Ansys 17.2 Thermal Verification

UNIVERSITY OF CENTRAL FLORIDA

Selvin Reyes

Abstract

Florida Turbine Technologies (FTT) relies on the software ANSYS Mechanical to perform structural and thermal analyses of turbine engines. It is important for FTT to upgrade ANSYS versions periodically to stay up to date and also because of ANSYS lack of support over older versions. Before upgrading, it is necessary for a verification process to take place and check that key features and capabilities are working similar if not better than older versions. Being a heat transfer engineer, the responsibility of performing thermal verification on the new 17.2 Ansys version was only practical. This project involved performing thermal analyses on a 2 stage turbine model that was modeled from scratch where conduction, convection, advection, and radiation forms of heat transfer are applied. Fictitious aerodynamic boundary conditions and rule of thumb heat transfer coefficients (HTCs) were provided to avoid use of proprietary information. The success criteria for the project was to compare metal nodal temperature of all nodes for both ANSYS versions 17.2 (New) and 14.5 (Current) and for the maximum temperature difference (max ΔT) to be no greater than 1°F. The max ΔT obtained was 0.135 °F. Oscillation limit (OSLM) was to be compared as well which expresses the stability of results and it had to be less than 1 which for the most part remained under 0.5 and was over 1 for no longer than 0.1 seconds. For this analysis, version 14.5 is taken as the baseline version and is assumed to be reliable.

Project Requirements & Overview

2D Model

- Conduction
  - Axisymmetric
  - Plane with thickness
- Convection
  - W/ Xtra node
  - W/out Xtra node
  - In holes
- Radiation
  - To a node
  - To a surface
- Advection
  - W/ standard convection
  - W/ multiple fluids
  - W/ windage and surface heat generation
  - W/ Radiation
- Contact
  - High TCC
  - Low TCC

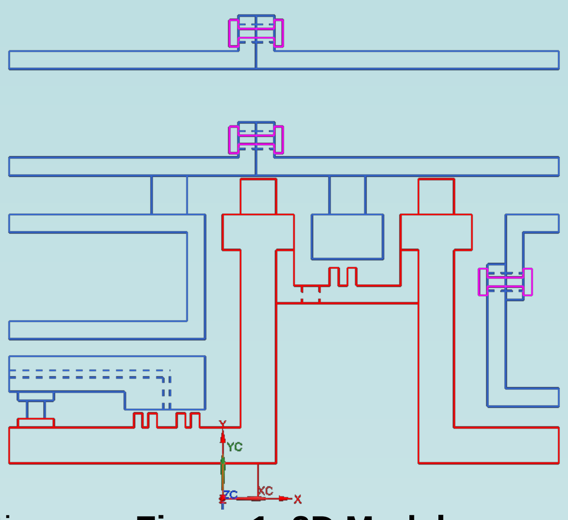

Figure 1: 2D Model

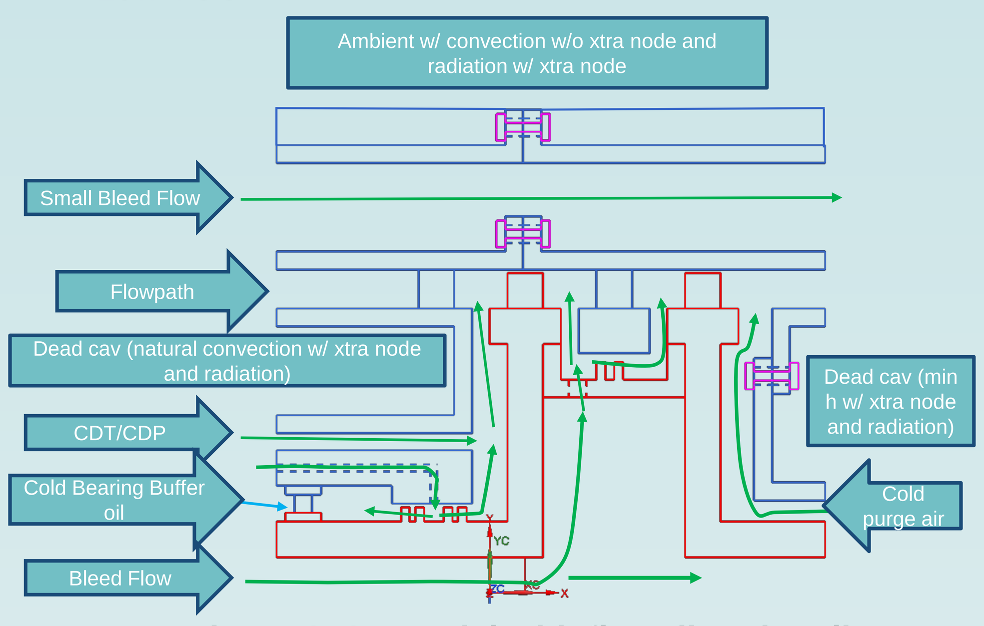

Figure 2: 2D Model with flow directionality

Building Up The Model

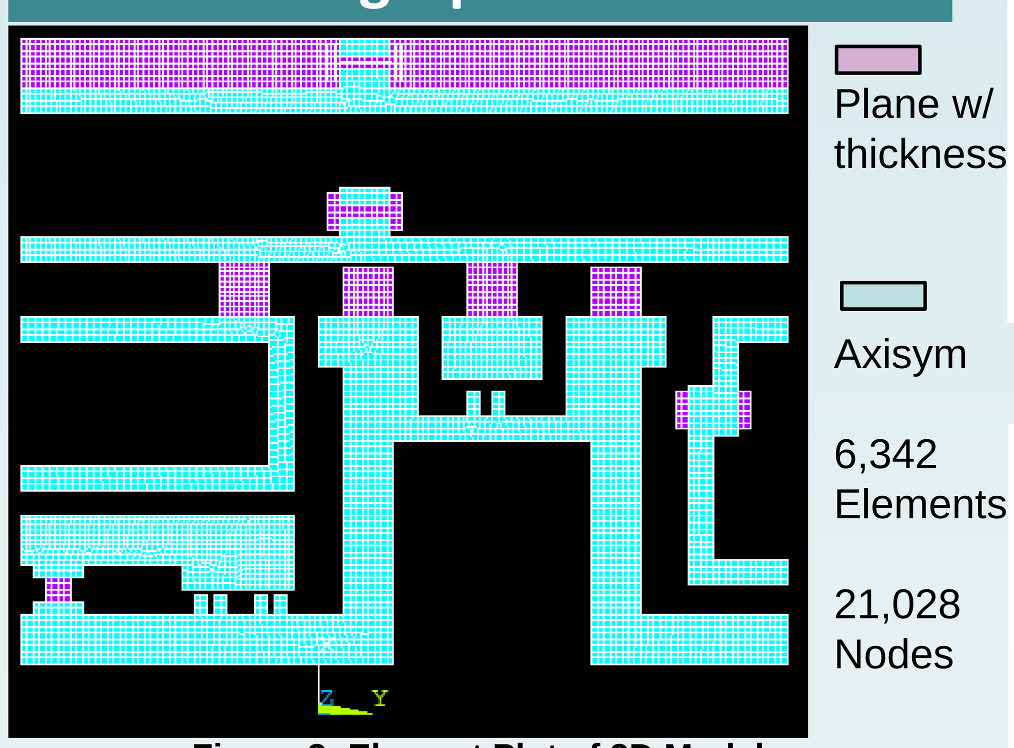

Figure 3: Element Plot of 2D Model

Element Type

PLANE77

Elements for metal components

Axisymmetric or Plane w/ thickness

SURF151

To model convection/advection on surfaces w/ or w/o extra node

Radiation to a node

SURF152

Same as surf151 but to plane w/ thickness elements

FLUID116

To model fluid flow

TARGE169 & CONTACT172

To specify contact resistance b/w parts

LINK33 & MATRIX50

To model radiation in an enclosure

Material

Material properties were given

2 – air, 3 – oil, 718 – Nickel Based Alloy

Density [ρ]

Specific Heat [cp]

Viscosity [μ]

Thermal Conductivity [k]

Emissivity [ε]

Thermal Expansion [1/°F]

Real Constants

Thickness

Angular velocity

Swirl factor

Gravitational constant

Joule constant

Stephan Boltzmann constant

View/Form factor

Etc.…

Macro for BCs

Boundary conditions have to change throughout the simulation run in order to mimic thermal behavior from the start of the engine to the end. A macro was coded in ANSYS Parametric Design Language (APDL) to change various parameters such as

- Flow rate
- Upstream temperature to all entry fluid nodes
- Heat generation on fluid nodes and bearing
- Bulk temperature and HTC values to all SURF 151 and SURF 152 elements
- Speed in rad/s

These parameters are dependent on whether to run transiently or steady state (SS) and whether the engine was at full speed 20,000 RPM or if dwelling at 0 RPM. A Do-Loop was utilized to define every BC accordingly. Figure 4 Displays the load steps that are running either at transiently or SS.

Step 1; 0.01 s
SS 0 RPM
Step 2; 30 s
End Transient Accel
Step 3; 3629.999 s
End Transient Dwell @ N RPM
Step 4; 3630 s
SS ADP
Step 5; 3660 s
End Transient Decel
Step 6; 7260 s
End Transient Dwell @ 0 RPM
Step 7; 7261 s
SS ADP Accel
Step 8; 7264 s
End Transient Estop Decel
Step 9; 10864 s
Post Estop Transient Dwell @ 0 RPM

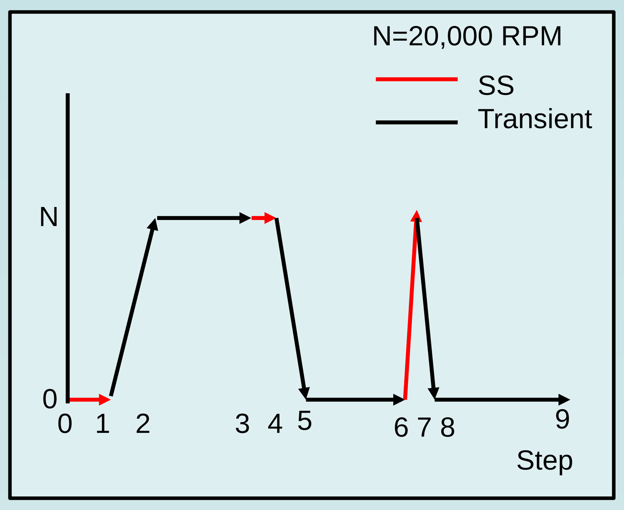

Figure 4: SS/Transient

Metal Temp at Load Steps

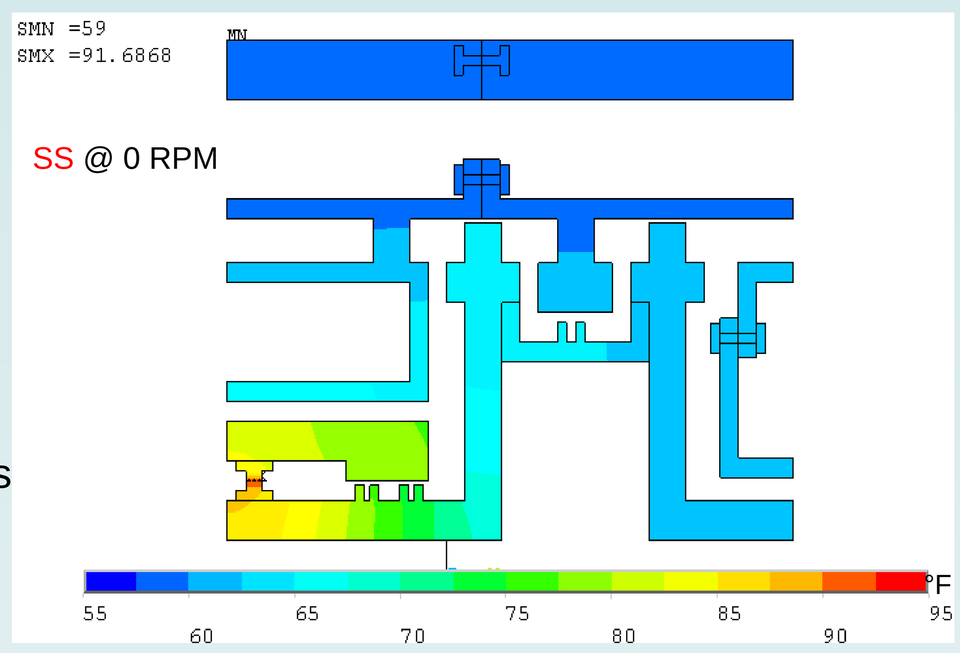

Figure 5: Load step 1 @ 0.01 Seconds

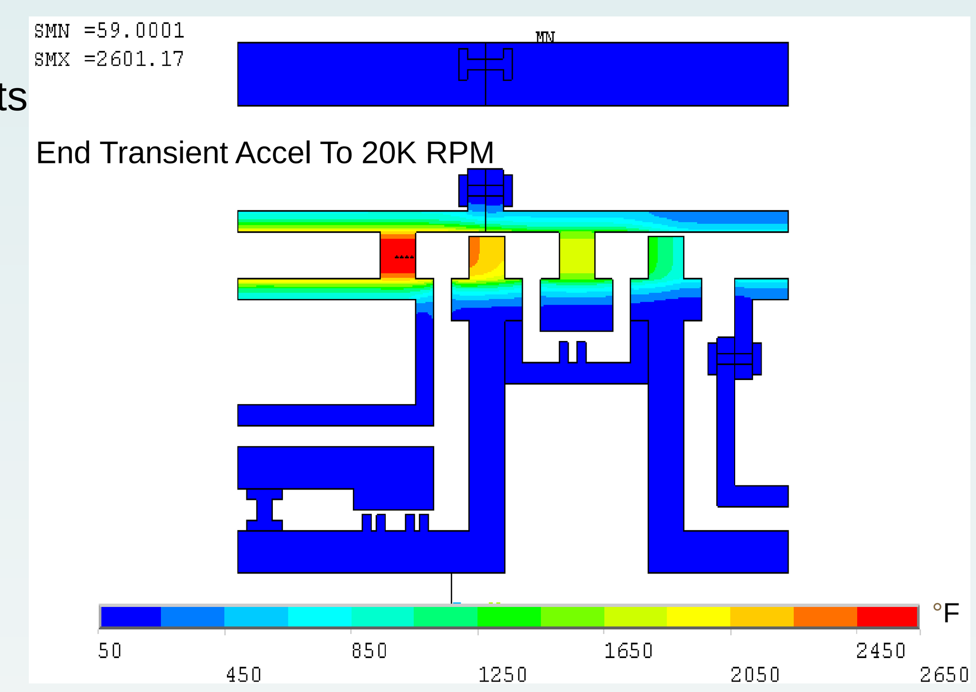

Figure 6: Load step 2 @ 30 seconds

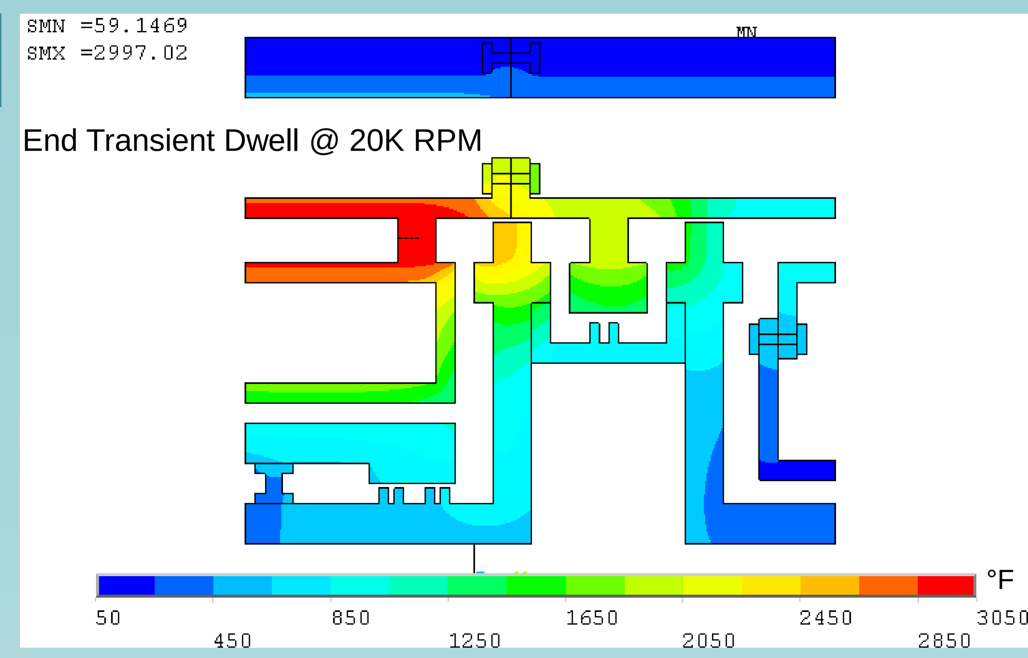

Figure 7: Load step 3 @ 3629.999 Seconds

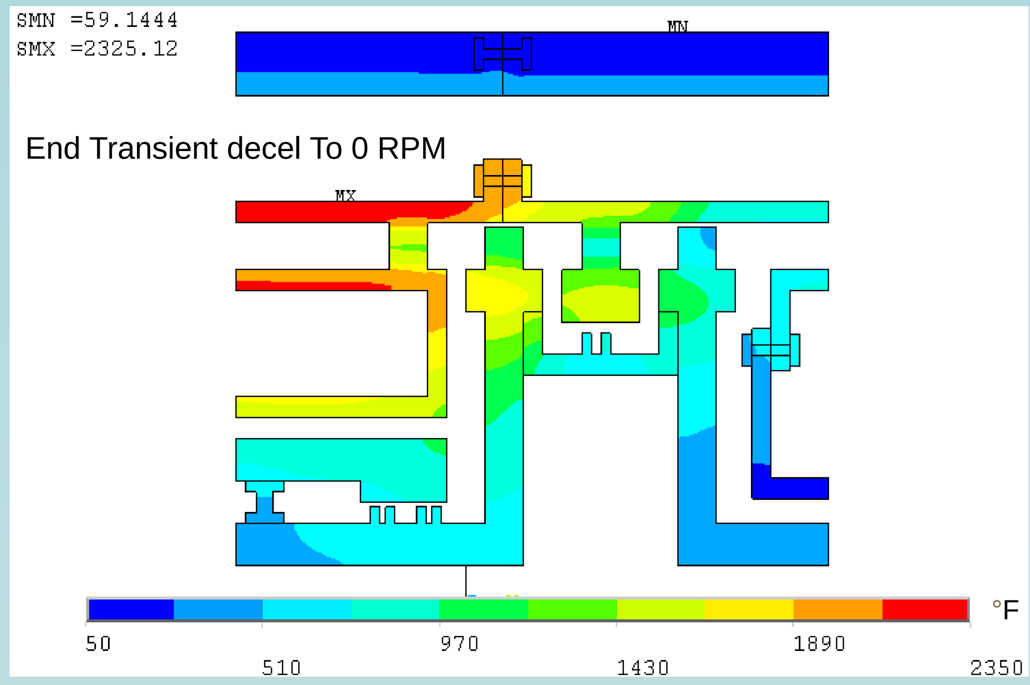

Figure 8: Load step 5 @ 3660 Seconds

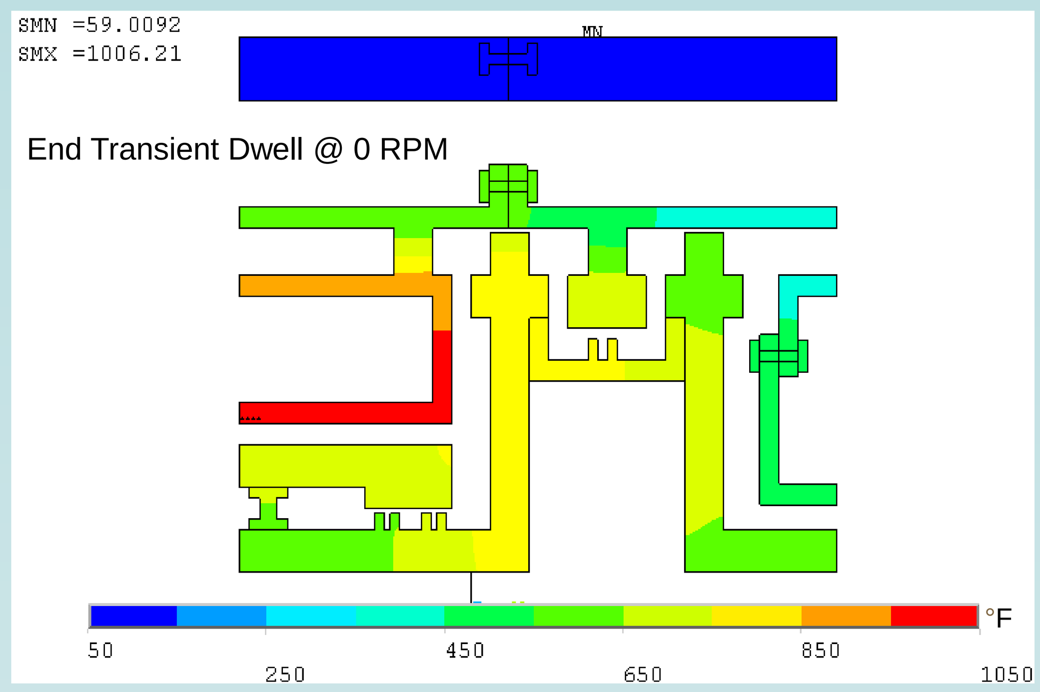

Figure 9: Load step 6 @ 7260 Seconds

Hand Calculations

In order to ensure that the output metal nodal temperatures are correct in the sense that the software ANSYS is performing correct heat transfer calculations and applying the correct boundary conditions, hand calculations were done in different locations of the model for verification purposes. 1D SS hand calculation was investigated between the 2nd rotor and right cavity to get temperatures close to what ANSYS got even though the temperature distribution was more 2D than 1D as seen on figure 10. 1D Transient calculations were also performed at a bolt nut and first vane where Biot number is less than 0.1 and 1 respectively which lumped capacitance is assumed. Figure 12 calculated temperature over time for the bolt head and figure 14 for the first vane.

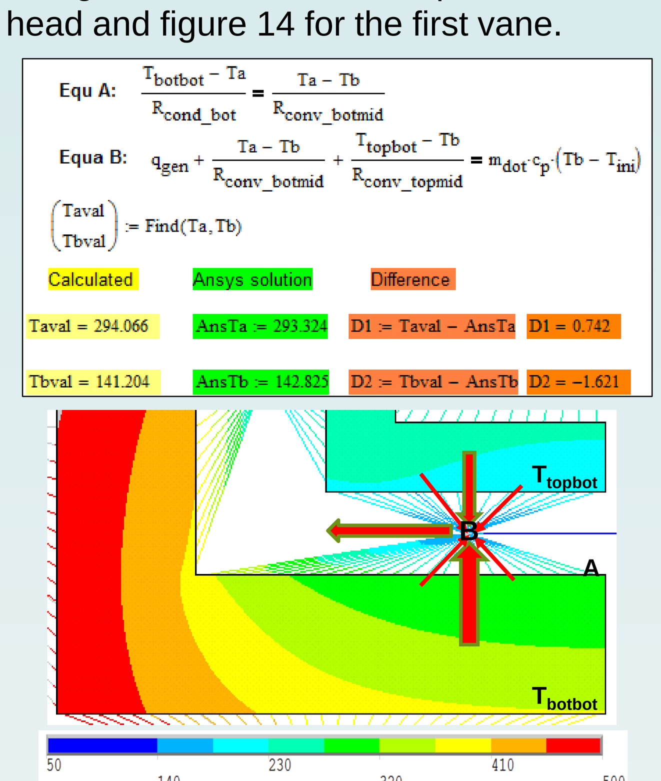

Figure 10: 1D SS Calculation

$$T = T_\infty + (Ti - T_\infty) * e^{-(t/\tau)}$$

$$\tau = \frac{\rho * V * cp}{h * As}$$

$$T_{eff} = \frac{T_1 * A_1 + T_2 * A_2}{A_1 + A_2}$$

$$\frac{1}{U * Atot} = \frac{1}{h_1 * A_1} + \frac{1}{TCC * A_2}$$

TCC = thermal contact conductance

$$\tau_{eff} = \frac{\rho * V * cp}{U * A_{tot}}$$

Equations used for transient calculations

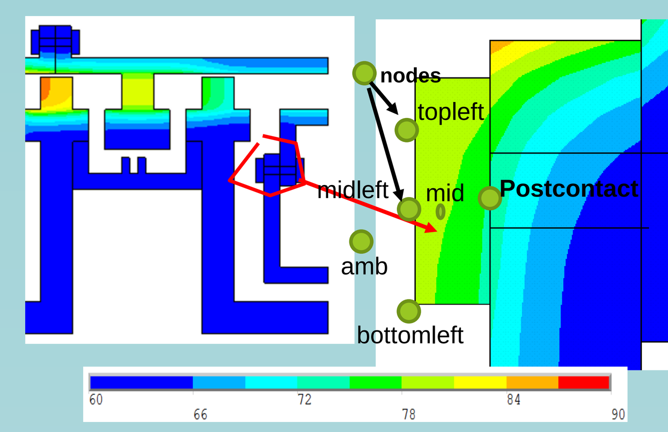

Figure 11: Transient Calculation of Bolt Nut

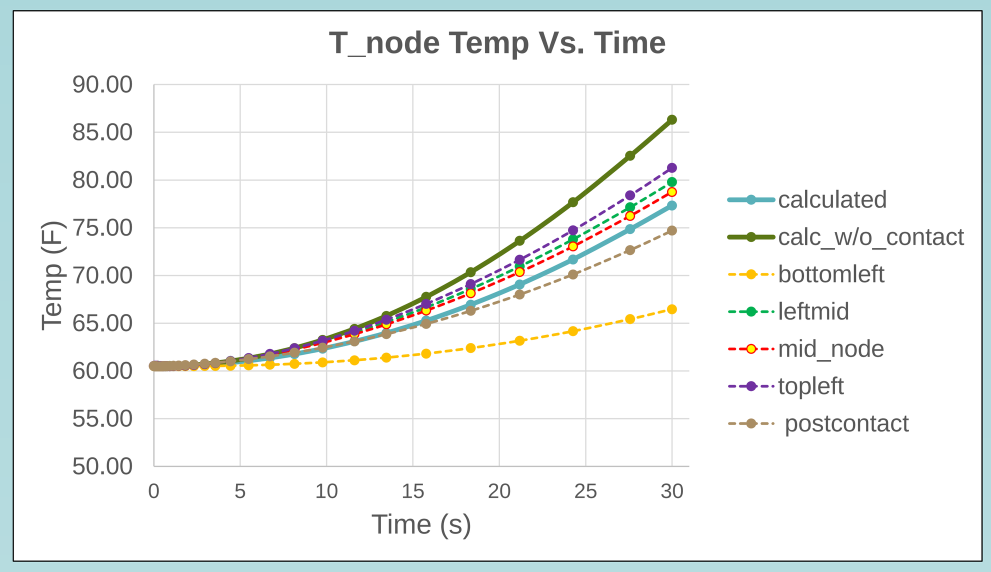

Figure 12: Calculated Temp Vs. ANSYS of Bolt Nut

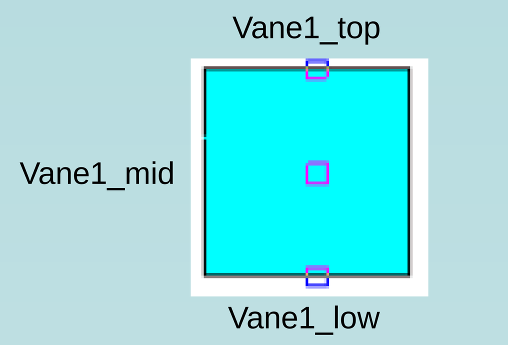

Figure 13: Nodes Evaluated for 1st Vane

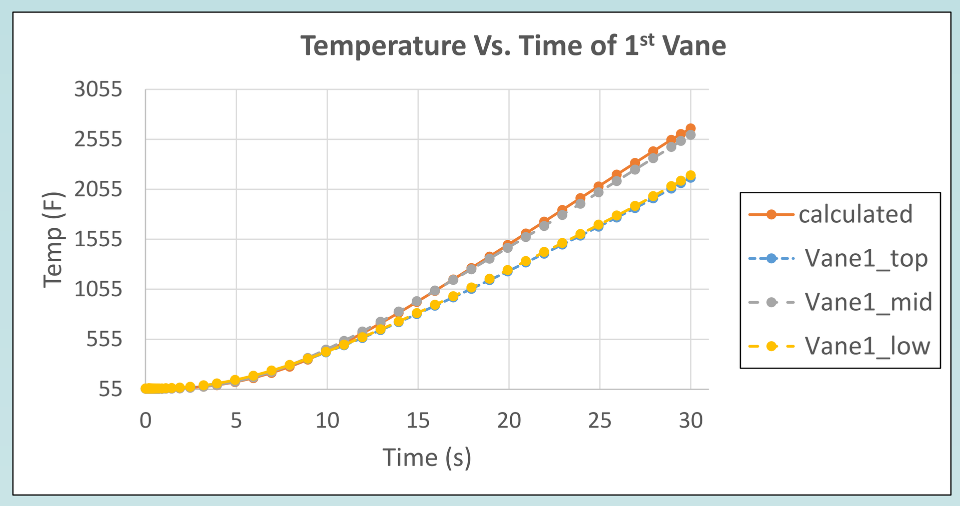

Figure 14: Calculated Temp of 1st Vane

Post Processing & Conclusion

The post processing of the thermal analysis involved creating 2 more macros making a total of 3 macros. The 2nd macro was to obtain the nodal metal temperature of every node and create a **21,067 x 387** array outputting its respective temperature, node numbers and time sub step for each version. The 3rd macro would then take the temperature difference between both versions from the previous outputted arrays, find the max ΔT, and link that maximum to its corresponding node and time sub step. If the time sub steps are not the same between both versions, then the code would interpolate the temperature of a node in version 17.2 to match the time sub step of the baseline version 14.5.

In conclusion, the maximum temperature difference obtained after comparing every nodal temperature was 0.135°F. The OSLM versus time as seen on figure 16 was obtained from within the software expressing the stability of the results. Values remained under 0.5 for most times and was greater than 1 with a duration that never exceeded 0.1 seconds. These would occur at the first sub step of a transient load step run where the previous run was at SS which is usually expected to occur. Comparing figure 15 to figure 16, it is observed that the greatest ΔT occurs when the OSLM is greater than 1. In the end, thermal verification of the 2D model was deemed successful as the success criteria were met and future verifications of ANSYS versions should now be easier for FTT to use when upgrading.

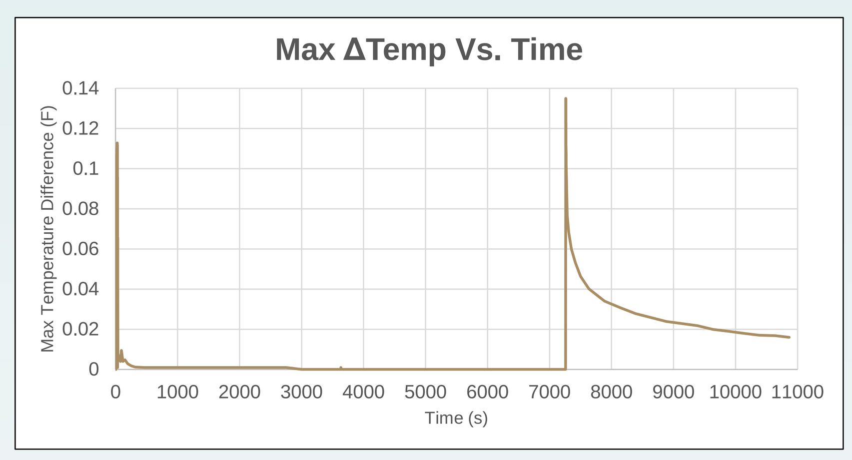

Figure 15: Max ΔTemp Vs. Time

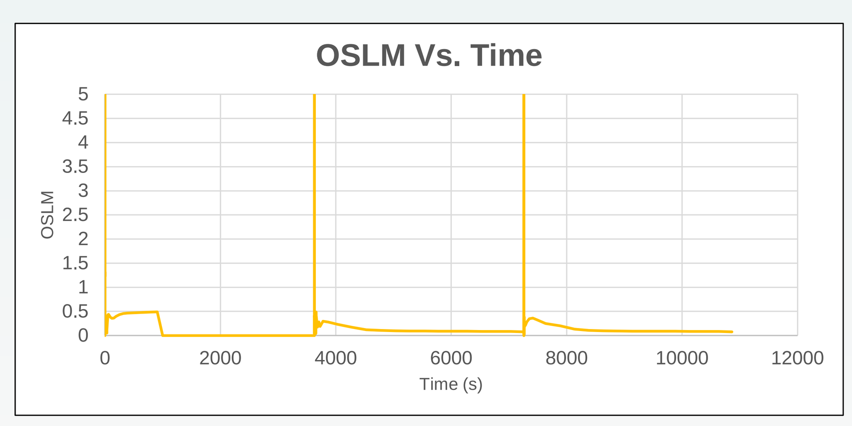

Figure 16: OSLM Vs. Time