



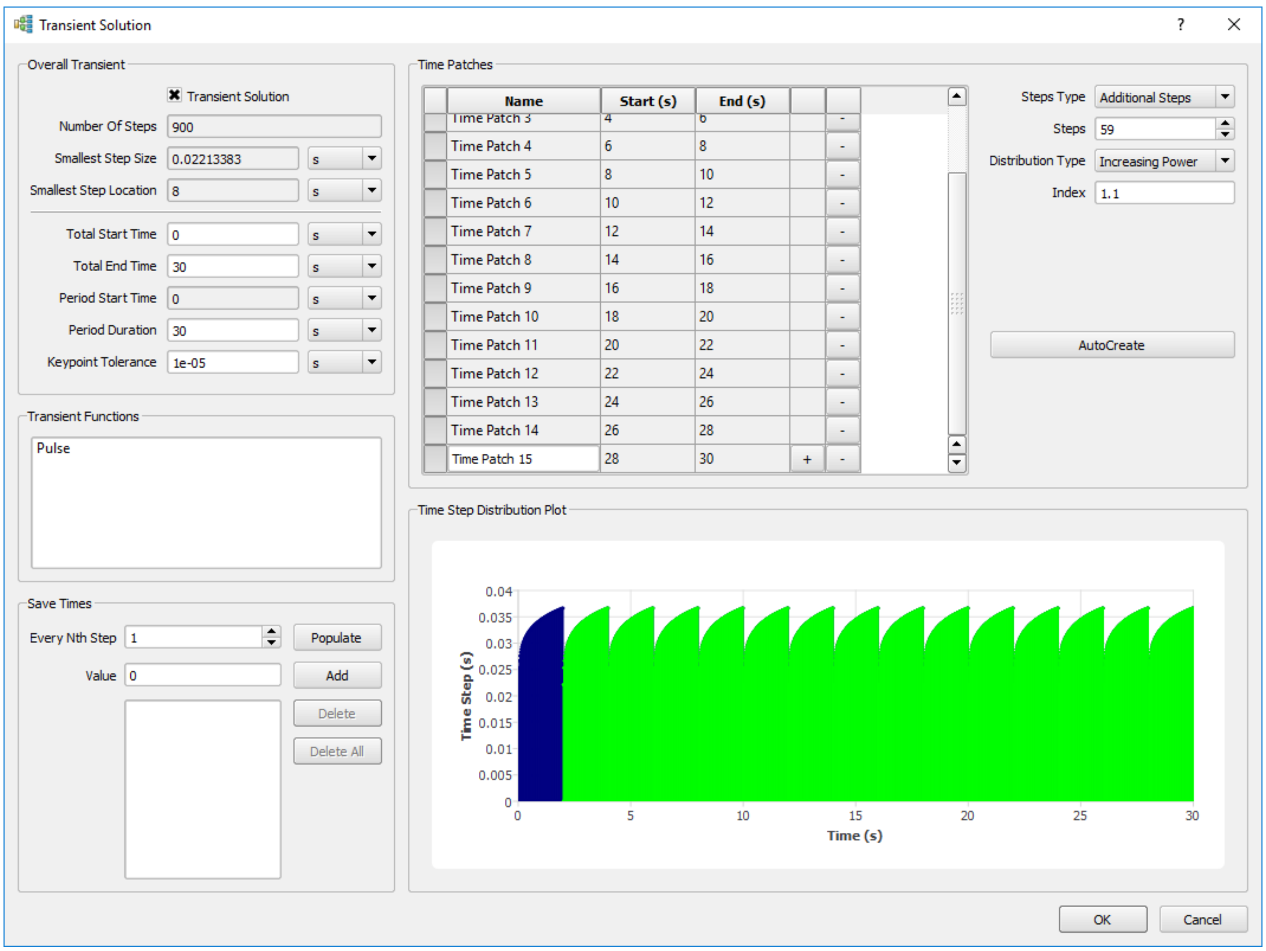
Transient Function Overlay

Time patches can be created manually or automatically. "AutoCreate" option in Transient Solution dialogue uses the transient functions to create patches that span function change (in this case each pulse). These are named by time span and can be edited one at a time or multiples together by highlighting in them in the list and changing their attributes.

Click the "AutoCreate" button to the right of the "Time Patches" section to create automatic time patches based on the pulse transient function.

For "Steps type" select "Additional Steps" and set "Steps" to 59 so each time patch has a total of 60 time steps within it. Under "Time Step Distribution" **select** "Increasing Power" and set it to 1.1.

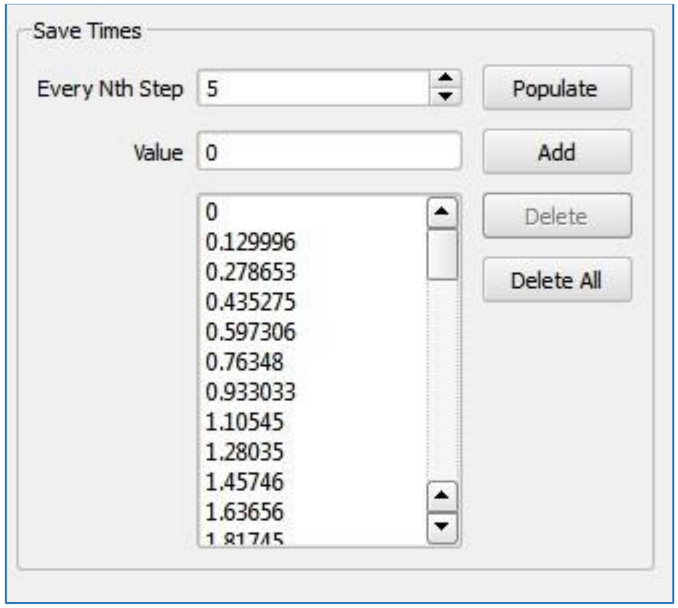
This changes the time steps within a time patch from equal length to increasing length. This is useful when large changes occur suddenly within the model (like a power source being switched on) and allows for small time steps when the rate of change in the model will be highest, the time steps can increase in length as the solution continues and the rate of change decreases.



Time Patch Setup

You need to add in the times you want to save data for so that results can be interrogated with respect to time.

You can automatically populate the save time list for every nth time step by using the "Populate" button. Alternatively use "Add" to manually enter times when data should be saved. Under "Save Times" **enter** 5 for "Every nth step" and **click** "Populate"



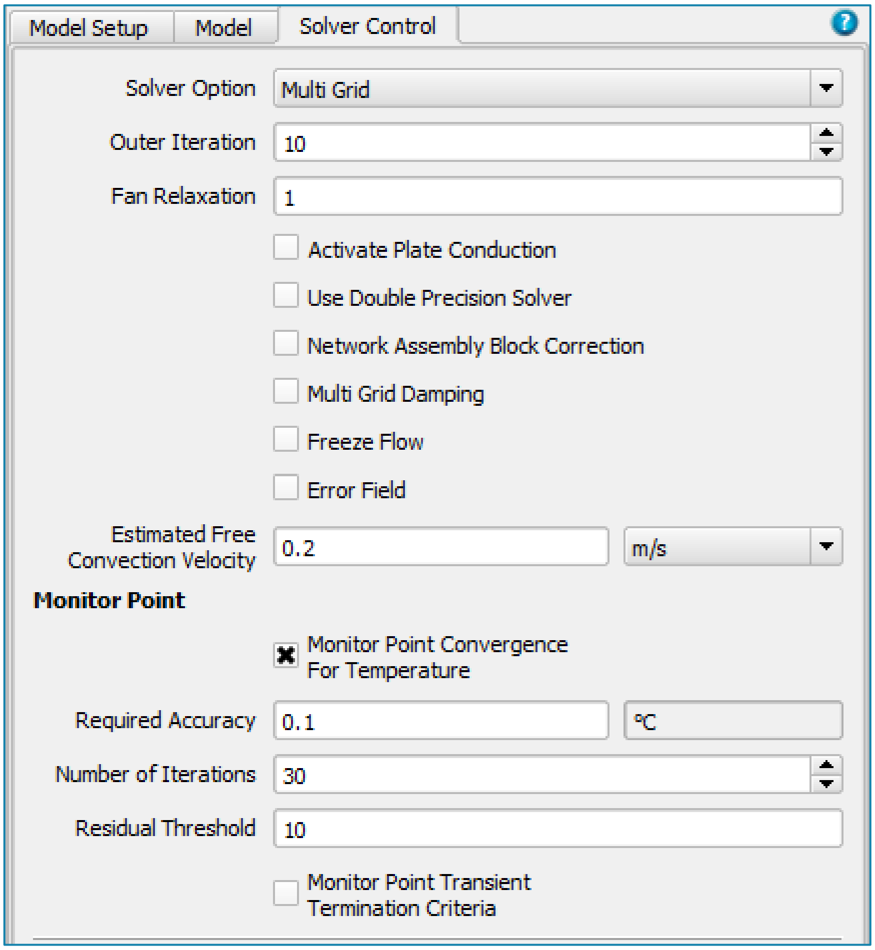
Save Times

If you manually enter save times Simcenter Flotherm will create a save point at the end of a time step closest to the time you specified.

The Transient Solution set-up is now complete so **click** OK to save and exit the dialogue.

You can use monitor point convergence criteria to check for convergence at each time step.

In Project Manager go to “Solver Control” tab, **set** the outer iterations to 10 and **Switch** on “Monitor Point Convergence for Temperature”. **Set** the required accuracy to 0.1 deg C but leave the other settings at their default values.



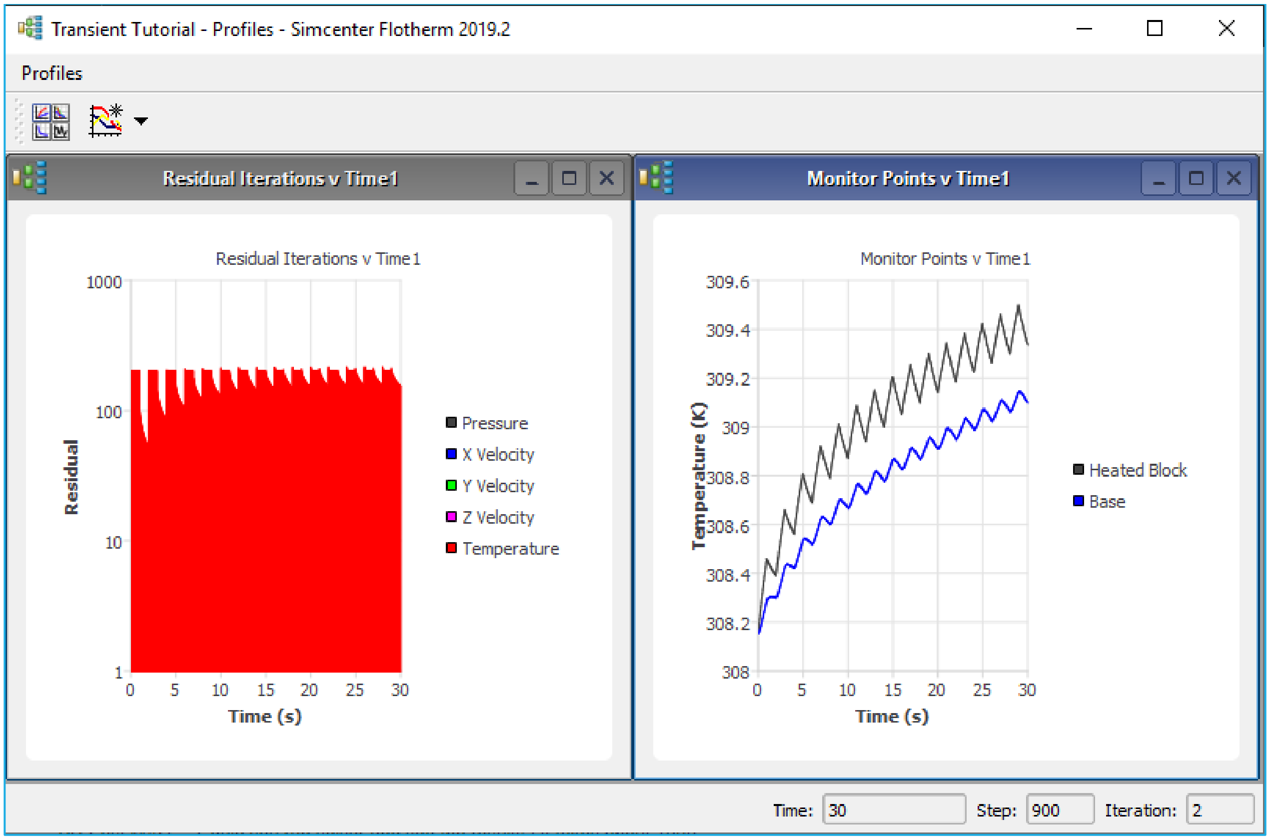
Monitor Point Convergence

This setup ensures that the solution at a given time step will be considered to be converged if the monitor point temperatures are within 0.1 deg C over 30 iterations and all the residual values are also below 10.

You can now solve the model.

In Project Manager **go to** [Solve/Re-initialize and Solve]

As the project solves you can see how the monitor points follow a saw-tooth pattern as the blocks are heated and then cool slightly when the power is off.

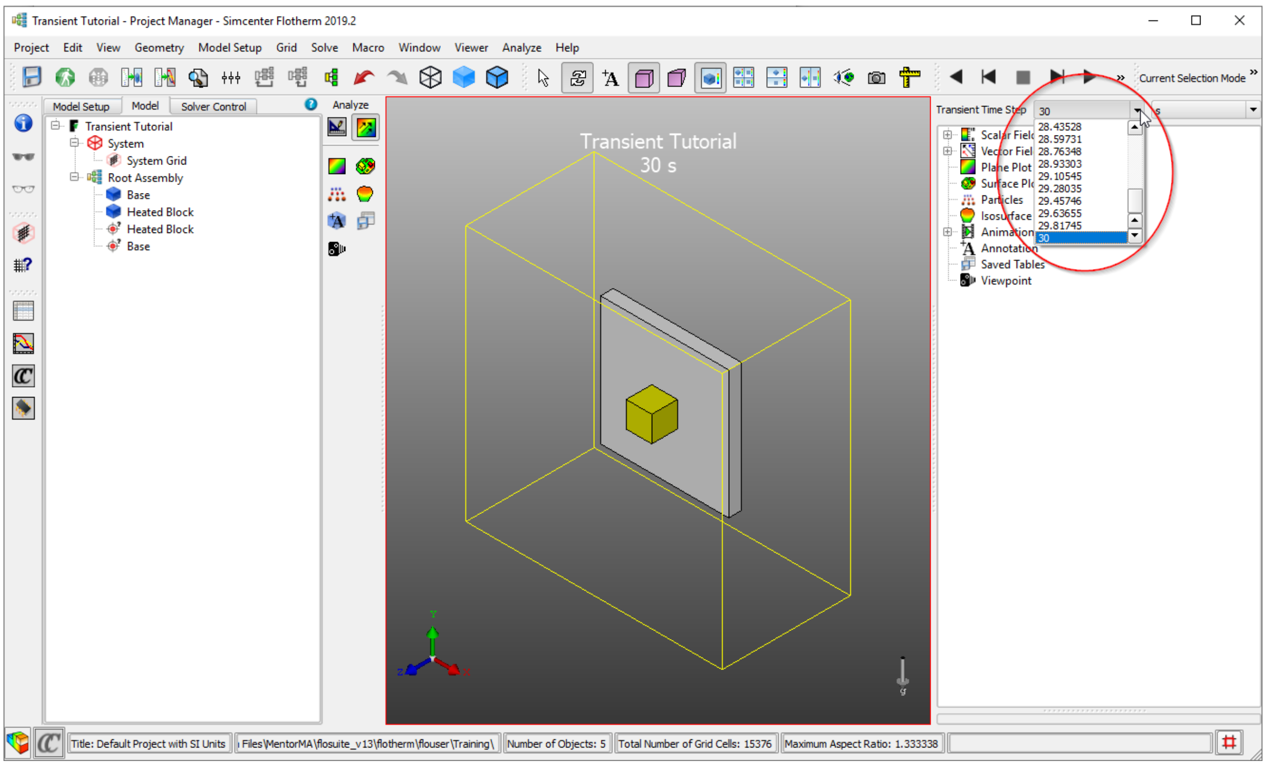


Profiles Window

Once solved you can investigate the project results in Analyze Results.

If **Analyze Results** in not already open**, click** the [] icon.

By default the results for the final time step are loaded (results for t=30s in this case) To load results for a different save time, select the time from the drop down on the top right-hand corner.

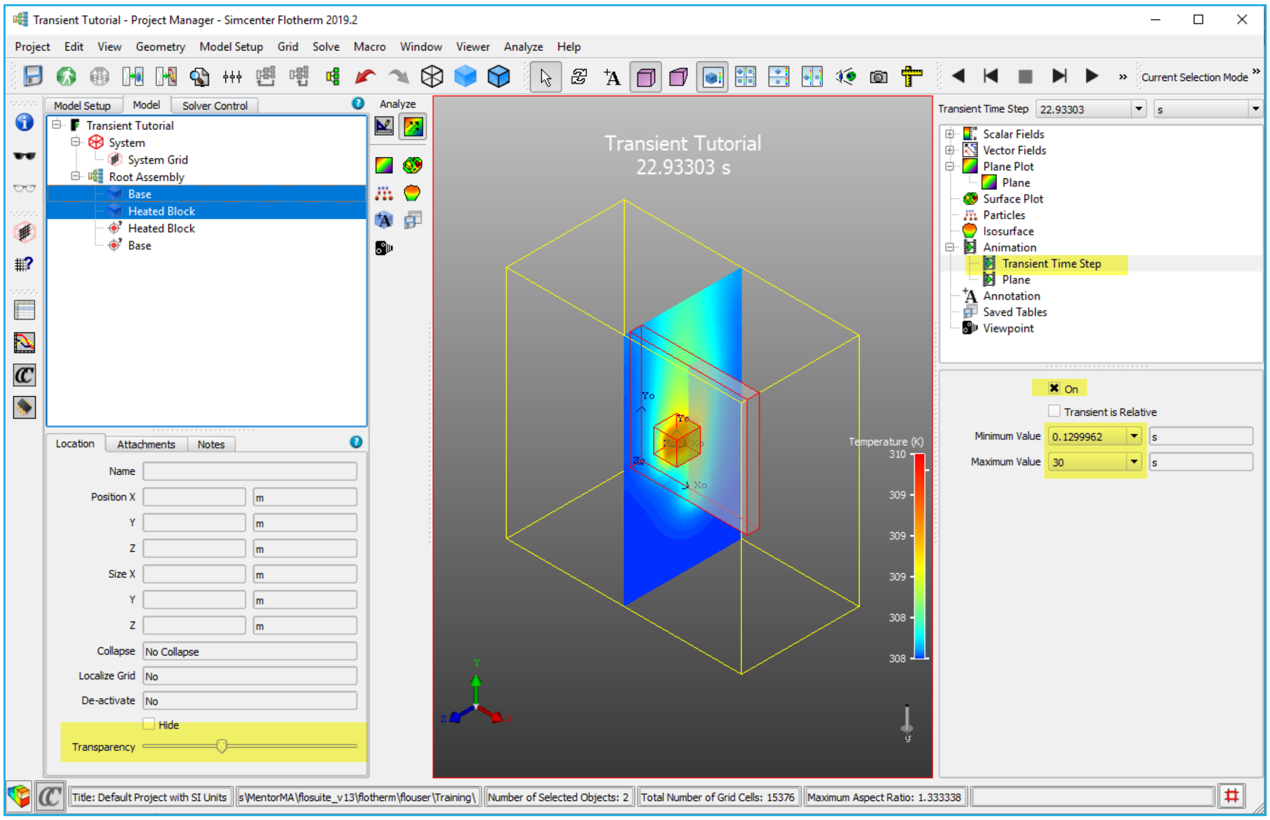


Loading Save Time Data

Once the data is loaded for a particular time step you can display results in graphical space in the usual way.

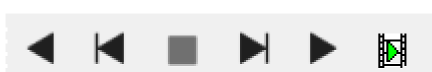
You can also create animations that show a particular plane plot or surface plot and how it changes over the course of the transient simulation.

Create a plane plot on the X axis at 0.1m and display temperature. **Change** the transparency of the two cuboids on the left hand side. In the top right side of the window, **expand** the “Animation” menu and **highlight** “Transient Time Step”. In the property sheet below **switch-on** the transient settings, and set the minimum and maximum save time values to the extremes.



Transient Animation

Use the controls in the animation toolbar to play and control the animation.



Animation Controls

Use the output animation icon [] to save an “.avi” file.

In the “Results” property sheet there is an option to switch on “Transient is Relative”. Switching on this option will set the animation process to be timed proportionally to the actual time values.