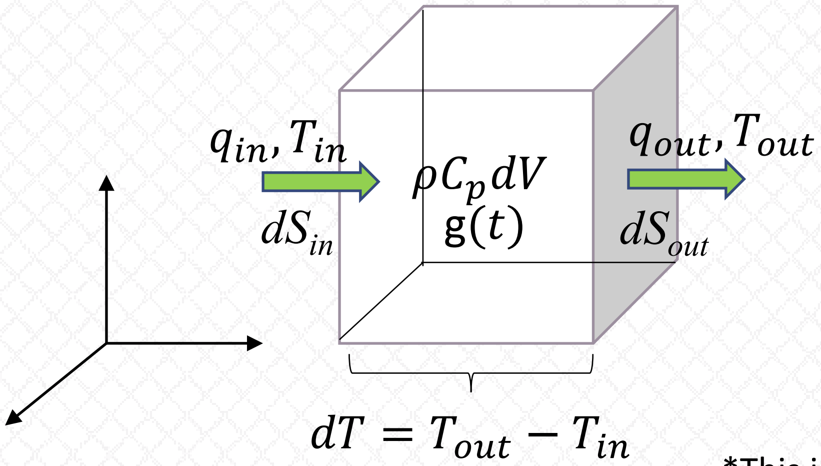
How to Perform Equilibrium Checks on Transient Heat Transfer Numerical Models

Standard Equlibrium Checks

- When conducting heat transfer studies using finite element models, users often check that the model satisfies thermal equlibrium
- With Steady-State models, this is straightforward: On all surfaces, $\sum \dot{Q}$=0
- In other words: When the direction of heat flow is known and all heat sources and sinks lie on domain boundaries, we can identify surfaces S_{in} and S_{out} such that the total heat flow across each surface (or set of surfaces) follows the relation $\dot{Q_{in}} = \dot{Q_{out}}$
- But when evaluating transient thermal models far from steady-state, $\sum \dot{Q} \neq$0. In fact, one could use this as a *definition* of the transient condition. We could name the condition $\sum \dot{Q}$=0 the *Static Equlibrium Condition*.
- Most engineers know how to perform the standard check for static equluilibrium, but fewer know how to perform a similar check for the transient condition
- Worse still: Many believe that $\sum \dot{Q}$=0 still holds in the transient case, and are surprised when it doesn't. They believe something may be wrong with the model when they see that $\dot{Q_{in}} \neq \dot{Q_{out}}$
- In this article, we'd like to correct this misimpression and show how to perform an equilibrium check "the right way" on transient models.

Flux Equlibrium for a Transient Heat Transfer Control Volume

- To recap: In a transient thermal problem, $\sum \dot{Q} \neq 0$
- Instead: $\sum \dot{Q} = f(t, Q_{in}, T(t), k(t), C_p(t))$
- If we knew f for any location, we could (in principle) perform a flux equilibrium check
- f can be easily obtained by applying energy conservation to an arbitrary infinitesimal control volume per unit time as below*
- The surface dS_{in} is associated with the heat source (net heat flow into the surface), while dS_{out} is associated with the heat sink (net heat flows out of the surface)
- We can then express the governing equations for transient heat transfer across this volume.
- Below, q_{in} and q_{out} are the net fluxes on face dS_{in} and dS_{out} respectively.
- q_{in}^0 and q_{out}^0 are prescribed fluxes *on* these surfaces



- @dS_{in}: $$\sum_{dS_{in}} q_{in} = q_{in}^0 - k\nabla T_{out} + \rho c_p dx_i \frac{\partial T}{\partial t} - g(t)dx_i$$

  where:

- @dS_{out}: $$\sum_{dS_{in}} q_{out} = q_{out}^0 - k\nabla T_{out}$$

*This is NOT the most general formulation of this problem. In particular, we're enforcing a uni-directional heat flow parallel to a Cartesian axis. The results transfer easily to more complicated situations, however. The intended audience is Engineers

Flux Equlibrium for a Transient Heat Transfer Control Volume

- As useful as this is, don't be discouraged if you haven't seen it before (it's not in my Heat Transfer textbook explicitly – but one can put it together from what *IS* there)
- Note that the direction of heat flow (or flux in this case), q is critical. This is because the capacitance term, $\rho c_p dx \frac{\partial T}{\partial t}$ strictly measures the rate of change of the internal energy of the control volume, dV (per unit area the way we're using it)
- In other words, this term measures the absorbtion of heat into the volume. A value of q measured at any surface downstream of the heat flow will reflect this volume's internal energy subtracted (filtered) –which we'll show later.
- $g(t)$ is pescribed volumetric heat generation (units of energy per unit time per unit volume)
- The mot important point about these equations for net heat flow through faces S_{in} and S_{in} is that it must equal *Zero*!

- The *Transient Equilibriuim Condition*

- $@dS_{in}$:

$$\sum_{dS_{in}} q_{in} = q_{in}^0 - k\nabla\dot{T}_{out} + h(t) = 0 \qquad \text{where:} \quad h(t) = \rho c_p dx_i \frac{\partial T}{\partial t} - \mathrm{g(t)} dx_i$$

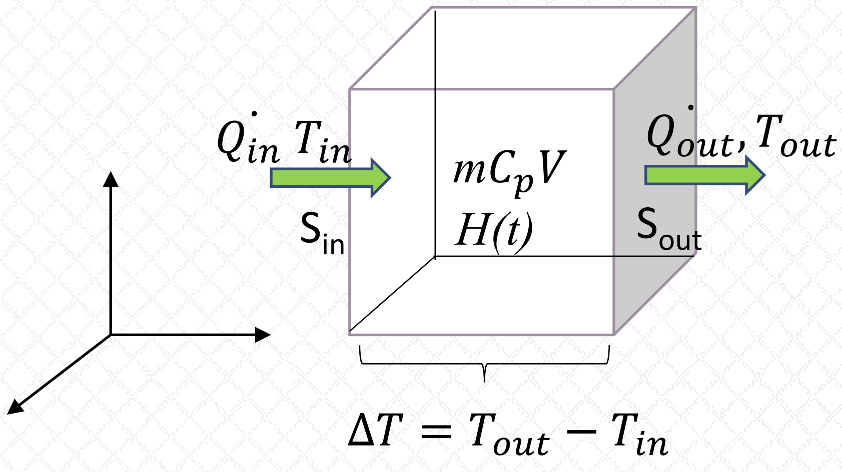
- $@dS_{out}$:

$$\sum_{dS_{in}} q_{out} = q_{out}^0 - k\dot{\nabla}T_{out} = 0$$

- Now, if we can identify a finite region of our model with bounding surfaces, S_{in} and S_{out}, such that the fluxes across these surfaces are reasonably uniform, and the region is isotropic and homogeneous, then we can replace the infinitesimal control volume, dV with this region

Flux Equlibrium for a Transient Heat Transfer Control Volume

- Replacing the infinitesimal control volume with a finite region means we have to integrate terms (the capacitance term is integrated over the volume, while the conduction and boundary terms are integrated over S_{in} and S_{out}. We could use the Divergence Theorem here)
- Also, we replace the derivatives with their finite difference counterpart (we're checking a numerical model)



- @S_{in}: $\displaystyle\sum_{dS_{in}} \dot{Q} = \dot{Q}^0_{in} - k\frac{\Delta T}{\Delta x_i}A_{out} + H(t)$

where: $$H(t) = mc_pV\frac{\Delta T}{\Delta t} - gV(t) \qquad (1)$$

- @S_{out}: $\displaystyle\sum_{dS_{out}} \dot{Q} = \dot{Q}^0_{out} - k\frac{\Delta T}{\Delta x_i}A_{out} = 0$

- Here,we've replaced the fluxes with their surface-integrated counterparts $\dot{Q}$ and (these have units of power), and $H(t)$ is now the volume-integrated net heat generation (units of power)
- Hopefully, all other variables are obvious from the context (dS_{in} and dS_{out} become S_{in} and S_{out}, dV becomes V, etc...). A_{in} and A_{out} are the respective surface areas.

Flux Equlibrium for a Transient Heat Transfer Control Volume

- Notice also how the conduction term, which was constant in the infinitesimal control volume, no longer is (dT/dx is no longer constant) within the finite region
- What this means is that the conduction terms must be evaluated at the surface S_{out}, and Δx_i represents a small spatial increment opposite the heat flow from this surface
- Ideally, $\Delta x_i \to 0$, but since we're evaluating a finite element model, Δx_i will in general represent the smallest spatial increment available to query in ANSYS, and this will correspond to an element thickness

Predictions

- Suppose we're modeling a volume entirely within a larger domain (no surface boundary conditions)
- Equation (1) still tells us that:
  - @S_{in}: $\sum_{dS_{in}} \dot{Q} = \dot{Q}_{in}^{0} - k\frac{\Delta T}{\Delta x_i} A_{out} + H(t) = 0$
- Or

$$\dot{Q}_{in}^{0} = k\frac{\Delta T}{\Delta x_i} A_{out} - H(t) \quad (2)$$

Flux Equlibrium for a Transient Heat Transfer Control Volume

- Equation (2) tells us that the power (heat dissipation), $\dot{Q}_{in}^{0}$ on an internal face will be equal to the difference of the conductive heat flow and the heat generation
- Similarly, the second part of equation (1) tells us that, at the heat sink:

$$\dot{Q}_{out}^{0} = k\frac{\Delta T}{\Delta x_i}A_{out} \qquad (3)$$

- This immediately suggests a general procedure to check for transient heat equilibrium in ANSYS, because we can insert a heat flux probe to directly query the value of $\dot{Q}_{out}^{0}$ (after multiplying that the heat flux by A_{out}) ,and according to (3), this should equal the conductive heat flow term on that surface*
- Once we have that, equation (3) can be plugged into equation (2):

$$\dot{Q}_{in}^{0} = \dot{Q}_{out}^{0} - H(t)$$

Or $$\dot{Q}_{in}^{0} - \dot{Q}_{out}^{0} = H(t)$$

- This can be used to find the $\dot{Q}_{in}^{0}$, once we have the heat generation term. And this may be obtained from an ANSYS User Defined Result (worksheet item='EHEAT')
- As a further bonus, equation (2) could then be checked by inserting another heat flux probe at S_{in}

*Checking this value against a reaction probe result is also a good check, because these quantities are calculated differently and should equal one-another

The General Procedure

- The procedure we're going to outline extends more generally than the simple model we'll demonstrate it on*.
- Ideally, the steps that follow could all be handled entirely within Workbench using result expressions, but there appear to be limitations to this**. Therefore, we recommend that the checking procedure be performed by cutting-and-pasting result items into a spreadsheet (at least up to ANSYS 2022R1).

1. Identify the model's heat source and sink. These may involve points, edges, and element types other than volumetric. And a source and sink may each consist of multiple geometric entities (not just a single surface)
2. Start at the sink and insert a heat flux probe to it. Copy-and-paste this into a spreadsheet. In the spreadsheet, multiply this by A_{out} to obtain the heat dissipation
3. Move on to surface S_{in} (toward the heat source). Create a User Defined 'EHEAT' result on the volume bounded by S_{in} and S_{out}. Cut-and-paste this result into the spreadsheet as well. This time, you don't have to multiply by anything. In the spreadshet, perform equation (2). Then check equation (2) against another heat flux probe result at S_{in}. Equation (2) tells us these two results should equal
4. Repeat 3. for every distinct volumetric region (control volume) until you arrive at the source. Once there, equation (2) should equal the thermal source load.

*The procedure may fail on thermal shell elements (although there's a probably a workaround for that),

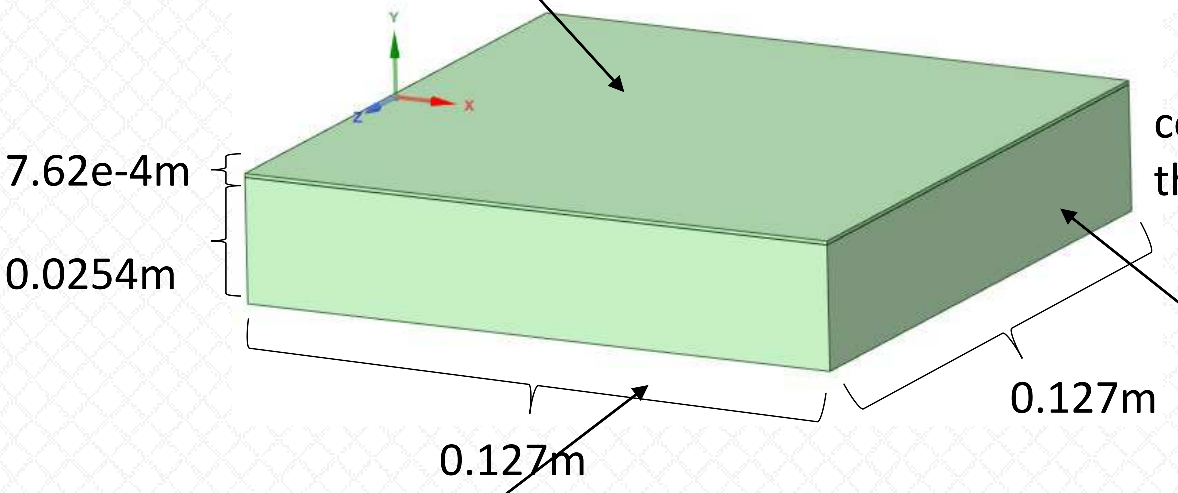
** At 2022R1, you can't combine more than one result expression, for example. You also can't use probe results for this purpose

A Simple Example

- Consider the transient heat transfer model problem as below
- Note there are two distinct domains (this isn't quite a single homogeneous domain)

- Source Convection

Time [s]	Convection Coefficient [W/m²·°C]	Temperature [°C]
0.	20.	23.
1.	1000.	1000.
60.	1000.	1000.



conductive thick block

- Sink Convection

Time [s]	Convection Coefficient [W/m²·°C]	Temperature [°C]
0.	5.	23.
10.	5.	23.
60.	5.	23.

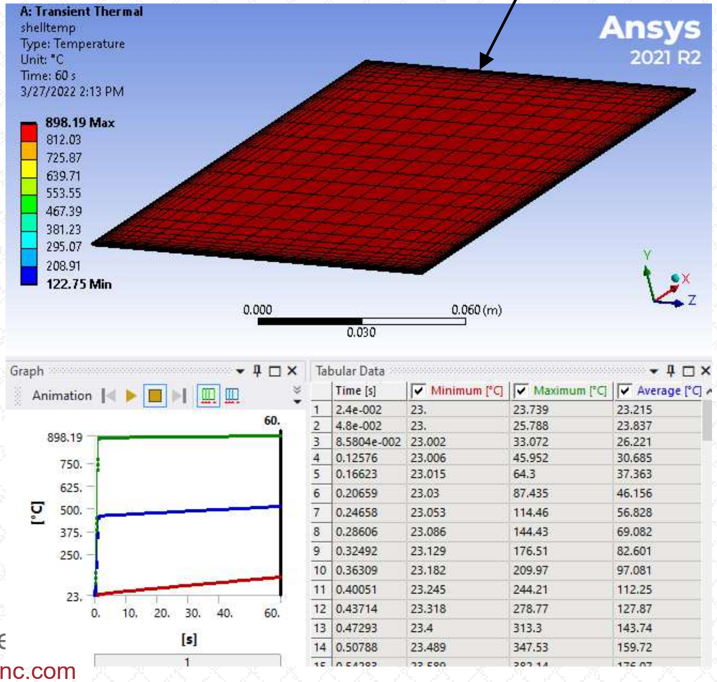
- internal heat generation

Time [s]	Internal Heat Generation [W/m³]
0.	0.
10.	1.6273e+007
20.	0.
60.	0.

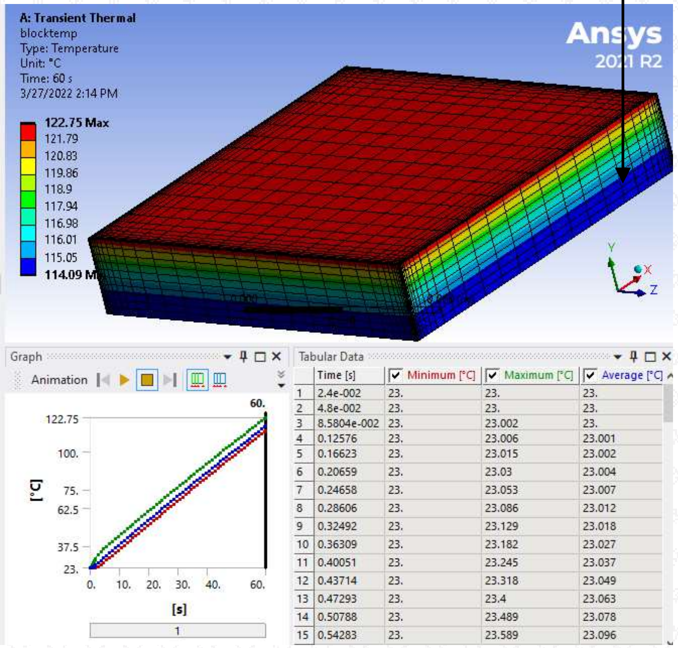
- Insulating thin block:
  - K = 0.1 W/m C
  - m = 2.4581e-3 kg
  - cp = 945.4 J/kg C
  - t = 7.62e-4 m
  - A = 0.016129 in^2
- Conductive thick block:
  - K = 150 W/m C
  - m = 1.1471 kg
  - cp = 945.4 J/kg C
  - t = 0.0254 m
  - A = 0.016129 in^2

A Simple Example

- At the end of 60 seconds, the model’s temperature is as shown below
- A quick look at the Min, Max, and Average temperature graphs (vs time. Workbench gives us these for free) shows that the thin block is nearing steady-state while the thick block is still far from this condition
- What we want to do is perform the procedure on slide 8 on each block individually, treating each as a homogeneous control volume, V_i
- We’ll call the thin, top block, V_1, and bottom, thick block V_2 . According the model configuration, the top surface of V_1 (S_{in}) is the source, while the bottom surface of V_2 (S_{out}) is the sink



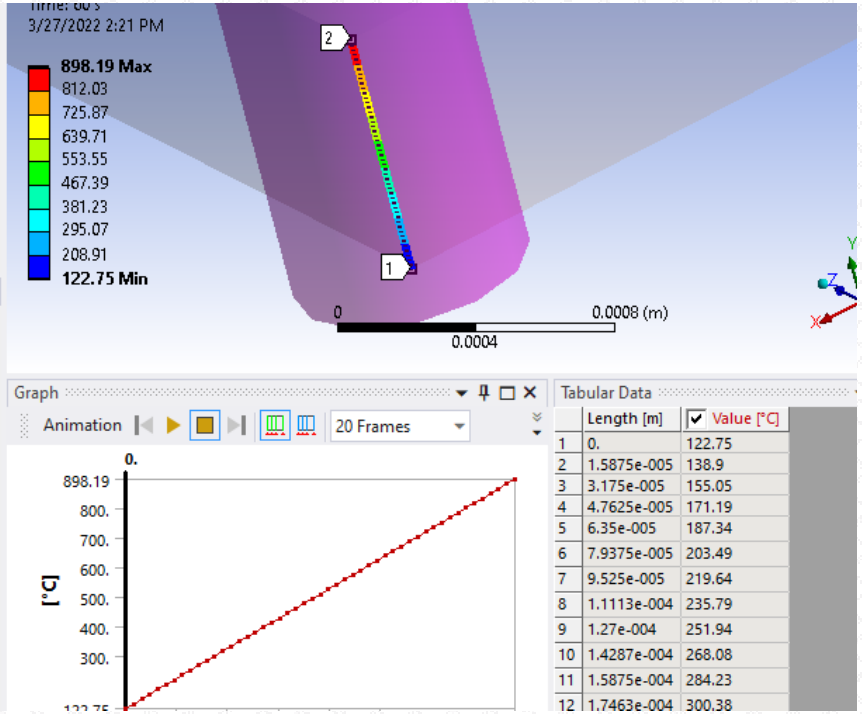
	Time [s]	Minimum [°C]	Maximum [°C]	Average [°C]
1	2.4e-002	23.	23.739	23.215
2	4.8e-002	23.	25.788	23.837
3	8.5804e-002	23.002	33.072	26.221
4	0.12576	23.006	45.952	30.685
5	0.16623	23.015	64.3	37.363
6	0.20659	23.03	87.435	46.156
7	0.24658	23.053	114.46	56.828
8	0.28606	23.086	144.43	69.082
9	0.32492	23.129	176.51	82.601
10	0.36309	23.182	209.97	97.081
11	0.40051	23.245	244.21	112.25
12	0.43714	23.318	278.77	127.87
13	0.47293	23.4	313.3	143.74
14	0.50788	23.489	347.53	159.72



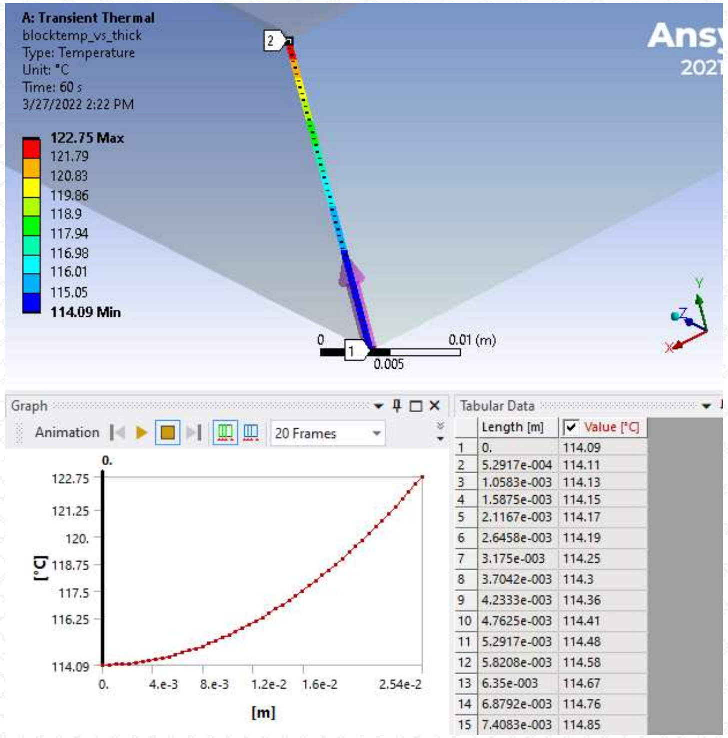
	Time [s]	Minimum [°C]	Maximum [°C]	Average [°C]
1	2.4e-002	23.	23.	23.
2	4.8e-002	23.	23.	23.
3	8.5804e-002	23.	23.002	23.
4	0.12576	23.	23.006	23.001
5	0.16623	23.	23.015	23.002
6	0.20659	23.	23.03	23.004
7	0.24658	23.	23.053	23.007
8	0.28606	23.	23.086	23.012
9	0.32492	23.	23.129	23.018
10	0.36309	23.	23.182	23.027
11	0.40051	23.	23.245	23.037
12	0.43714	23.	23.318	23.049
13	0.47293	23.	23.4	23.063
14	0.50788	23.	23.489	23.078
15	0.54283	23.	23.589	23.096

A Simple Example

- Since V_1 is near steady-state, the temperature distribution through the thickness must be nearly linear (because dT/dx = constant), while this will not be the case for V_2 (which is also evident just by looking at the temperature contour distribution
- We can verify this in Workbench with a “path” result (tempeature vs. thickness direction):
- The linear condition will make certain calculations easier for V_1 (we’ll come back to this)



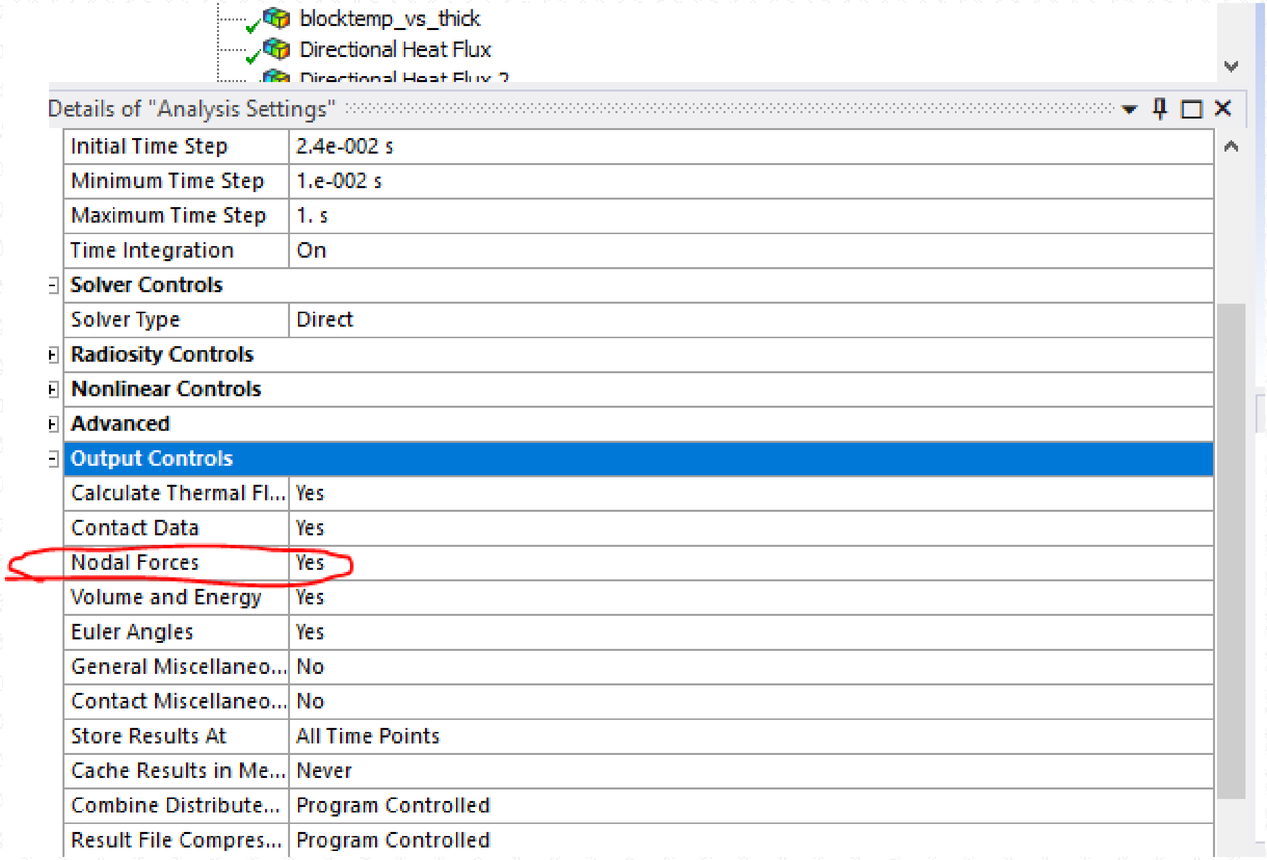
Temperature vs thickness for thin block



Temperature vs thickness for thick block

Model Preliminary Considerations

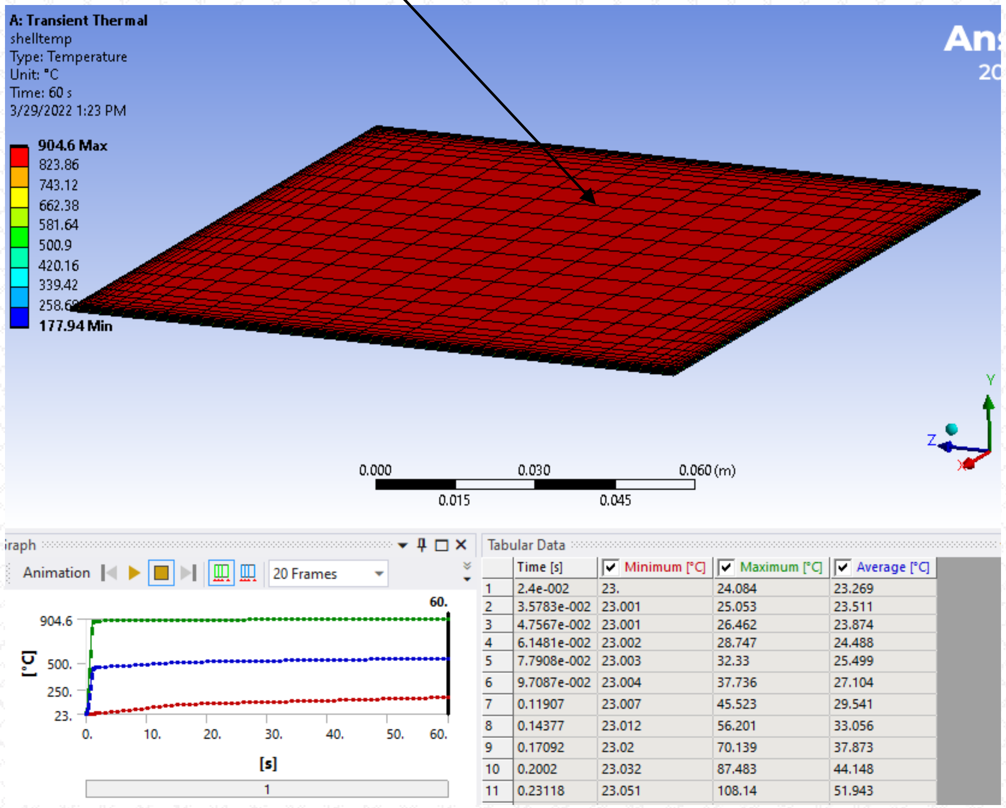
- Before querying model results to plug in to equation (1) thru (3), make sure the appropriate result quantities are stored
- This can be accomplished with the output controls shown below
- In particular, make sure ‘Nodal Forces’ is set to ‘Yes’ (surface flux extraction won’t work if this isn’t done)



Step 1: Identify Heat Source and Sink

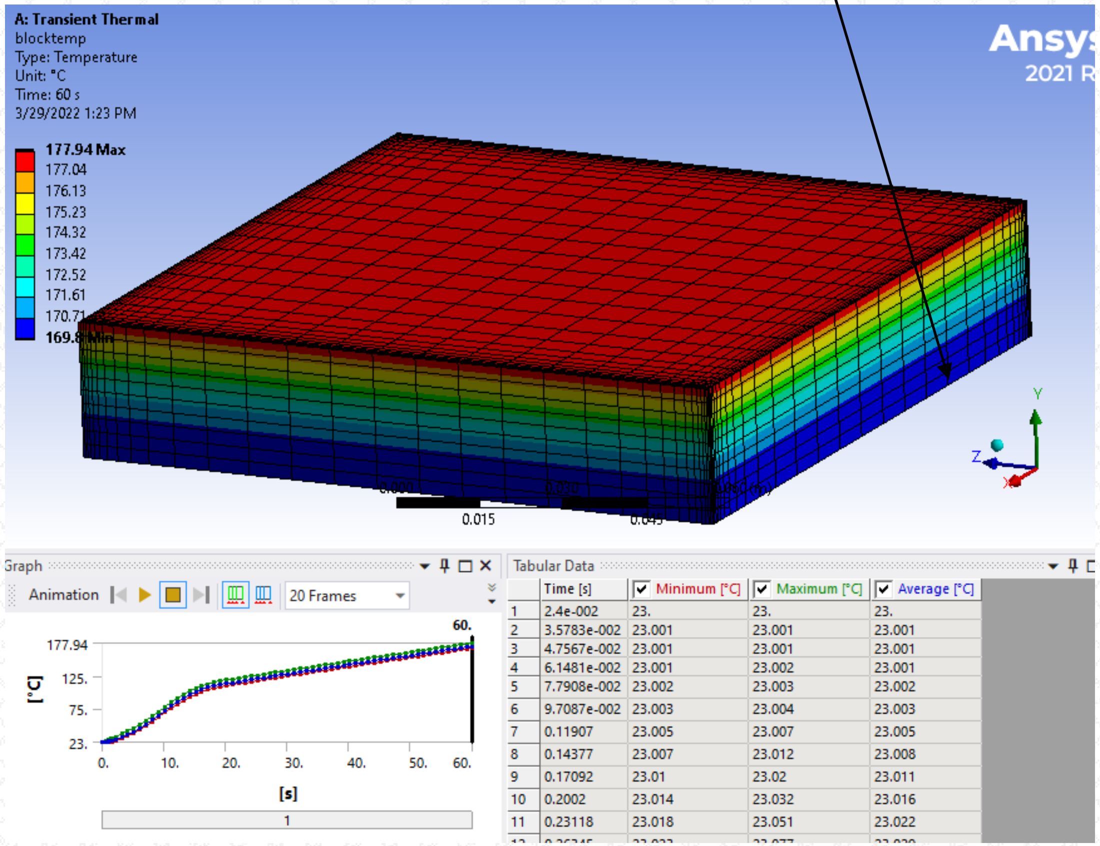
- For our simple model, its' fairly obvious. The source in the top surface of V_1 and the sink is the bottom surface of V_2

Source



	Time [s]	Minimum [°C]	Maximum [°C]	Average [°C]
1	2.4e-002	23.	24.084	23.269
2	3.5783e-002	23.001	25.053	23.511
3	4.7567e-002	23.001	26.462	23.874
4	6.1481e-002	23.002	28.747	24.488
5	7.7908e-002	23.003	32.33	25.499
6	9.7087e-002	23.004	37.736	27.104
7	0.11907	23.007	45.523	29.541
8	0.14377	23.012	56.201	33.056
9	0.17092	23.02	70.139	37.873
10	0.2002	23.032	87.483	44.148
11	0.23118	23.051	108.14	51.943

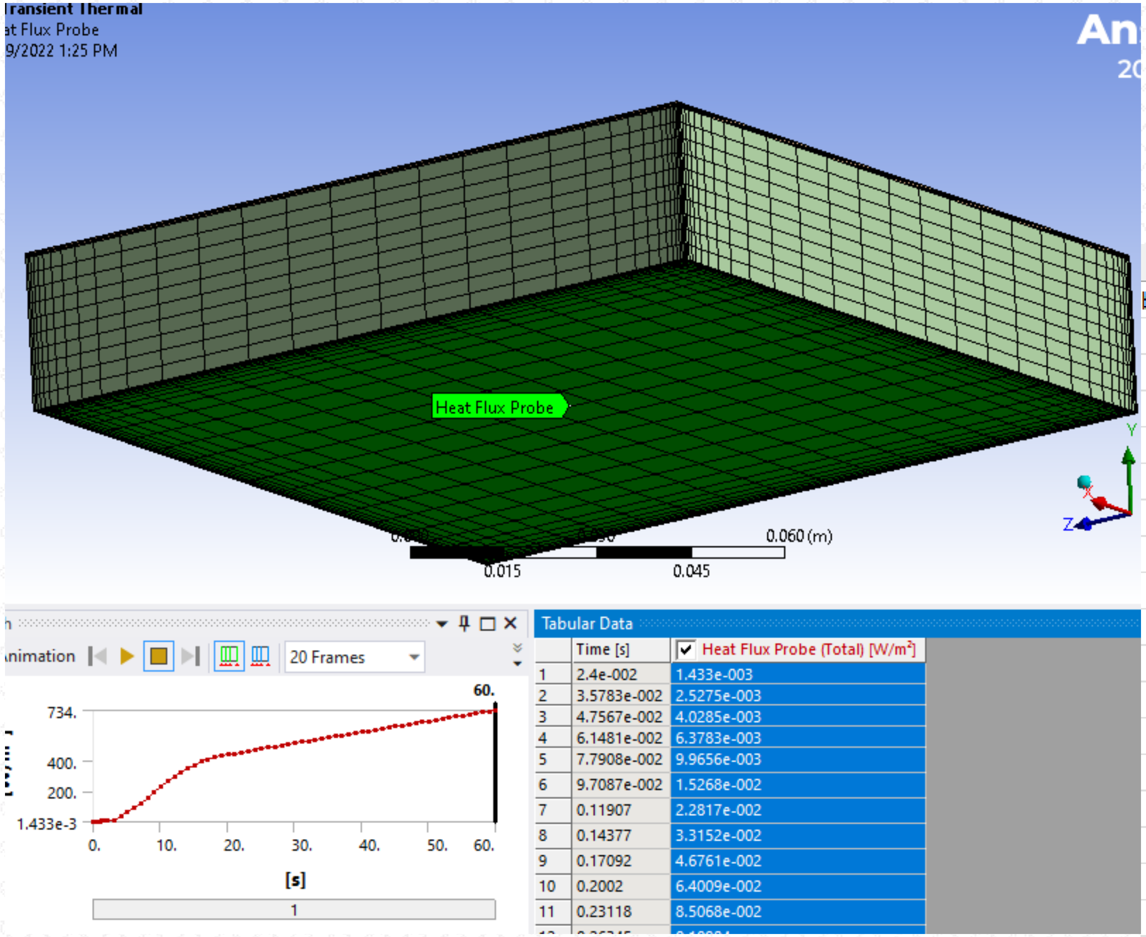
Sink

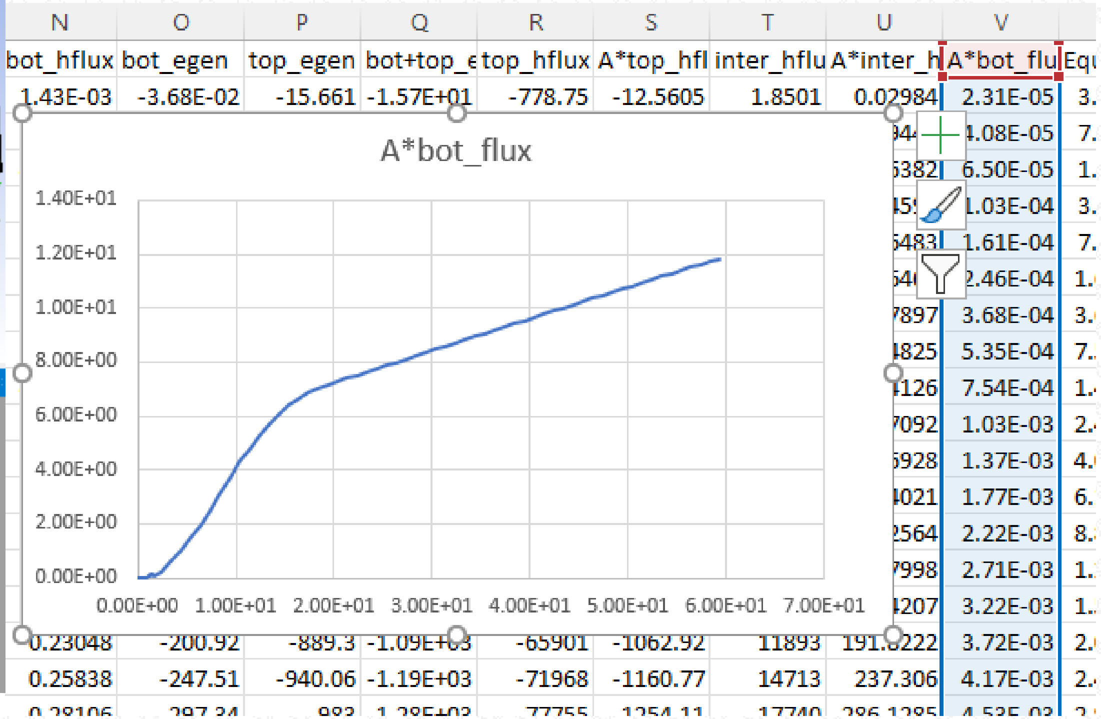


	Time [s]	Minimum [°C]	Maximum [°C]	Average [°C]
1	2.4e-002	23.	23.	23.
2	3.5783e-002	23.001	23.001	23.001
3	4.7567e-002	23.001	23.001	23.001
4	6.1481e-002	23.001	23.002	23.001
5	7.7908e-002	23.002	23.003	23.002
6	9.7087e-002	23.003	23.004	23.003
7	0.11907	23.005	23.007	23.005
8	0.14377	23.007	23.012	23.008
9	0.17092	23.01	23.02	23.011
10	0.2002	23.014	23.032	23.016
11	0.23118	23.018	23.051	23.022

Step 2: Extract Heat Dissipation from Sink

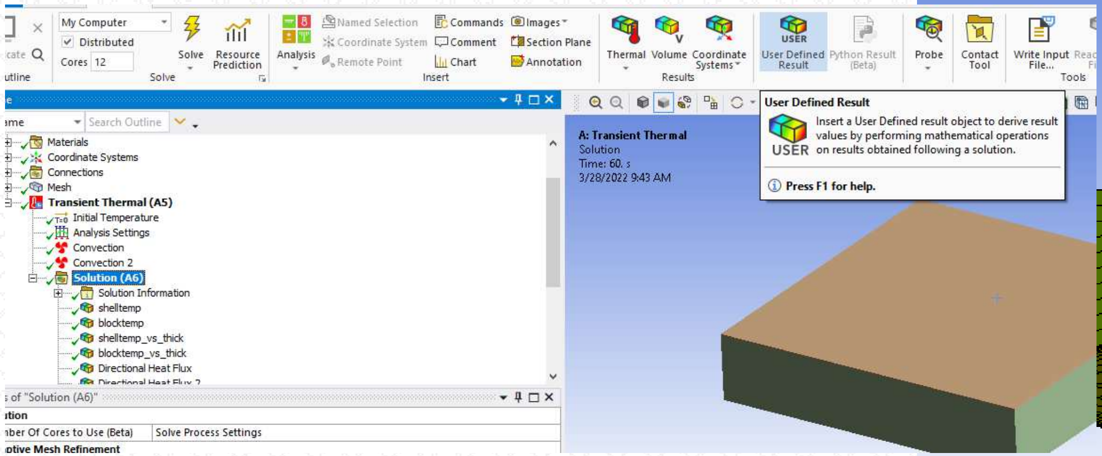
- Insert a heat flux probe on the sink surface. Copy-and-paste this into a spreadsheet. In the spreadsheet, multiply this by A_{out} to obtain the heat dissipation

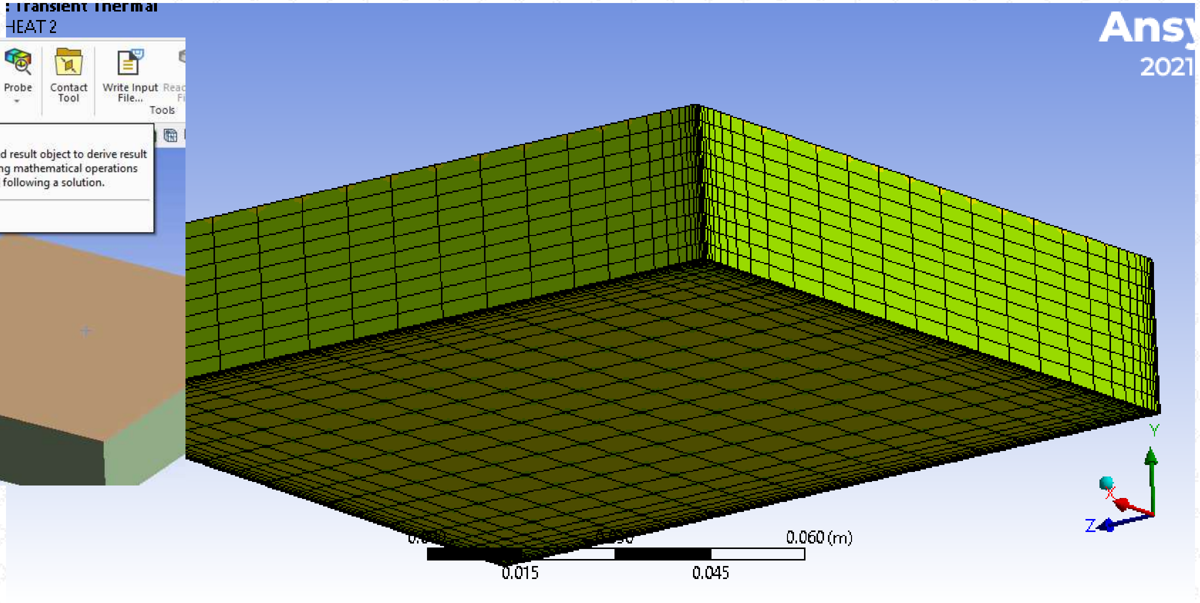


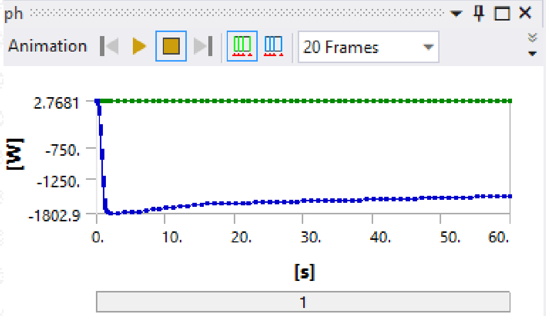


Step 3: Extract Heat Absorption Term and Perform Equation (2)

- Insert a User Defined ('EHEAT') Result for V_2
- Extract this result into the spreadsheet and subtract it from the previous result (Equation (2))
- Note: It's important to extract the Total (W) column*







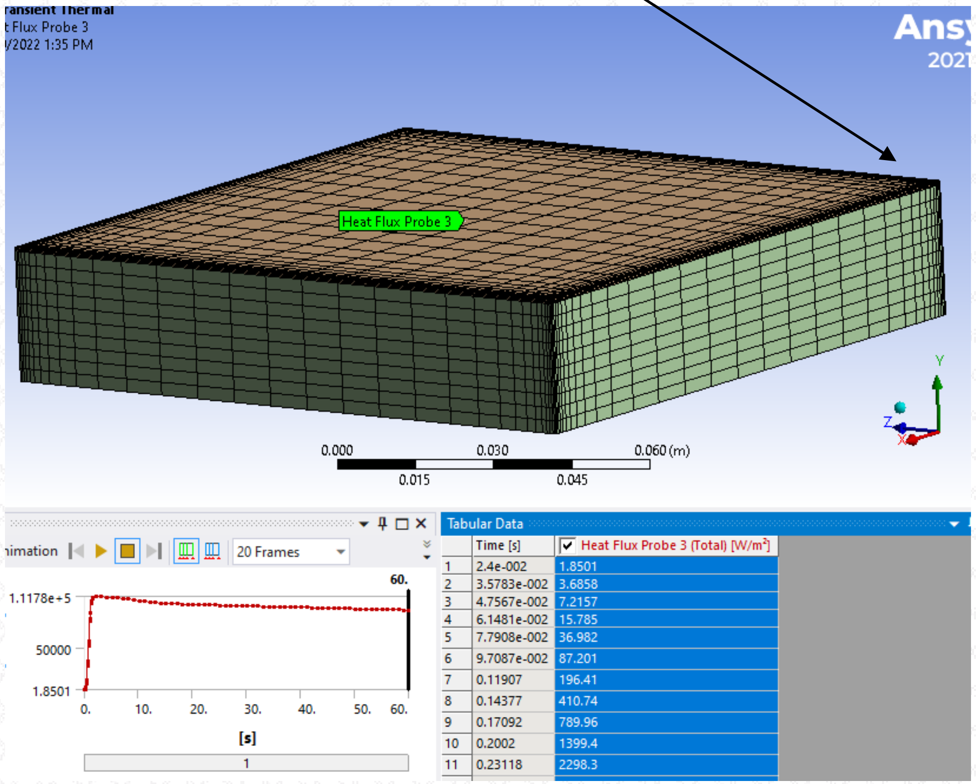
Tabular Data

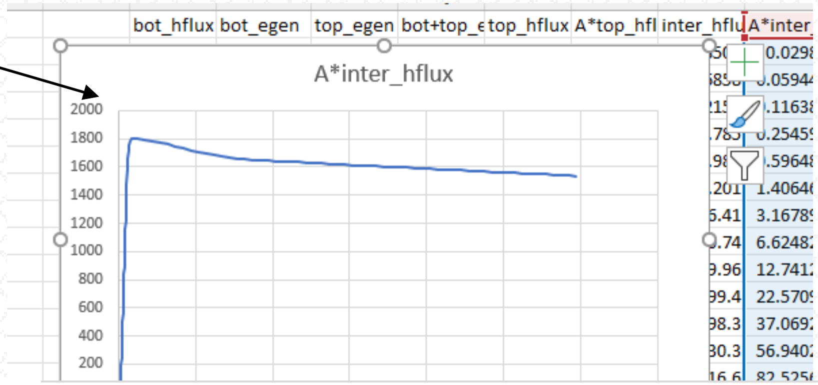
	Time [s]	Minimum [W]	Maximum [W]	Total [W]
1	2.4e-002	-1.1322e-004	5.6608e-005	-3.6844e-002
2	3.5783e-002	-2.2604e-004	1.1302e-004	-7.3569e-002
3	4.7567e-002	-4.4135e-004	2.2067e-004	-0.14366
4	6.1481e-002	-9.5642e-004	4.7821e-004	-0.31137
5	7.7908e-002	-2.2044e-003	1.1022e-003	-0.71775
6	9.7087e-002	-5.0938e-003	2.5469e-003	-1.6587
7	0.11907	-1.1238e-002	5.6188e-003	-3.6595
8	0.14377	-2.3054e-002	1.1527e-002	-7.5077
9	0.17092	-4.3604e-002	2.1802e-002	-14.2
10	0.2002	-7.6151e-002	3.8076e-002	-24.8
11	0.23118	-0.1236	6.1797e-002	-40.251

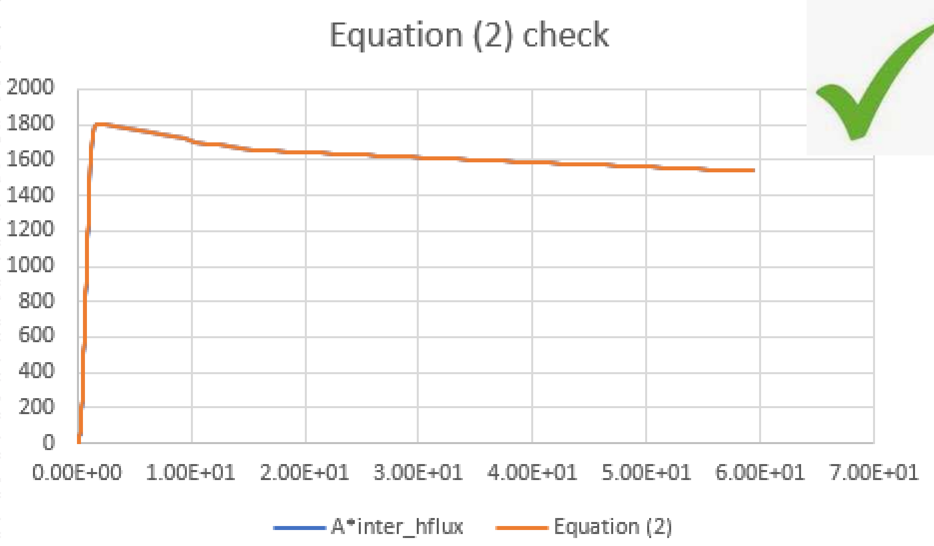
*Also, you can't just perform this step on multiple volumes at once, because we have to add Total values – not eheat components)

Step 3: Extract Heat Absorption Term and Perform Equation (2)

- Now, in the spreadsheet, subtract the heat generation term we just extracted from the heat flux from the block bottom per Equation (2)
- Place another heat flux probe at the surface interface between V_1 and V_2 and multiply this by A_{in} (in the spreadsheet)
- In the spreadsheet, compare the two results. They should match!

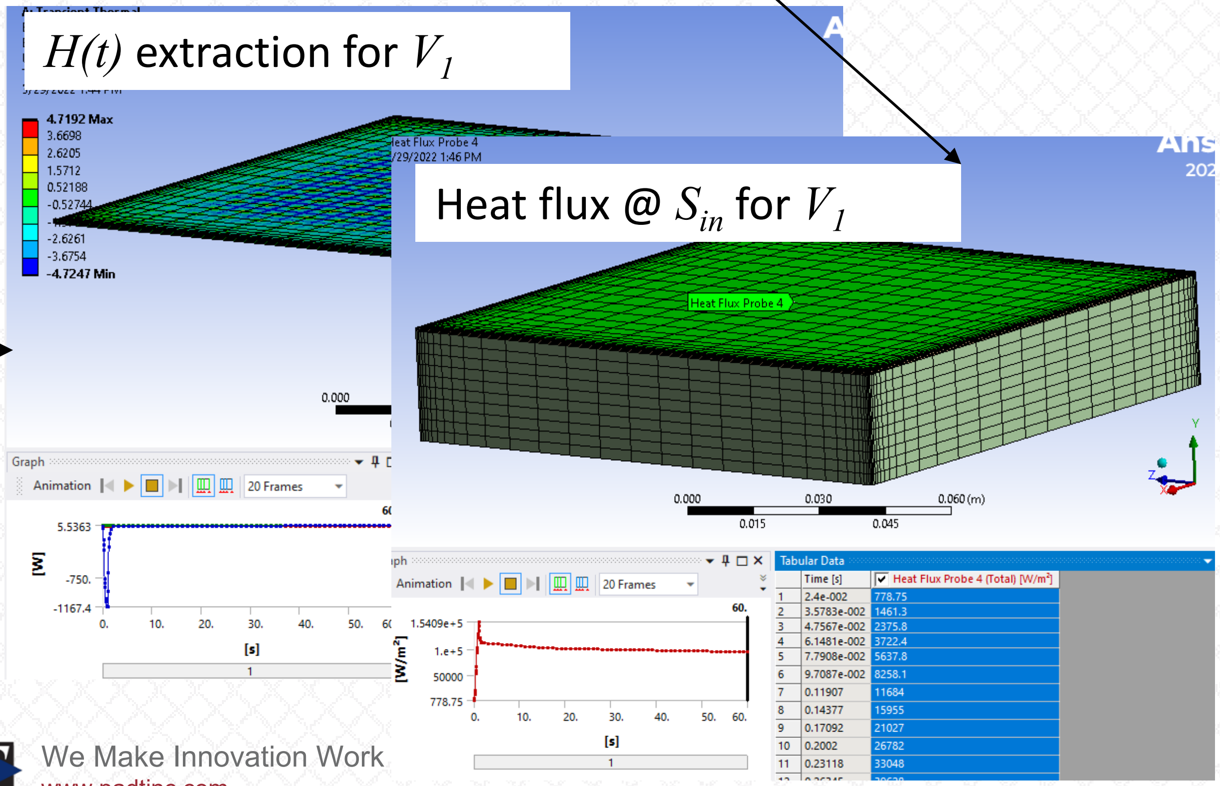


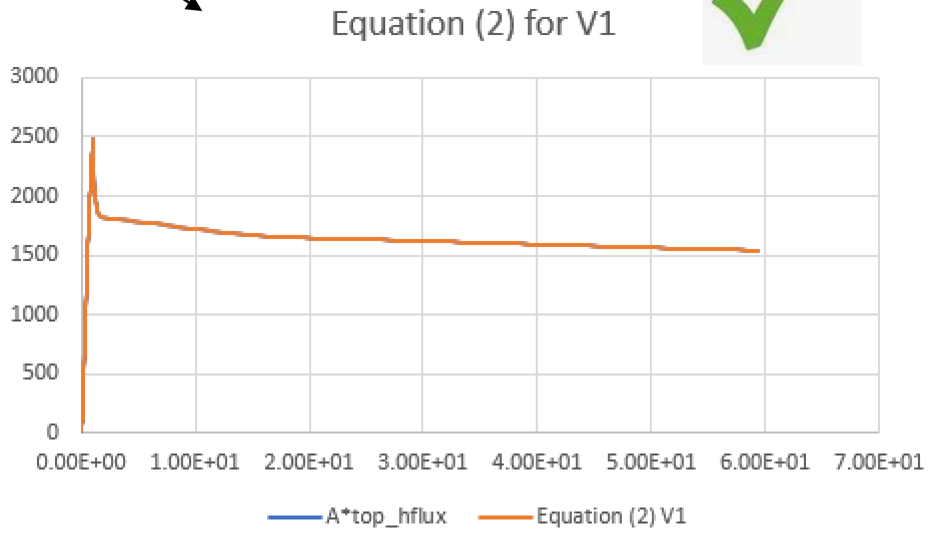




Step 4: Repeat Step 3, but this time for V1

- Extract another heat generation term (slide 15), but this time for V1
- Now, in the spreadsheet, subtract the heat generation term we just extracted from the heat flux from the interface dissipation (slide 16. Note: this now S_{out} for V_1) per Equation (2)
- Place another heat flux probe at the top of V_1 and multiply this by A_{in} (in the spreadsheet)
- In the spreadsheet, compare the two results. They should match!





Final Notes

- The transient equlibrium checks (see slide 4) we are proposing provide a 'heat balance' check for the S_{in} surface of a control volume of the user's choosing
- Note that we have been using the heat flux probe results with the 'Total' option. This was done intentially to avoid the fact that ANSYS' heat generation sign convention is opposite ours (I'm pressed for time)
- Users who want to be more careful and precise should use the appropriate vector component option when exttracting heat flux, and then simply remember to add H(t) per Equation (2)
- The transient equilibrium checks rely crucially on ANSYS heat flux results. These are obtained internally through element reaction calculations, which will, in general, be mesh-dependent
- This is especially important in thermal transient calculations where temperature gradients are not constant. In particular, if users change the mesh in this example to a linear one, they will see discripencies in some of the estimates –especially the heat flux at the sink! this happens because the gradient is almost zero there (see slide 11), and a linear mesh simply cannot resolve such a gradient with high accuracy.
- But there are more checks that users should perform. For instance, when one extracts the heat flux from a model's *convection* sink surface, one should compare that to a calculation of $hA(T-T_{\infty})$ directly. The reason this is non-trivial is that these are two calcuations are done differently and should match (the heat flux probe extracts heat flux directly from nodes and elements).