Transient Thermal Analysis for Process Optimization: A Comparative Study of Lumped Capacitance Modeling and Finite Element Simulation for Automated Hot Nut Insertion in Polycarbonate Components

Self-Initiated Study Based on my Internship Experience at SMCV (Stage PFA)

Author: Rafi NASRALAH

Affiliation/Status: Final Year Student in Electrical Engineering and Industrial Systems' Control | **ENSET**

Abstract

This self-initiated research documents the transition from theoretical thermal analysis to advanced **Finite Element Analysis (FEA)**, motivated by practical limitations observed during the design of an automated hot-press insertion machine for nuts into polycarbonate components at SMCV.

The initial design phase necessitated a rigorous determination of the optimal heating time and source temperature to ensure reliable hot-insertion while preventing thermal damage to the polycarbonate part. This was first addressed using the **Lumped Capacitance Model (LCM)**, which provided a simplified theoretical baseline for cycle time estimation.

However, the inherent simplifications of the analytical approach—which neglects spatial temperature gradients and complex 3D heat transfer paths—prompted a direct exploration into advanced FEA. This investigation focused on developing a detailed **transient thermal study using ANSYS** to characterize the time-dependent heat transfer mechanism between the heating element and the nut.

The objective is to accurately predict the thermal evolution curve of the nut, providing the necessary data for fine-tuning the automated press's operational cycle. This investigation not only validates the initial theoretical assumptions but, more importantly, establishes a precise and robust foundation for thermal parameter control, significantly contributing to consistent product quality and process efficiency.

Keywords

Thermal Simulation, Hot Nut Insertion, Transient Thermal Analysis, Lumped Capacitance Method (LCM), Finite Element Analysis (FEA), Process Optimization, Polycarbonate, ANSYS, Heat Transfer.

List of Figures

List of Tables

Contents

I. Context, Motivation, and Scope of Exploration

A. Background and Initial Problem Statement

The core of this investigation originates from a practical quality and efficiency challenge identified within the production workshop at **SMCV**. Specific plastic components require the precision hot-insertion of brass nuts prior to final assembly.

The established process utilized a **pneumatic press** characterized by **suboptimal temperature control** and **inadequate fixturing** for the nuts. This manual setup inevitably led to significant **variability in parts quality**, necessitating time-consuming manual rework of poorly inserted nuts. This variability not only posed a quality risk but also resulted in **process inefficiency** and a direct reduction in workshop productivity.

B. The Initial Engineering Response

To address this critical bottleneck, the primary engineering objective was the design and implementation of an **Automated Hot-Insertion Press**. The new system was required to execute the task **reliably, repeatedly, and seamlessly** integrated into the production cycle (Tooling Design: Autodesk Inventor).

A prerequisite for reliable automation was the determination of critical thermal parameters. Consequently, a preliminary **theoretical thermal analysis** was conducted. This study employed the **Lumped Capacitance Model** to estimate the essential **heating time** required to bring the nut to the target insertion temperature (the softening range of the polycarbonate), while rigorously ensuring the heat source temperature did not exceed the material's degradation threshold.

The results of this analytical phase provided necessary initial inputs for the design of the heating element and contributed to the estimation of the operational **cycle time**.

C. Motivation for Advanced Thermal Exploration

The inherent limitations of the theoretical LCM—specifically its inability to account for spatial temperature gradients or the transient heat flux into the surrounding air and fixturing—prompted this deeper, self-directed technical investigation. A highly accurate understanding of the time-dependent temperature evolution ($T(x,t)$) is essential to two primary objectives:

- Process Efficiency: Precisely calculate the minimum time required to reach the insertion temperature, thus minimizing the automatic cycle time.
- Part Integrity: Ensure that the heat source temperature and duration do not cause the nut surface to exceed the PC's critical softening temperature, preventing material damage and structural failure.

D. Objectives and Scope of this Exploration Report

This report documents the methodical transition to numerical simulation, adhering to the principles of the scientific method.

The specific objectives are:

1. **Modeling and Preparation:** To accurately construct the 3D geometry of the nut and the heating element interface (Fusion 360).
2. **Core Objective - Transient Analysis:** To conduct a comprehensive **Transient Thermal Analysis** using **ANSYS** to precisely map the evolution of the nut's temperature over time. This includes determining the **exact heating curve** under defined boundary conditions.
3. **Validation and Optimization:** To use the results of the transient simulation to **validate or refine** the time-cycle parameter initially derived from the Lumped Capacitance Model, thereby establishing a precise thermal control strategy for the automated press.

The scope is strictly limited to the **thermal characterization of the hot-insertion process**, focusing on the **heat transfer from the source to the nut**, and its subsequent temperature evolution during the heating phase.

II. Theoretical Thermal Analysis: The Lumped Capacitance Method (LCM)

A. Principle and Applicability of the LCM

The initial phase of the thermal design utilized the **Lumped Capacitance Method (LCM)** to rapidly estimate the transient heating time of the insert nut. The LCM, a cornerstone of simple transient heat transfer analysis, allows the complex partial differential equations governing heat conduction to be simplified into a more manageable ordinary differential equation.[1]

The fundamental hypothesis of the LCM is that the temperature within the solid body is **spatially uniform** at any given instant during the heating or cooling process, meaning:

$$T(x, y, z, t) \approx T(t)$$

This idealization is physically valid only when the resistance to heat conduction *within* the solid is negligible compared to the resistance to heat transfer *at the solid's surface* (i.e., convection/radiation).

B. Modeling Assumptions and Boundary Conditions (LCM)

To facilitate the application of the Lumped Capacitance Method (LCM) and to prepare the groundwork for the subsequent numerical simulation, several key assumptions regarding the system's geometry, materials, and thermal environment were made:

a. System Configuration and Component Geometry

The thermal system involves three primary components: the resistive collar (heater), the brass transfer tube, and the insert nut.

- **Insert Nut:** The nut is modeled as a hollow cylinder made of Brass, featuring both external and internal grooves. The LCM, for initial estimations, approximates this complex shape into a single lumped thermal mass defined by its total volume and surface area.
- **Heating Object (Transfer Tube):** The Brass transfer tube serves as the primary thermal interface. This tube is also cylindrical and is designed to house the nut with the tightest possible fit to maximize the heat transfer surface area, while still allowing the nut's vertical descent during the automated cycle.

- **Heating Source:** The heat is generated by a resistive collar surrounding the brass transfer tube.

b. Thermal Interface Simplifications

The simplified LCM analysis required major assumptions regarding the heat exchange mechanism:

- **Primary Heat Transfer:** In the LCM, the heat transfer from the brass tube to the nut is the critical factor. This interface is assumed to be the sole active heating surface, dominating the thermal response.
- **Effective Contact:** Despite the complex surface contact (not perfect due to the necessary clearance for descent, and the grooves), the heat transfer across the interface is simplified and quantified by a single, constant equivalent heat transfer coefficient (h). This coefficient attempts to account for the conduction through microscopic contacts and the convection across the tiny air gap.
- **Isothermal Source (T_∞):** The surrounding brass transfer tube is assumed to be an **infinite thermal reservoir** that maintains a constant and uniform temperature (T_∞) throughout the rapid heating cycle. This assumes the resistive collar is powerful enough to prevent any significant temperature drop in the tube as it heats the nut.

c. Material and Operational Assumptions

- **Constant Properties:** All thermophysical properties—density, specific heat, and thermal conductivity—for both the brass nut and the brass transfer tube are assumed to **remain constant** over the expected operational temperature range.
- **Uniform Initial State:** The nut begins the heating process at a **uniform initial temperature**, corresponding to the ambient workshop conditions.

These necessary simplifications in the LCM—particularly the use of a single, estimated coefficient h and the neglect of temperature gradients within the nut due to the **specific, tight-fit cylindrical geometry**—are the core reasons driving the need for the spatially resolved, time-dependent simulation using ANSYS.

C. Justification for using LCM

The **Lumped Capacitance Method (LCM)** was selected for the initial thermal analysis due to its simplicity and ability to provide a rapid, first-order approximation of the system's thermal transient behavior. Its application is mathematically justified when the temperature gradients within the object being heated (the screw insert) are considered negligible compared to the temperature difference between the object and its surroundings (the heating element).

The primary condition for the validity of the LCM is determined by the **Biot number:**

$$B_i = \frac{hL_c}{k}$$

- h is the convective heat transfer coefficient.
- L_c is the characteristic length of the object (Volume/Surface Area).[2]
- k is the thermal conductivity of the object's material (the metallic screw insert).

The premise dictates that the LCM is applicable when: $B_i \leq 0.1$

This means that the resistance to heat transfer within the solid object (which is inversely proportional to k) is significantly smaller than the resistance to heat transfer at the surface (convection or contact).

Justification for the Screw Insert:

For the Brass made screw insert:

- $\textit{The screw insert is modeled as a } \boldsymbol{\textit{hollow cylinder}}$

$$\text{Ø}\, ext = 6\, mm, \text{Ø}\, int = 3{,}3\, mm, height = 7\, mm$$

- $Volume$:

$$V = \pi(r_{ext}^2 - r_{int}^2)h \approx 1{,}38 \times 10^{-7} m^3$$

- Heat Exchange Area ：

$$A = A_{Lat,int} + 2A_{base} = 2\pi h r_{int} + 2\pi(r_{ext}^2 - r_{int}^2) \approx 1.12 \times 10^{-4} m^2$$

- Characteristic Length:

$$L_c = V/A \approx 1.232 \times 10^{-3} m$$

- Convective heat transfer coefficient:

$$h = 2000W \cdot m^{-2} \cdot K^{-1}$$

- Thermal conductivity of the object's material:

$$k = 120\, W \cdot m^{-1} \cdot K^{-1}$$

- $Bi = 0.02$

Since Bi $\approx 0.02 < 0{,}1$ the condition for the Lumped Capacitance Method is **met**. This theoretically justifies the use of the LCM to obtain a *first approximation* of the time-dependent temperature, serving as an initial benchmark for the automated press's cycle time.

D. Key Calculations: Determination of Source Temperature and Cycle Times

This section utilizes the Lumped Capacitance Method (LCM) to derive the critical time parameters for the automated cycle: the time required for heating the screw insert and the subsequent time needed for cooling. The analysis is governed by the thermal time constant of the metallic insert.

a. Material Properties and Target Window

The design constraint requires the nut's final temperature to fall within the polycarbonate's softening range, T_{target}, but never exceed the polymer's thermal degradation limit, $T_{\max}$.

Parameter	**Value**	**Unit**	**Note**
Density of Brass	8500	$\text{kg} \cdot \text{m}^{-3}$	ρ
Specific Heat of Brass	380	$\text{J} \cdot \text{kg}^{-1} \cdot \text{K}^{-1}$	c_p
Volume	1.38×10^{-7}	m^3	V
Heat Exchange Area	1.45×10^{-4}	m^2	A_s\ (External area)
Initial Temperature	25	°C	Workshop ambient
Minimum Target Temperature	150	°C	Minimum for PC softening

Table 1: Material Properties

b. Thermal Time Constant τ

The thermal time constant τ, defines the intrinsic rate at which the screw insert changes temperature, based on its material properties (ρ, V, c_p) and the rate of heat exchange (h, A_s) Using the previously calculated parameters (Section 2.2):

$$\tau_t = \frac{\rho V c_p}{h A_s}$$

Parameter	Value Used	Unit
Time Constant	$\tau_t \approx 1.54$	s

Table 2: Numerical application for the thermal time constant

This time constant is used in the general LCM transient temperature equation:

$$\frac{T(t) - T_\infty}{T_i - T_\infty} = e^{-t/\tau_t}$$

Where T_i is the initial temperature, $T(t)$ is the temperature at time t, and T_∞ is the steady-state (ambient/source) temperature.

c. Case 1: Heating Phase (Determining t_{heat})

The heating phase determines the time the screw must remain in contact with the hot element to reach the target insertion temperature, T_{target}.

- **Initial Temperature:** T_{ambient}
- **Target Temperature:** T_{target}
- **Source Temperature:** T_{Source} (The temperature of the heating element, T_S)

The required **Heating Time (t_{heat})** is calculated by solving the LCM equation for t:

$$t_{\text{heat}} = -\tau_t \ln\left(\frac{T_{\text{target}} - T_{\text{Source}}}{T_{\text{ambient}} - T_{\text{Source}}}\right)$$

This t_{heat} establishes the **minimum thermal dwell time** required for the automated cycle.

This expression reveals the **critical dependence of the cycle time ($t_{heating}$) on the source temperature** (T_∞). This equation will be implemented in a programming environment to perform a **sensitivity analysis** by varying T_∞ and graphically determining the optimal operating point that minimizes $t_{heating}$ while respecting the T_{max} constraint.

d. Case 2: Cooling Phase (Determining t_{cool})

The cooling phase analysis determines how long a heated screw can be stored in the autofeeder system or remain idle before its temperature drops below the acceptable insertion threshold. Since the screw is cooled solely by surrounding air, the heat transfer coefficient drastically decreases.

$$\tau_{t,\text{cool}} = \frac{\rho V c_p}{h_{\text{cool}} A_s} = \frac{(8500 \times 1.38 \times 10^{-7} \times 380)}{(15 \times 1.45 \times 10^{-4})}$$

$$\tau_{t,\text{cool}} \approx 205 \text{ s}$$

- **Initial Temperature:** T_{Hot} (The maximum temperature the screw reaches in the feeder system, often equal to T_{Source})
- **Target Temperature:** T_{Loss} (A specified temperature drop/loss limit)
- **Ambient Temperature:** T_{Ambient} (The temperature of the surrounding air/components, T_A)

The required **Cooling Time (t_{cool})** to reach a specified temperature T_{Loss} is:

$$t_{\text{cool}} = -\tau_t \ln\left(\frac{T_{\text{Loss}} - T_{\text{Ambient}}}{T_{\text{Hot}} - T_{\text{Ambient}}}\right)$$

This calculation ensures that the system can maintain the heated inserts within the operational temperature window, T_{target} , for the duration of any necessary automatic cycle pauses.

E. Synthesis for MATLAB Sensitivity Analysis

The analytical framework developed through the Lumped Capacitance Method (LCM) for both the heating and cooling phases is synthesized into two principal equations, which are then used for sensitivity analysis in a computational tool. This allows for a robust determination of the operational window

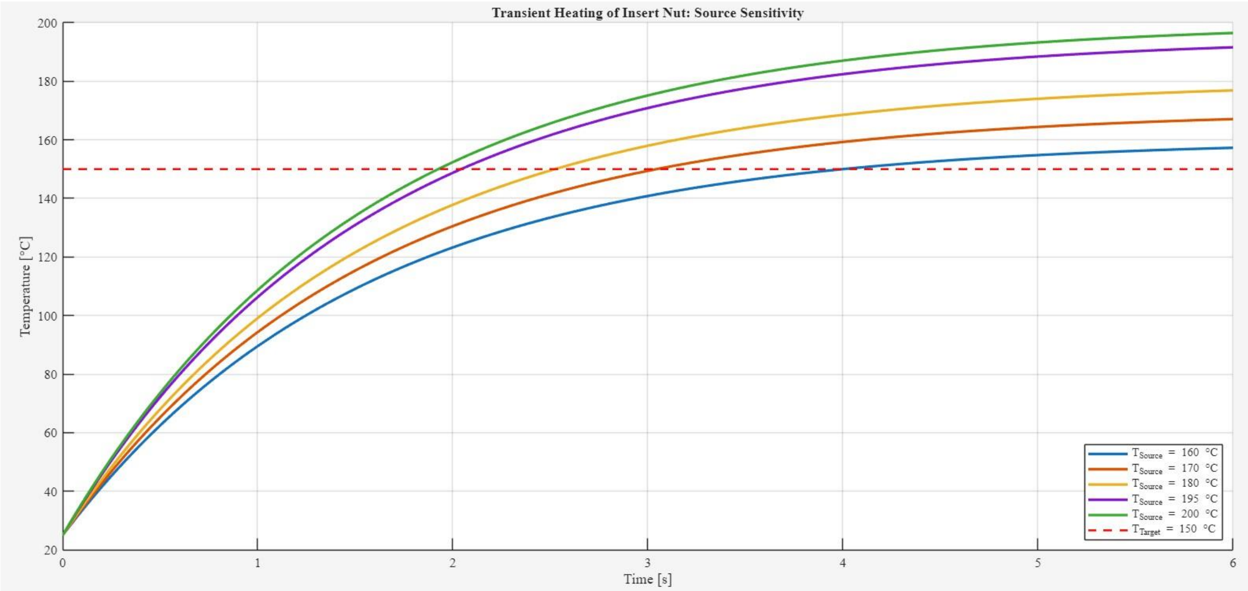

Figure 1: Transient Heating of Insert Nut

Key Takeaways (Remarks) - Heating Phase

T Source (T∞) [°C]	Time heating [s] to 150 °C	T∞ − T target [°C]
160	$\approx \mathbf{4.00}$	10
170	$\approx \mathbf{2.89}$	20
180	$\approx \mathbf{2.30}$	30
195	$\approx \mathbf{1.83}$	45
200	$\approx \mathbf{1.72}$	50

Table 3: Key values of the Heating Phase

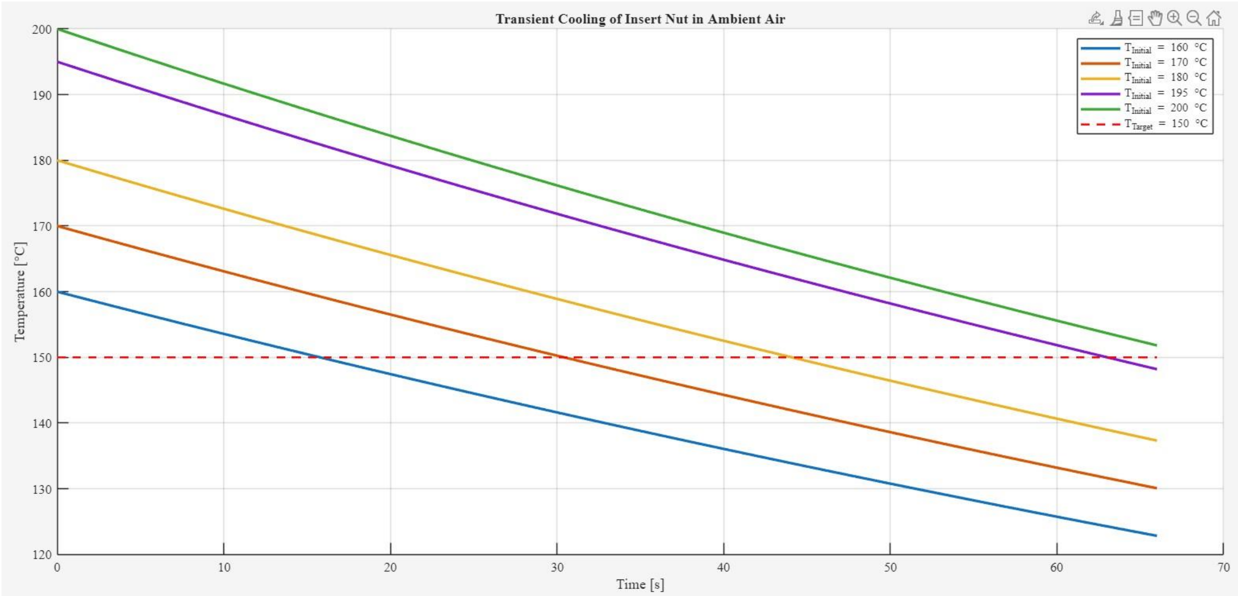

Figure 2: Transient Cooling of Insert Nut in Ambient Air

Key Takeaways (Remarks) - Cooling Phase

T Initial (T∞) [°C]	Time before reaching 150 °C
160	$\approx \mathbf{25.2}$ **s**
170	$\approx \mathbf{46.6}$ **s**
180	$\approx \mathbf{60.4}$ **s**
195	$\approx \mathbf{74.9}\ \boldsymbol{s}$
200	$\approx \mathbf{79.7}$ **s**

Table 4: Key values of the Cooling Phase

F. Design Impact: Direct Influence on the Automatic Cycle Time (t_{cycle})

The theoretical analysis of the heating and cooling transients directly quantified the operational constraints of the automated press. The primary objective was to define the **minimum total cycle time** (t_{cycle}) by establishing the critical time required for the nut to reach the **Optimal Insertion Temperature** $(\boldsymbol{T}_{\text{opt}})$.

a. Operational Definition of Cycle Time

We define T_{opt} as the target temperature (170 °C in the operational example) located within the safe processing window $\left(T_{\text{softening}} < T_{\text{opt}} < T_{\text{degradation}}\right)$.

1. Heating Phase $\left(t_{\text{heating}}\right)$: This is the minimum necessary time the nut must spend in contact with the heating element. It determines when the nut is ready to leave the loading mechanism and enter the insertion process.

$$t_{\ heating} \text{is the time to reach } T_{\text{opt}}.$$

2. **Cooling Phase** $(\boldsymbol{t}_{\textbf{buffer}})$: This defines the **maximum permissible delay** (buffer time) allowed after the nut has reached T_{opt}. It determines the operational margin before the nut's temperature drops below $T_{\text{Low limit}}(150\ °\text{C})$, rendering the insertion unreliable.

b. Quantified Operational Example

The analysis provides concrete parameters for the press control system:

Parameter	**Value**	**Calculation Based On**	**Impact on Automation**
Optimal Temp $\left(\boldsymbol{T}_{\textbf{opt}}\right)$	170 °C	Set by process engineering ($T_{\text{seuil}} = 150$ °C)	Defines the thermal readiness of the nut.
Source Temp $(\boldsymbol{T}_{\infty})$	180 °C	Chosen to maintain a safe margin below $T_{\text{degradation}}$.	Defines the thermal driving force.
Minimum Heating Time $\left(\boldsymbol{t}_{\textbf{heating}}\right)$	4.22 s	$Time for$ T_{nut}: 25 °C → 170 °C	**Directly limits the press cycle speed.**
Maximum Holding Time $(\boldsymbol{t}_{\textbf{buffer}})$	46.6 s	Time for T_{nut}: 170 °C → 150 °C	Defines the **safety margin** of the automated buffer.

Table 5: Quantified operational examples for the heating and cooling phases

The calculated $\boldsymbol{t}_{\ heating} = 4.22$ s represents the primary constraint on the achievable **production rate** of the automated press.

G. Limitations Identified

While the theoretical analysis provided the necessary initial design parameters and established the operational windows, its reliance on simplifying assumptions introduces significant uncertainties that undermine the goal of high-precision industrial automation.

The two main limitations mandating the transition to advanced numerical simulation are:

1. **Uncertainty of the Heat Transfer Coefficient ($\boldsymbol{h}$):**
   - The entire calculation, particularly the crucial $t_{heating} = 4.22$ s, is derived using an estimated effective contact coefficient ($h = 2000\ \mathrm{W} \cdot m^{-2} k^{-1}$)
   - In reality, the thermal resistance at the tight brass-on-brass interface is highly dependent on microscopic surface roughness, contact pressure, and the presence of minute air gaps. This **contact resistance** cannot be accurately represented by a single, constant, estimated h value, introducing an unquantified margin of error in t_{heating}.
2. **Inability to Resolve Spatial Temperature Gradients:**
   - The Lumped Capacitance Model assumes the nut temperature is uniform ($T(t) only$).
   - It **cannot predict the maximum surface temperature** transmitted to the polycarbonate at the instant of insertion. If the outer surface of the nut reaches $190\ °\mathrm{C}$ while the core is only at $170\ °\mathrm{C}$, the plastic will suffer **thermal damage** (melting or discoloration), even if $T_{\text{avg}} < T_{\text{degradation}}$. The nut's complex hollow geometry and grooves can generate these gradients.

These critical limitations necessitate replacing the analytical model with a **high-fidelity Transient Thermal Finite Element Analysis (FEA)**, which can model the complex 3D conduction, the precise contact geometry, and provide a detailed spatial and temporal map of the temperature distribution. This transition formalizes the next phase of the scientific investigation.

III. Simulation Preparation: 3D Modeling and Geometric Abstraction

The transition from the 1D analytical model (LCM) to a 3D Finite Element Analysis (FEA) necessitates the creation of a precise and computationally efficient geometric representation. This chapter details the preparation of the CAD model, which serves as the foundation for the subsequent transient thermal analysis in ANSYS.

A. Objective of the CAD Preparation Phase

The primary objective of this phase is not simply to create a 3D model, but to perform **geometric abstraction**. A raw design model, created for manufacturing, often contains excessive detail (such as fillets, chamfers, or internal threads) that provides no significant thermal value but exponentially increases computational cost (mesh size and complexity).

This phase balances two competing needs:

1. **Geometric Fidelity:** Accurately capturing the key features that govern the heat transfer physics (e.g., the contact surfaces).
2. **Computational Efficiency:** Simplifying or removing features irrelevant to the thermal analysis to ensure the simulation can be solved in a reasonable timeframe.

The CAD preparation was performed using **Fusion 360**.

B. Component Modeling and Simplification

The simulation focuses on the "control volume" defined in the LCM: the nut and its immediate heating source, the brass transfer tube.

a. Overview of the Auto-Feed and Heating System

To understand the thermal problem, we must first locate the components within the mechanical assembly. The **Auto-Feed System** (illustrated in **Figure 3**) is a pneumatic mechanism designed to transport a single nut from a reservoir to the insertion point.

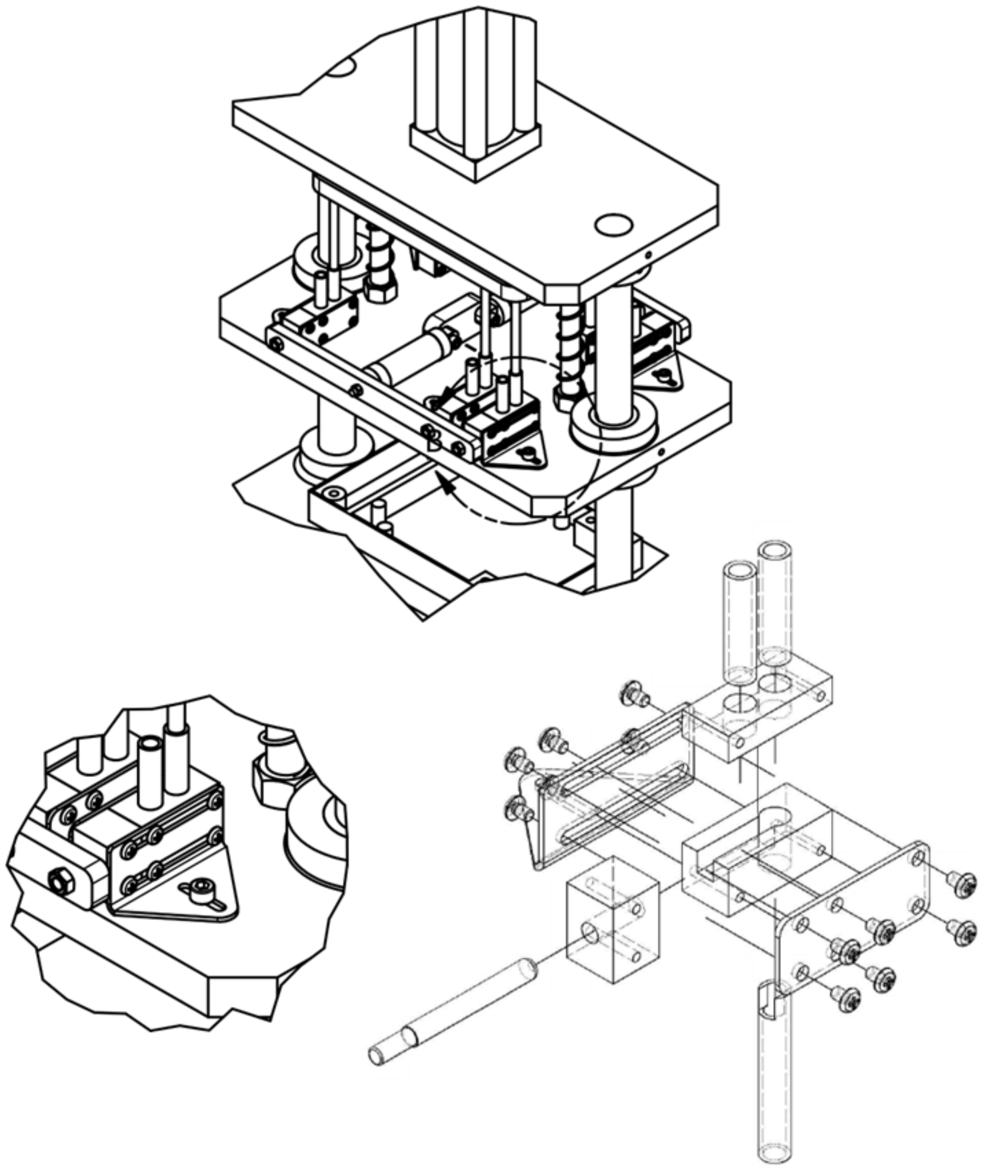

Figure 3:Technical Drawing of the Pneumatic Auto-Feed and Press Mechanism.

The heating process occurs *during* this transport phase. The nut is pushed vertically through a heating assembly, which must bring it to the optimal insertion temperature ($T_{ext\ opt}$) before the pneumatic press engages for insertion.

The heating assembly itself consists of three key parts:

1. **The Heating Collar (Figure 4):** This is the resistive element that generates heat.
2. **The Adapting Cylinder (Figure 5):** This tube is clamped *by* the collar and acts as the primary thermal interface, transferring heat to the nut.

3. **The Insert Nut (Figure 6):** The target component, which passes *through* the adapting cylinder to be heated.

b. Objective of the CAD Preparation Phase

The primary objective of this phase is not simply to model the entire machine, but to perform **geometric abstraction (or "defeaturing")**.

A raw design model (like **Figure 3**) is designed for manufacturing and assembly, containing excessive detail irrelevant to the thermal problem. These details would make the simulation computationally impossible.

We must isolate the "control volume" to balance two needs:

1. **Geometric Fidelity:** Accurately capturing the features that govern heat transfer.
2. **Computational Efficiency:** Removing features irrelevant to the thermal analysis.

c. The Brass Insert Nut

The nut, the core component of the analysis, was modeled with careful consideration of its features:

- **Features Retained:**
  - The **hollow cylindrical body**, as this defines the core thermal mass.
  - The **external grooves/knurling**. This feature was *retained* (unlike in the LCM) because it directly impacts the nature of the contact interface with the heating tube, creating intermittent air gaps and contact points. This is critical for replacing the simplified $\boldsymbol{h}$ coefficient.
- **Features Removed :**

- The **internal screw threads**. These threads are irrelevant to the *external* heating process and would add millions of unnecessary elements to the mesh. The internal bore was modeled as a simple, smooth cylinder.

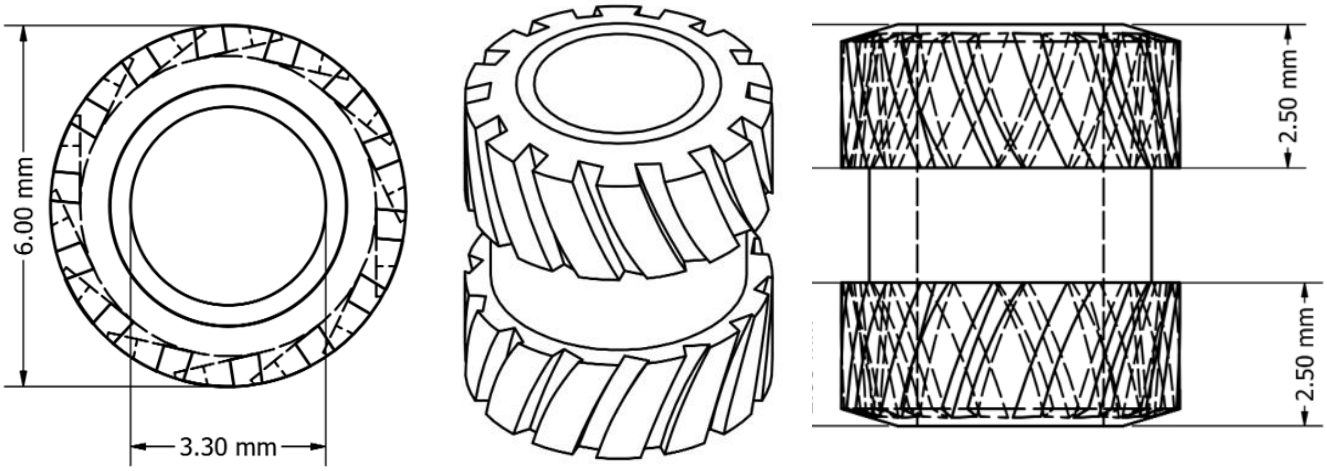

Figure 4: Technical Drawing of the Brass Insert Nut

d. The Heating Assembly (The Source)

The heat source is an assembly, shown in **Figure 5** and **Figure 6**

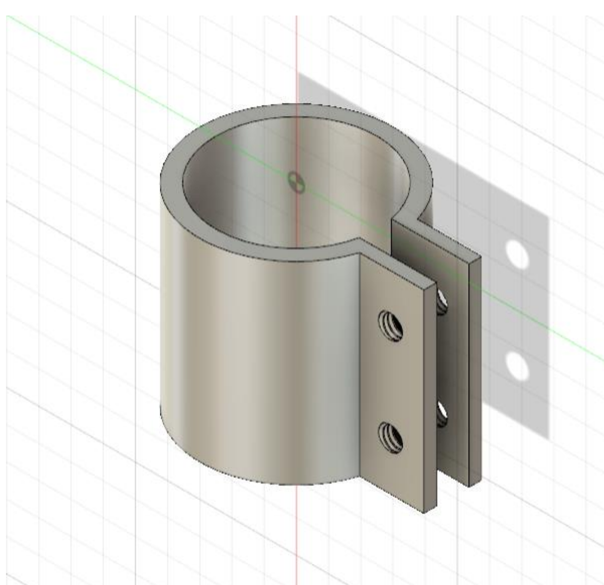

Figure 5: 3D Model of the Resistive Heating Collar

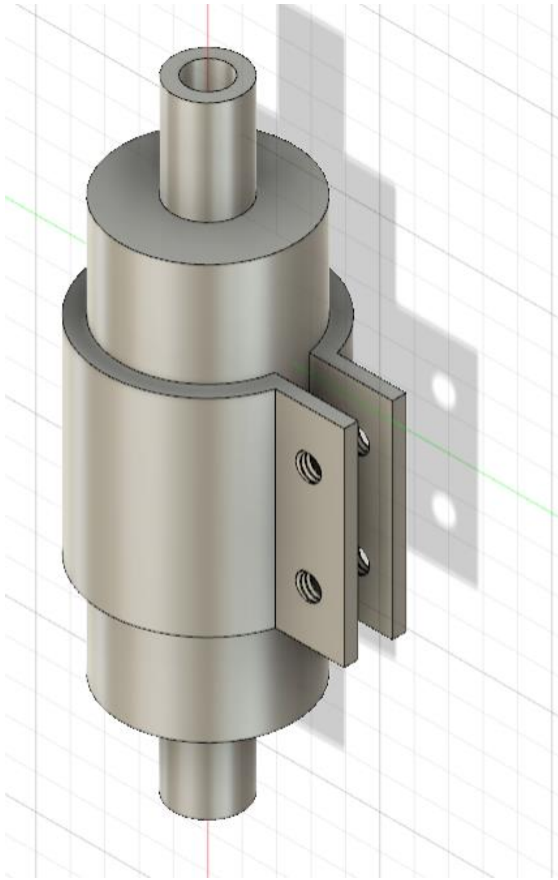

Figure 6: 3D Model of the Adapting Cylinder (Transfer Tube) assembled with the Heating Collar.

- **Heating Collar (Figure 5):** This is the resistive element.

Diametre D (mm)	Height H (mm)	Rated Power (W)	Rated Voltage (V)	Description
25	30	100	230	Heating Collar made from Brass

Table 6: Heating collar technical informations

- **Adapting Cylinder (Figure 6):** This tube is the critical thermal interface. It is clamped *by* the collar and transfers heat *to* the nut, which passes through its center (as conceptualized in **Figure 7**).

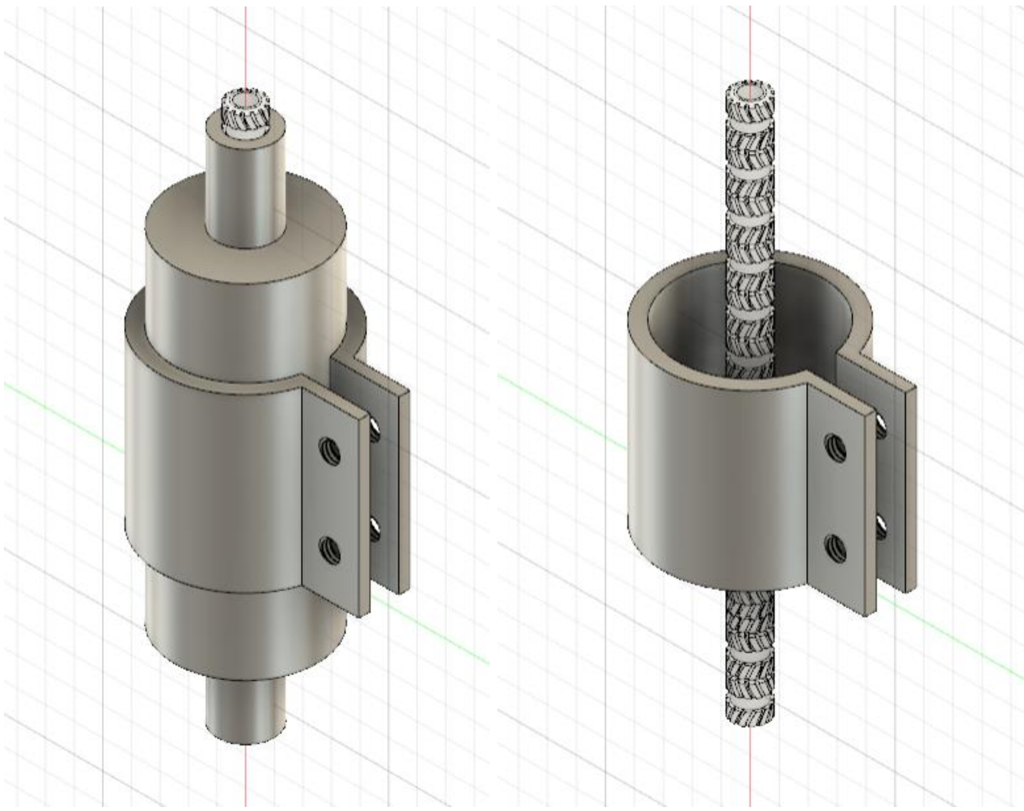

Figure 7: Conceptual Assembly showing multiple Insert Nut stacked inside the Adapting Cylinder.

e. Defining the Simulation-Ready Assembly (Control Volume)

For an efficient simulation, we abstract the function of the **Heating Collar (Figure 3)**. We assume its control system is perfect and that it maintains the **Adapting Cylinder** at a constant, uniform source temperature.

Therefore, the final "simulation-ready" assembly imported into ANSYS consists of only three bodies:

1. **The Insert Nut**
2. **The Adapting Cylinder**
3. **The Heating Collar**

f. The Critical Thermal Interface

The most important physical detail of this assembly is the **interface** between these three bodies.

- **Air Gap Modeling:** We explicitly model the physical **air gap** between the outer peaks of the nut's grooves and the inner wall of the adapting cylinder. This gap is necessary in the real design to allow the nut to slide freely.
- **Replacing the LCM:** This explicit modeling of the **air gap** (which ANSYS will treat as a medium for conduction and radiation) directly **replaces the highly estimated, single-value *h*** used in the LCM. This step eliminates the primary source of uncertainty from our theoretical analysis.

IV. Advanced Exploration: Transient Thermal Analysis (ANSYS)

This chapter details the high-fidelity simulation of the transient heating process. The objective is to move from a 1D analytical *estimation* (LCM) to a 3D numerical *prediction* (FEA) of the nut's temperature evolution ($T(x, y, z, t)$). This approach directly addresses the two critical limitations identified in Chapter 5: the uncertainty of the h coefficient and the neglect of spatial temperature gradients.

A. Methodology and Workflow

The simulation was configured using the **ANSYS Mechanical (Transient Thermal)** environment. This tool was specifically chosen for its robust solver capabilities in handling complex, time-dependent heat transfer, particularly its advanced modeling of contact interfaces and radiation.

a. Material Assignment and Meshing Strategy

The Three-body assembly (Nut, Adapting Cylinder and Heating Collar) prepared in Chapter 4 was imported into ANSYS as a .STEP file.

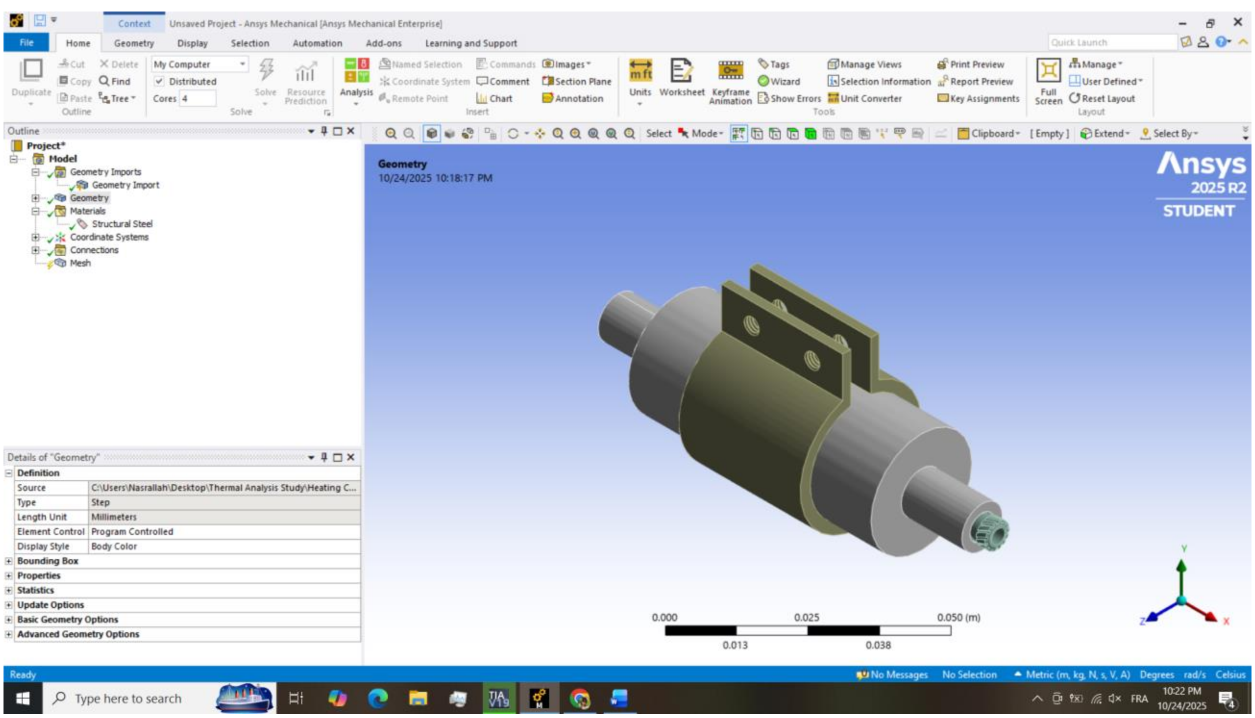

Figure 8: Imported CAD Geometry in the Ansys Mechanical Interface.

b. **Material Assignment:**

A significant refinement over the analytical model (which used generic properties) is the assignment of a specific, traceable engineering alloy. All three physical components (Nut, Adapting Cylinder, and Heating Collar) are specified as **Brass C37700 (Forging Brass)**.

For the simulation, the high-fidelity thermophysical properties for C37700 were applied to the Three bodies (Nut and Adapting Cylinder):

Property	Value (Brass C37700)	Unit
Thermal Conductivity	112.1	$W.m^{-2}.k^{-1}$
Density	8267	kg/m^3
Specific Heat	377.1	$J/kg \cdot K$

Table 7: Thermal properties of Brass C37700 (Forging Brass)

(Note: These precise values replace the generic estimations used in the LCM, thereby increasing the accuracy of the simulation's thermal time constant.)

c. **Implementation of Contact Regions**

This is the most critical step of the simulation. The simplified $h$coefficient from the LCM is replaced by modeling the *physics* of the interfaces. ANSYS automatically detected multiple "Contact Regions" to define the thermal interactions between the bodies. We configured three primary types of contact:

- Heating Collar to Cylinder (Figure 9)
- Cylinder to Insert Nut (Figure 10)
- Insert Nut to Insert Nut (Figure 11)

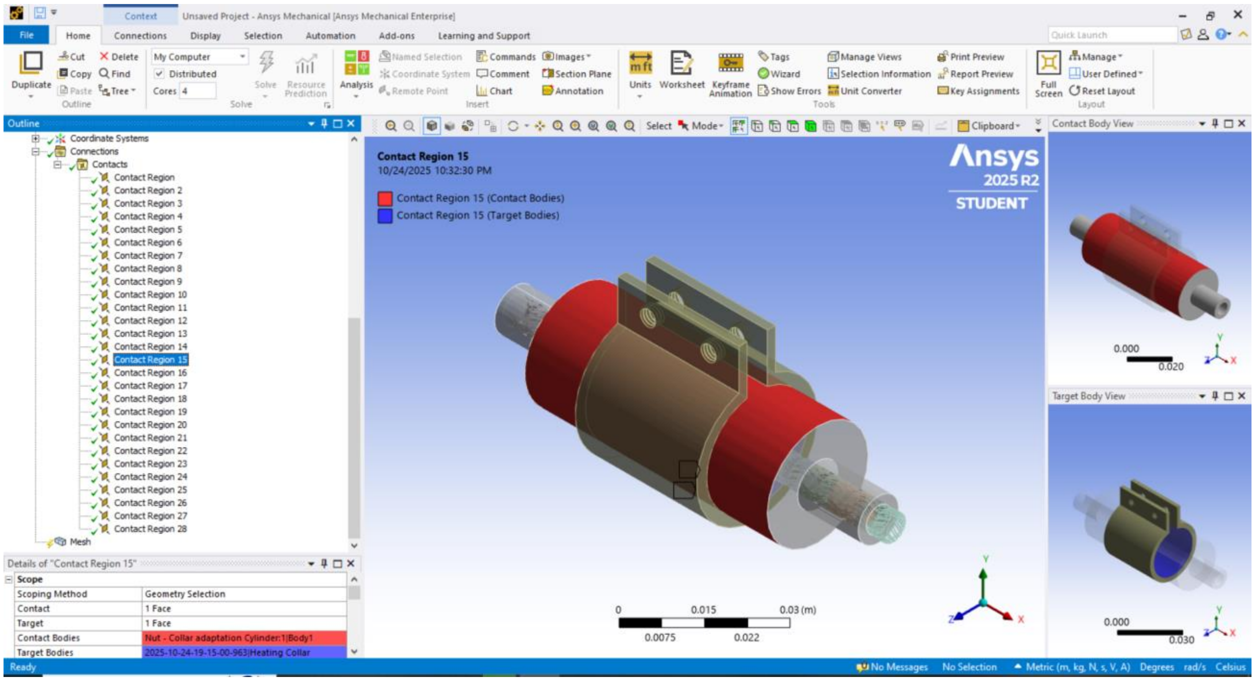

Figure 9: Contact Region Between Heating Collar and Adapting Cylinder

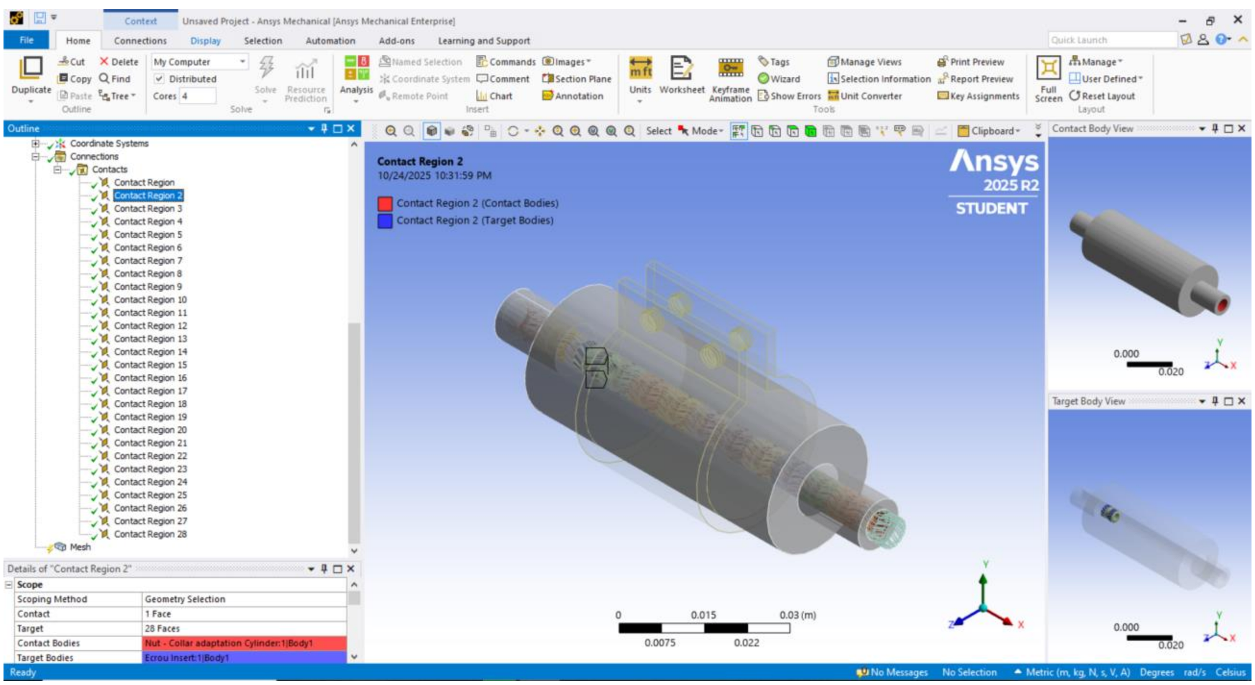

Figure 10: Contact Region Between Insert Nut and Adapting Cylinder

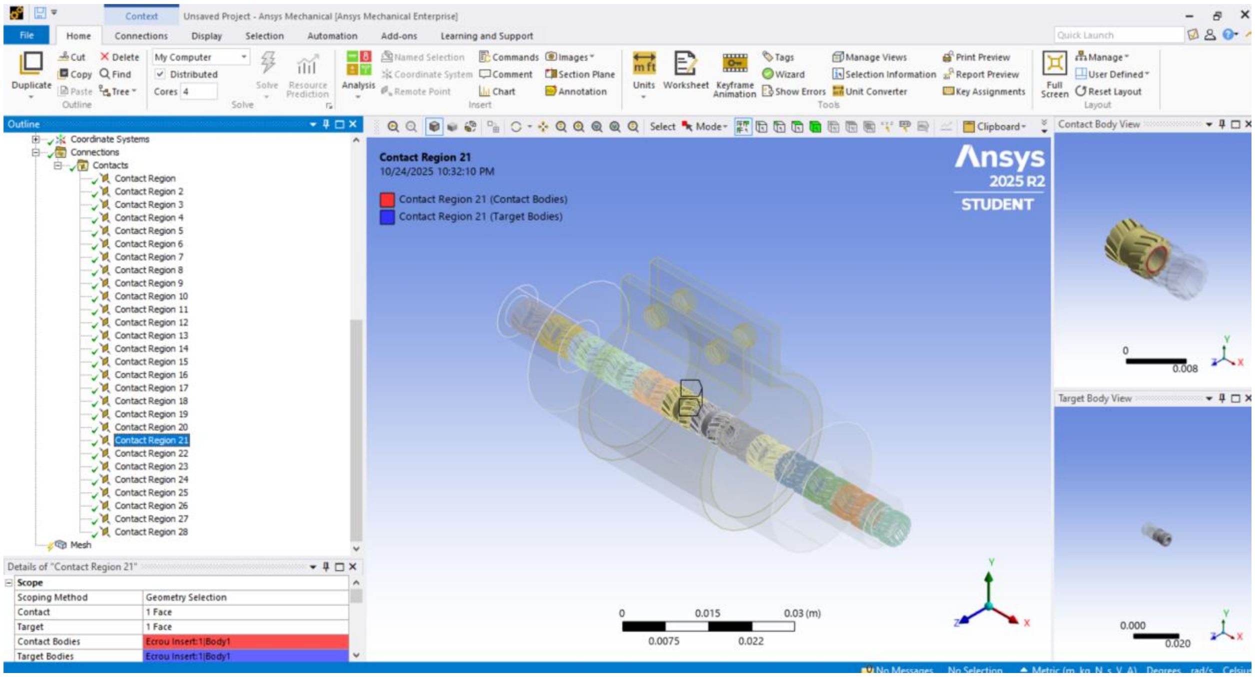

Figure 11: Contact Region Between two Insert Nuts stored in the Adapting Cylinder

d. Transient Setup: Time Stepping

To capture the rapid heating curve:

- **Step End Time**: Set to **30 s**. This duration is sufficient to observe the full transient heating (approx. 2-5 seconds) and confirm that the nut stack reaches a stable thermal equilibrium (steady-state).
- **Auto Time Stepping**: Set to **On**. This is the core of the strategy. It allows ANSYS to use very small time steps when the temperature changes rapidly (at $t = 0$) and larger steps as the system stabilizes, ensuring both accuracy and computational efficiency.
- **Initial Time Step**: Set to **0.01 s**. This forces the solver to begin with a high-resolution step to accurately capture the initial, most violent phase of the heat transfer.
- **Minimum Time Step**: Set to **0.001 s** (1 millisecond). This provides a lower bound, preventing the solver from failing if extreme thermal gradients cause convergence issues at the contact interfaces.
- **Maximum Time Step**: Set to **0.5 s**. This prevents the solver from taking excessively large steps once the curve begins to flatten, ensuring that no subtle thermal dynamics are missed.

e. Solver Controls

The solver is the mathematical engine that solves the system of heat transfer equations $\left([C]\{\dot{T}\} + [K]\{T\} = \{Q\}\right)$ for every node at every time step. The choice of solver type dictates the trade-off between computational speed, memory (RAM) usage, and numerical robustness.

ANSYS provides three options: "Program Controlled," "Iterative," and "Direct."

1. **Iterative Solver:** This solver (e.g., PCG - Preconditioned Conjugate Gradient) is an **approximation** method. It starts with a guess and "iterates" (refines) the solution at each step until the error is below a defined tolerance.
   - **Pros:** Very low memory (RAM) usage. Can be much faster for *extremely large* models (e.g., > 1 million degrees of freedom).
   - **Cons:** Less robust. It can fail to converge (find a solution) if the model is complex or "ill-conditioned" (e.g., has a poor-quality mesh or extreme differences in material properties).
2. **Direct Solver:** This solver (e.g., Gaussian elimination) is an **exact** method. It directly computes the solution to the matrix equations at each time step, within machine precision.
   - **Pros:** Extremely **robust** and **accurate**. It is the most reliable method for complex problems, especially those involving multiple contact regions (like our air gaps) and radiation.
   - **Cons:** Very high memory (RAM) and CPU cost, which scales rapidly with model size.

Best Option and Justification

For this analysis, "Program Controlled" (which often defaults to "Iterative" for thermal analysis) is not acceptable due to the uncertainty.

The **Direct** solver was explicitly selected.

Justification: The primary goal of this entire chapter is to obtain a **high-fidelity, numerically accurate** prediction to **validate or reject** the 1D analytical LCM model.

- The model geometry, while not massive, is complex due to the contact interfaces (air gaps) and the fine inflation-layer mesh, which can create ill-conditioned matrices.
- The "Direct" solver guarantees a **robust and non-approximated solution** at each time step.
- Using an "Iterative" solver would introduce a *second* source of approximation (the convergence tolerance) on top of the *first* source of approximation (the LCM), making a clear comparison impossible.

Therefore, the higher computational cost of the **Direct** solver is a necessary and acceptable trade-off for the guaranteed accuracy required by this academic investigation.

f. Radiosity Controls (Radiation Solver)

This control panel, shown in **Figure 12**, is fundamentally important as it governs the accuracy of the **surface-to-surface radiation** calculation—a key physical mechanism that was ignored in the LCM and is being explicitly modeled here.

The "Radiosity" method is the numerical engine ANSYS uses to solve for the radiative heat flux between the surfaces of the air gap (the hot inner wall of the cylinder and the cooler outer surface of the nut).

Radiosity Controls	
Radiosity ...	Program Controlled
Flux Conv...	1.e-004
Maximum ...	1000.
Solver Tol...	0.1 W/m²
Over Rela...	0.1
Hemicube...	10.

Figure 12: Radiosity solver controls for the transient analysis.

Justification of Solver Settings

We will **accept the default numerical tolerances** because they are already set for a high-precision, engineering-grade solution:

- **Radiosity ...**: This is set to **"Program Controlled,"** which automatically activates the Hemicube (view factor) calculation method required for the surface-to-surface radiation boundary condition

- **Flux Conv. (Flux Convergence):** The default value of **1.e-004** (or 0.01%) is a **relative convergence tolerance**. This means the radiation solver will continue to iterate until the change in heat flux between iterations is less than 0.01%. This is an extremely strict tolerance and is ideal for our goal of achieving a high-fidelity, accurate result.

- **Maximum ... (Maximum Iterations):** The default of **1000** is a high limit that provides a sufficient number of attempts for the solver to reach the 1.e-004 tolerance, ensuring a converged solution.

- **Solver Tol.**: The default of **0.1 W/m²** is an *absolute* tolerance that acts as a secondary convergence check.

Conclusion: Accepting these default settings is a deliberate choice. They ensure that the radiation component of the heat transfer—a critical element missing from our initial LCM approximation—is calculated with maximum accuracy, which is essential for the final validation.

g. Nonlinear Controls

The "Nonlinear Controls" panel, shown in **Figure 13**, is critical for this analysis. Although the material properties themselves are linear, the simulation is fundamentally **non-linear**.

Nonlinear Controls	
Heat Convergence	Program Controlled
Temperature Convergence	Program Controlled
Line Search	Program Controlled
Nonlinear Formulation	Program Controlled

Figure 13: Nonlinear solver controls for the transient analysis.

Source of Non-Linearity: The **Surface-to-Surface Radiation** model calculates heat flux as a function of the fourth power of absolute temperature ($q\infty T^4$). This T^4 relationship makes the governing heat transfer equations non-linear.

Justification of Solver Settings

The default value of **"Program Controlled"** was accepted for all non-linear controls.

- **Heat Convergence / Temperature Convergence:** This setting allows ANSYS to automatically determine the best criteria (e.g., change in heat flux or change in temperature) to check if the non-linear solution has converged at each time step.
- **Rationale:** The "Program Controlled" setting is the most robust choice. It instructs the **Direct Solver** to automatically handle the complex, iterative calculations required to solve the T^4 radiation problem at each time step.

Manually overriding these settings is an advanced procedure, typically reserved for cases where a simulation fails to converge. For this analysis, the default settings provide the necessary robustness to ensure the non-linear radiation effects are solved accurately.

h. Advanced settings: DMP Controls

The final solver setting, shown in **Figure 14**, relates to how ANSYS manages the computational workload in a multi-core environment (parallel processing).

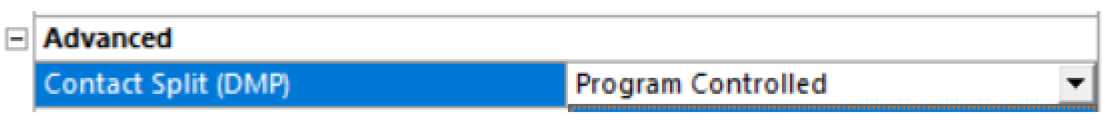

Figure 14: Advanced Distributed Memory Parallel (DMP) controls.

Analysis:

- **Contact Split (DMP)**: DMP stands for **Distributed Memory Parallel**. This setting controls how the solver divides the complex contact calculations (like our air gaps and nut-to-nut interfaces) among the available computer processor cores.
- **Justification:** The default value of **"Program Controlled"** was accepted. This instructs ANSYS to automatically optimize the workload distribution. Manually overriding this setting is an advanced troubleshooting step for very large-scale simulations (millions of elements) that are failing due to memory imbalances on a specific processing core.

For this analysis, the default setting provides the most stable and efficient computational performance.

i. Output Controls

The **"Output Controls"** (shown in **Figure 15**) are not solver settings, but rather a **results request** filter. They instruct the solver on which specific data sets to calculate and save to the results file (.rst). This is a critical step, as data that is not requested is not calculated and cannot be viewed later.

Output Controls	
Output Selection	None
Calculate Thermal Flux	Yes
Contact Data	Yes
Nodal Forces	No
Volume and Energy	Yes
Euler Angles	Yes
General Miscellaneous	No
Contact Miscellaneous	No
Store Results At	All Time Points
Result File Compression	Program Controlled

Figure 15: Output Controls for requesting specific solution data.

Justification of Key Result Requests

The default settings are insufficient for our analysis. The following controls were deliberately set to "Yes" to ensure the data required would be generated:

- **Calculate Thermal Flux**: Set to **Yes**.
  - **Rationale:** This is essential. We need this result to visualize the flow of heat (q) from the cylinder, across the air gap, and into the nut. It allows us to identify the primary heat paths and any unexpected thermal bottlenecks.
- **Contact Data**: Set to **Yes**.
  - **Rationale:** This works with Calculate Thermal Flux to allow us to probe the specific heat transfer *at the contact interfaces*. This is the only way to quantify the heat transfer breakdown (e.g., "how much heat is from Conduction vs. how much from Radiation") across the critical air gap, which is the core of this investigation.
- **Volume and Energy**: Set to **Yes**.

- **Rationale:** This provides data to confirm the energy balance of the system (i.e., $E_{\text{in}} = E_{\text{stored}} + E_{\text{lost}}$), which serves as a critical check on the simulation's overall validity.

- **Store Results At**: Set to **All Time Points**.
  - **Rationale:** This is the **most important setting for a transient analysis**. The default is often "Last Time Point," which would only show the final temperature at $t = 30\,\text{s}$. By selecting **"All Time Points,"** we command ANSYS to save the complete temperature and flux solution for *every single time step* calculated. This is the raw data that will be used to generate the new T vs. t heating curve (the t_{ANSYS}) to compare against the LCM graph.

The other settings (e.g., Nodal Forces, Euler Angles) were left at their default values ("No" or "Yes") as they are irrelevant to a purely thermal analysis.

j. Meshing Strategy: Modifying the Default Settings

A high-quality mesh is non-negotiable for an accurate FEA. The default settings (shown in **Figure 16**) are optimized for fast, simple mechanical problems and are unsuitable for our high-fidelity thermal analysis.

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	0.10621 m
Average Surface Area	1.097e-005 m²
Minimum Edge Length	5.8386e-005 m
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Target Element Quality	Default (5.e-002)
Smoothing	Medium
Mesh Metric	None
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
Inflation Element Type	Wedges
View Advanced Options	No
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled

Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Straight Sided Elements	No
Rigid Body Behavior	Dimensionally Reduced
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Auto-Map Fillets	No
Automatic Methods	
Sheet Body Method	Prime Quad Dominant
Sweepable Body Method	Sweep
Statistics	
Nodes	94414
Elements	48644
Show Detailed Statistics	No

Figure 16: Default ANSYS Meshing controls

The default settings would produce a coarse mesh of only **~48,000 elements**, which is too low to accurately resolve the critical thermal gradients. The following **critical modifications** were implemented to ensure simulation fidelity:

1. **Element Order**: Default is "Program Controlled."
    - **Action:** Changed to **Quadratic (2nd Order)**.

- **Justification:** This is a fundamental change. Linear elements (default) can only calculate a linear temperature gradient *within* each element. Quadratic elements have mid-side nodes that can accurately model the non-linear, curved temperature profile that *actually occurs* within the solid. For precision, this is mandatory.

2. **Sizing -> Resolution**: Default is 2.
    - **Action:** Increased to a higher value **6**.
    - **Justification:** The default resolution is too coarse and will not capture the fine details of the nut's grooves. A higher resolution creates a finer base mesh for the entire model.
3. **Sizing -> Span Angle Center**: Default is "Coarse."
    - **Action:** Changed to **Fine**.
    - **Justification:** This setting controls how accurately curved surfaces are meshed. "Coarse" creates large, flat elements on cylindrical faces, which introduces geometric error. "Fine" ensures all curved faces (of the nut, cylinder, and grooves) are accurately represented.
4. **Sizing -> Mesh Defeaturing**: Default is "Yes."
    - **Action:** Changed to **No**.
    - **Justification:** This default is dangerous. It is designed to automatically remove "small" features. In our case, it would likely identify the **nut's grooves** as "small" and remove them. Since these grooves define the critical air gap, they *must* be preserved.

By rejecting the default settings and implementing these five specific modifications (Quadratic elements, fine resolution and no defeaturing), we ensure a high-fidelity mesh. The resulting element count will be significantly higher than the 48k default, but this is a necessary cost for obtaining a numerically accurate and reliable solution.

The implementation of these modifications to the meshing strategy is complete. The resulting high-fidelity mesh is shown in **Figure 17**.

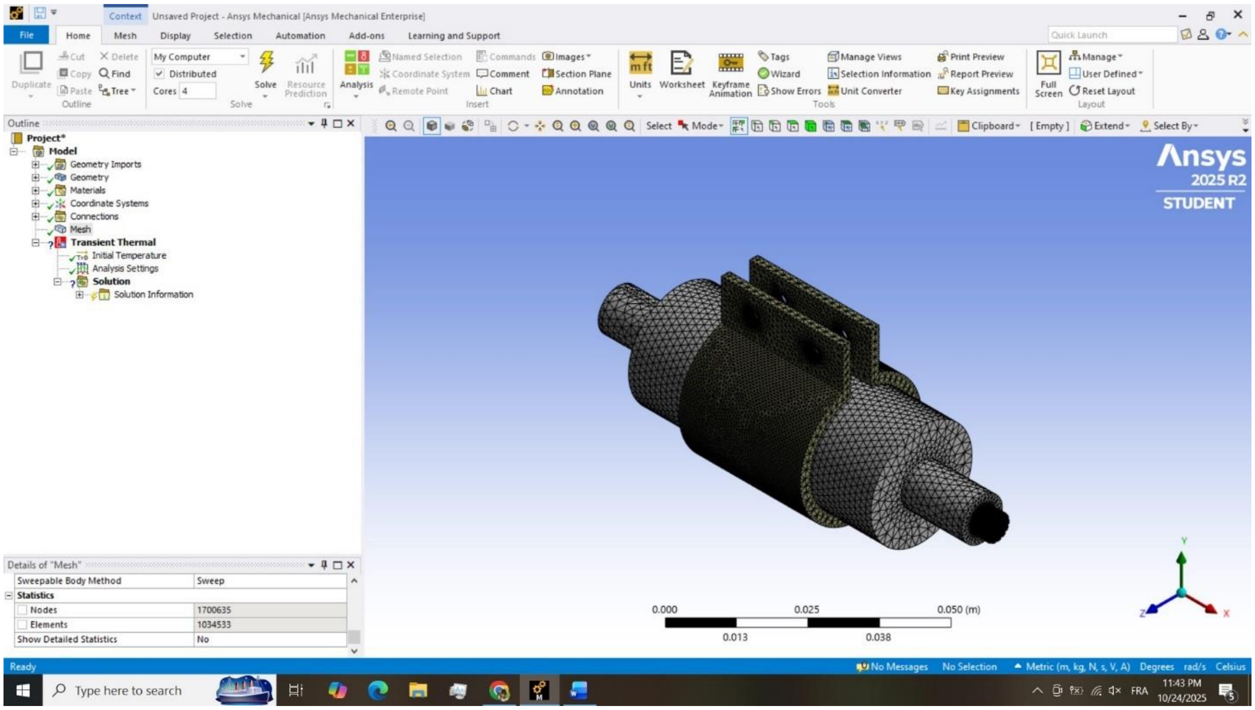

Figure 17: The final, high-fidelity mesh generated in ANSYS, with statistics shown in the "Details of Mesh" pane.

The mesh statistics reveal the direct impact of our refined settings. The final model consists of **1,034,633 elements** and **1,700,035 nodes**. This represents a more than **20-fold increase** in density compared to the default ~48k elements.

This significant increase in element count is not a cost; it is the *objective*. It is the necessary result of applying **Quadratic elements**. This high-density, "simulation-ready" mesh is now capable of accurately resolving the steep thermal gradients and non-linear radiation effects, ensuring the validity of the final transient solution.

B. Implementation of Boundary Conditions

With the mesh and solver physics defined, the final step is to apply the thermal loads that drive the simulation. These loads define the initial state of the system ($t = 0$) and the external heat sources and sinks ($t > 0$).

All necessary tools for this are found in the "Transient Thermal" context menu, as shown in **Figure 18**

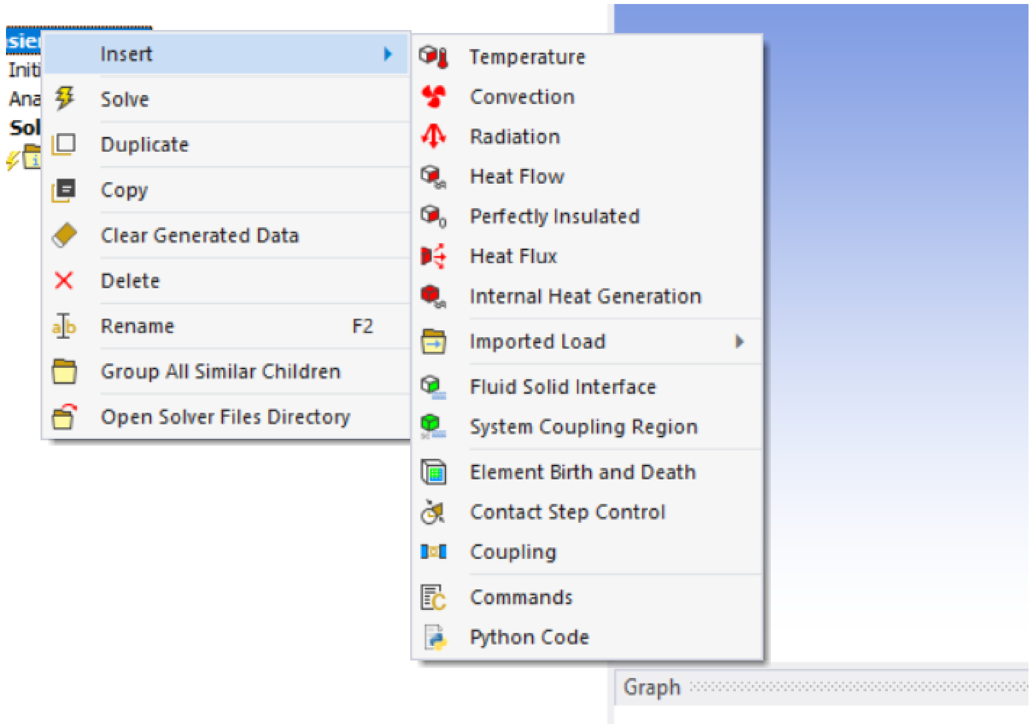

Figure 18: The ANSYS "Insert" menu showing the available thermal boundary conditions.

Load Application (for $t > 0$)

Two primary boundary conditions were applied to simulate the operational environment of the press:

1. The Heat Source (Load)

The heat is generated by the resistive heating collar. This is simulated by applying a fixed temperature to that body, which then conducts heat into the rest of the assembly via the contact regions.

- **Tool Used:** Temperature
- **Scoping:** Applied to the **Heating Collar** body.
- **Definition:** Set to a constant value of $T_{Inifinity} = 180C$.
- **Rationale:** This acts as the primary thermal load. Heat flows from the collar, across "Contact Region 15" into the Adapting Cylinder, and finally across the critical air gap ("Contact Region 2") into the nut.

2. Heat Loss to Ambient Air (Sink)

This boundary condition simulates how the assembly loses heat to the surrounding workshop environment.

- **Tool Used:** Convection
- **Scoping** (Surfaces Selected):

  This condition is applied only to faces that are "open" and exposed to the ambient workshop air. This includes:

  - The outer faces of the **Heating Collar**.
  - The **top face** of the top-most nut in the stack.
  - The **bottom face** of the bottom-most nut in the stack.
  - The **internal bores** (the 3.30 mm holes) of the *entire* nut stack.
- **Scoping (Surfaces *Not* Selected):**
  - Crucially, this Convection load is **NOT** applied to the outer grooved surfaces of the nut or the inner wall of the adapting cylinder. Those faces are part of the **Contact Region** (the trapped air gap) and are handled separately.
- **Definition:**
  - **Film Coefficient:** $h_{cooling} = 15W/m^2K$ (the same value used in the LCM cooling analysis).
  - **Ambient Temperature:** $T_{ambient} = 25C$.

*Note: The **Radiation** load, also seen in the menu, was not applied globally. It was modeled more precisely as a **surface-to-surface** phenomenon embedded within the physics of the "Contact Region" (the air gap).*

C. Challenges Encountered: Computational Limits

After completing the high-fidelity simulation setup (including the advanced meshing, boundary conditions, and solver controls), the "Solve" command was executed.

However, the simulation did not complete. Instead, it terminated immediately with a critical error, as shown in **Figure 7.D**.

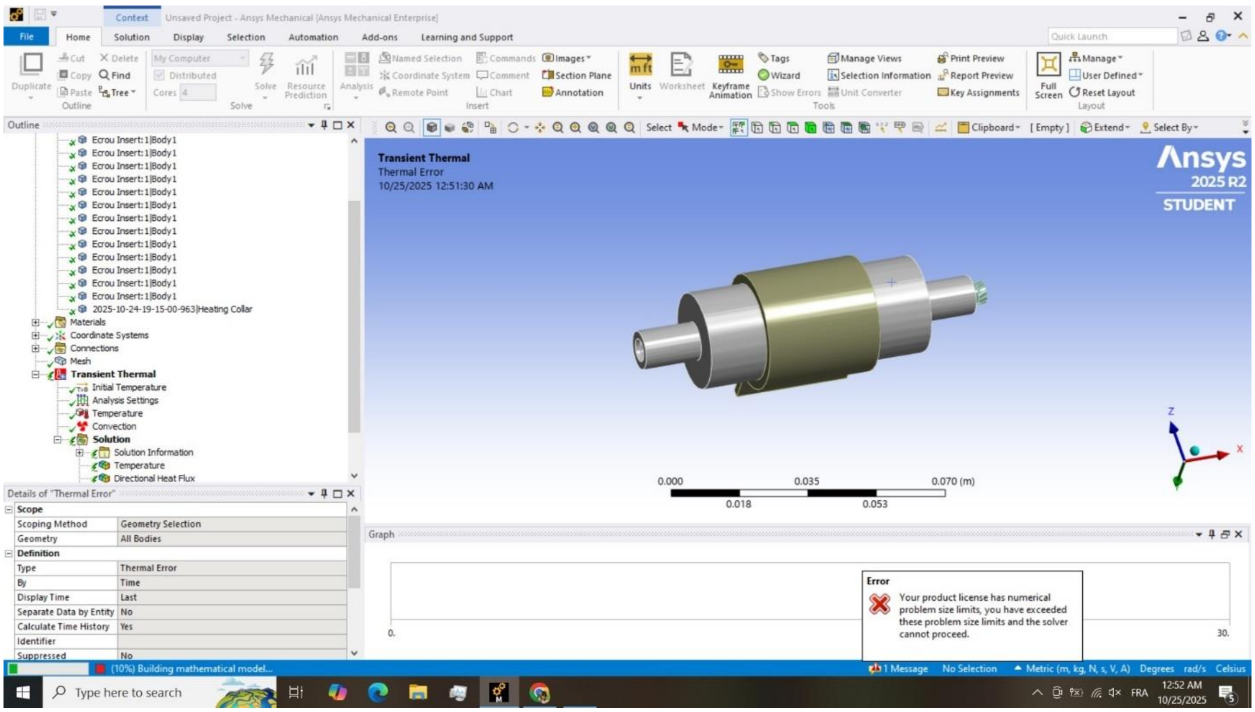

Figure 19: A critical solver error reporting that "numerical problem size limits" have been exceeded.

a. Analysis of the Challenge

The error message is not a *physics* or *setup* error; it is a **computational resource constraint**.

The root cause is the **ANSYS Student 2025 R2** license being used for this academic exploration. This free license imposes a hard limit of **128,000 nodes/elements** for structural and thermal analyses.

Our pursuit of maximum numerical accuracy—by implementing:

1. A **full 12-nut stack** geometry,
2. **Quadratic (2nd Order) elements**,

3. **High-resolution meshing** (Resolution 6, Fine Span Angle)

These settings resulted in a high-fidelity model with **1,700,035 nodes and 1,034,533 elements.** This professional-grade mesh, while ideal for accuracy, **far exceeds the node limit** of the student license.

b. The Engineering Trade-Off

This presents a classic engineering challenge: a direct conflict between the desire for **maximum accuracy** (academic rigor) and the **practical constraints** of the available software license.

The simulation, as configured, is "correct" but "unsolvable."

c. The Path Forward: Model Reduction Strategy

To proceed with the analysis, the simulation's complexity must be drastically reduced to fit within the 128,000-node limit. This is a standard engineering practice known as **model reduction**.

A **model reduction** was mandatory. However, removing Insert Nuts (geometric reduction) or using Linear elements (physics reduction) would compromise the simulation's core objectives.

A more nuanced, iterative approach was adopted instead, focusing *only* on the Sizing controls:

- **Initial Resolution:** 6 (This was the primary driver of the excessive element count).
- **Updated Resolution:** The Resolution setting was iteratively reduced from 6 down to **2**.

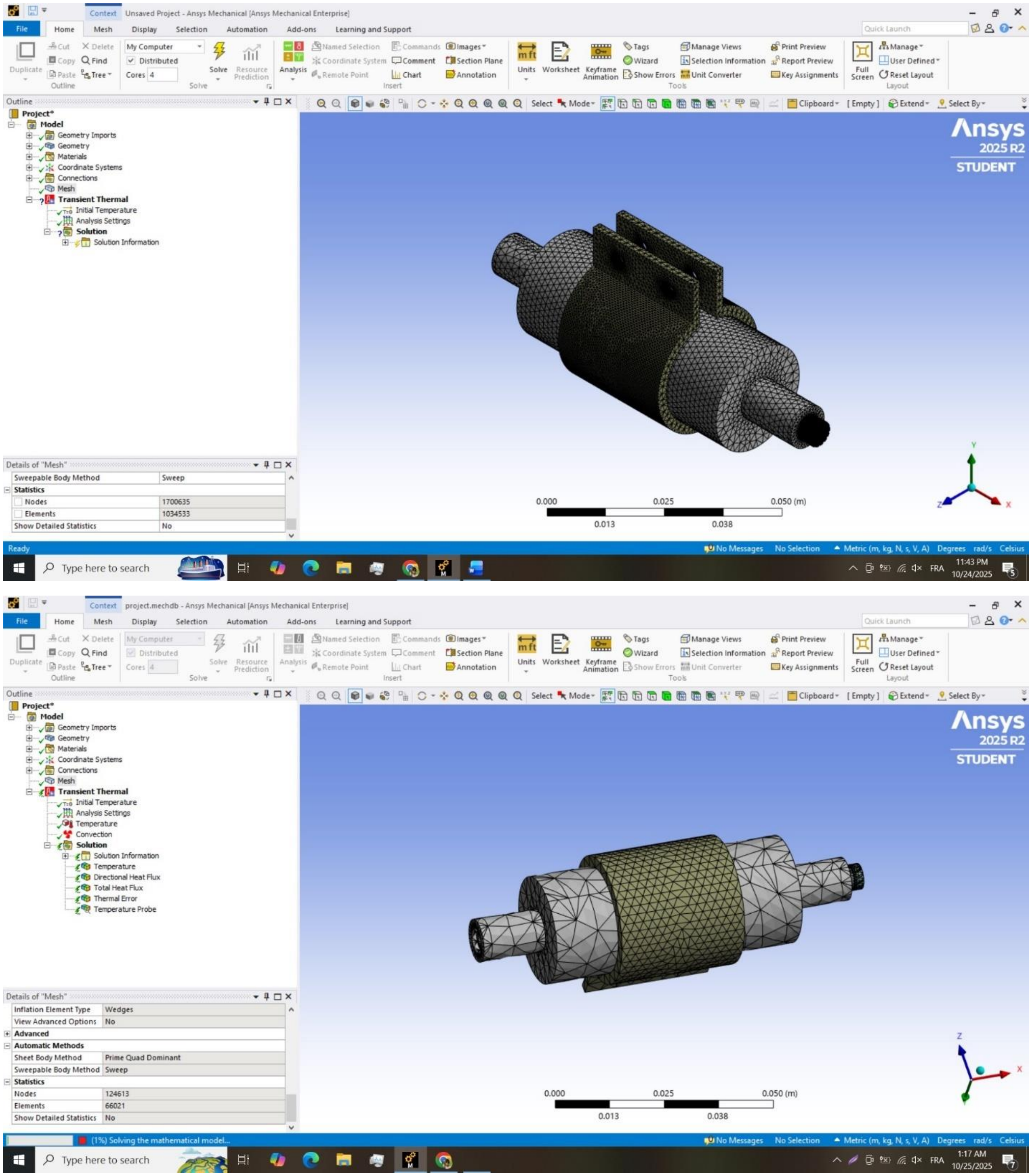

Figure 20: Mesh Refinement Comparison: (a) Initial High-Fidelity Mesh (1,700,035 Nodes) vs. (b) Final Solvable Mesh (124,613 Nodes).

The final, solvable mesh statistics are:

- **Nodes: 124,613**
- **Elements: 66,021**

Conclusion: The model is now fully configured, computationally solvable, and retains the high-fidelity physics required to validate the LCM. We can now proceed to solve the analysis and review the results.

D. Transient Results and Analysis

The modified simulation, with its solvable 124,613-node mesh, was successfully run for the full **30-second** duration. The "Solution" branch is now populated with the complete time-history data.

To analyze these results, we use the post-processing tools to extract the key findings. The analysis is threefold:

1. Verify the spatial temperature gradients (to disprove the LCM assumption).
2. Validate the heat flow paths (to confirm the model physics).
3. Extract the numerical heating curve (t_{ANSYS}) for comparison.

a. Visualization of Spatial Gradients (Steady-State)

The first objective was to disprove the LCM's assumption of a uniform nut temperature. This was achieved by inserting a **Thermal > Temperature** result and setting the Time to **30 s** to observe the final steady-state condition.

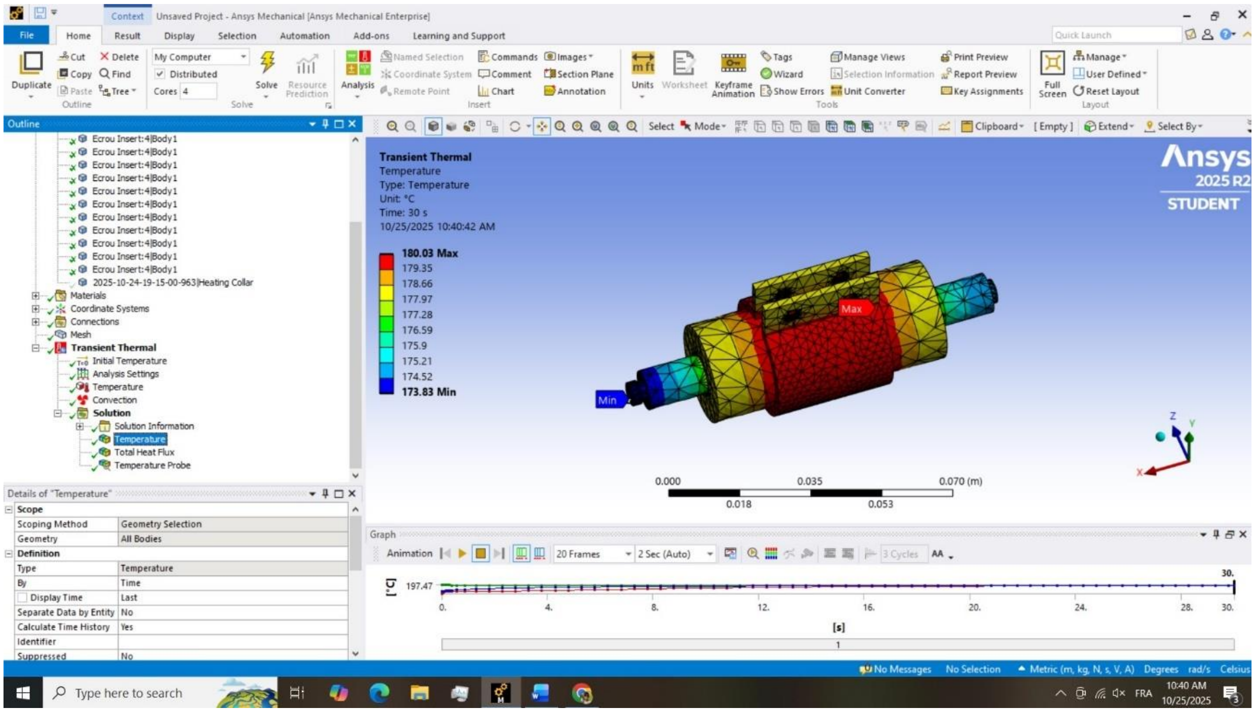

Figure 21: Final Temperature Distribution (t=30s) of the Full Assembly

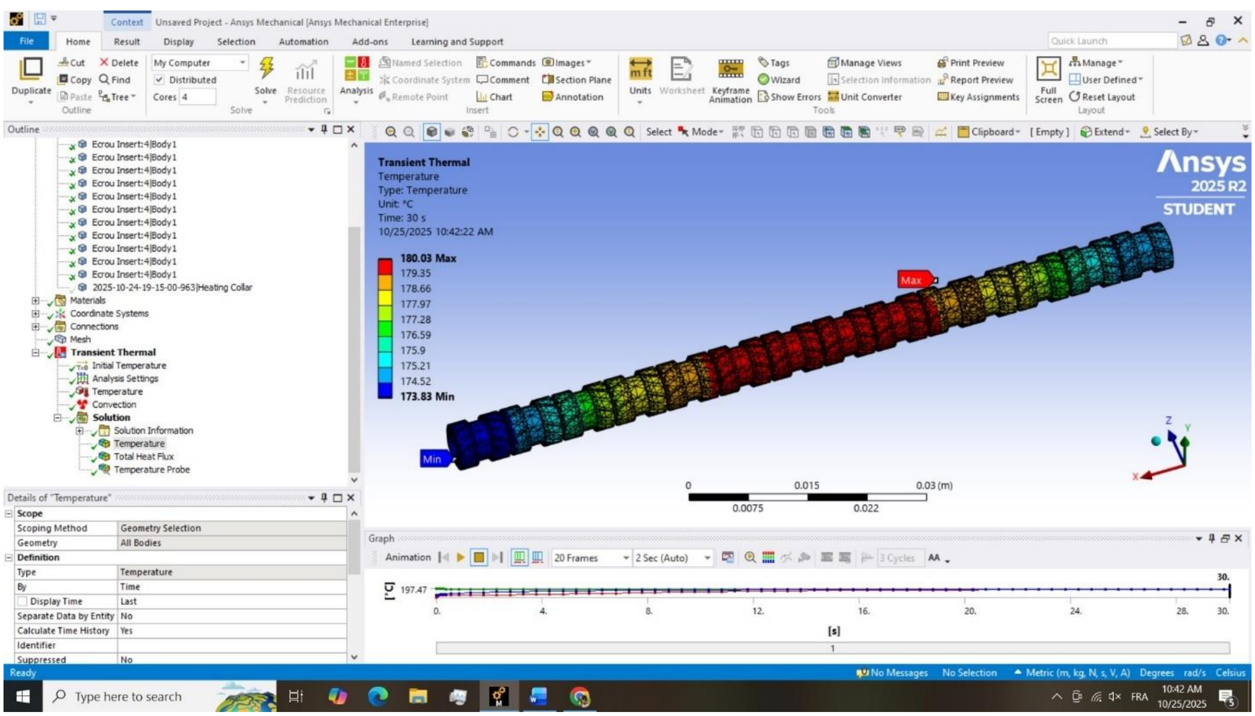

Figure 22: Final Temperature Distribution (t=30s) of the Nut Stack (Isolated View).

Key Findings (LCM Disproven):

1. **System-Level Gradient:** As shown in **Figure 21**, the assembly is not isothermal. The heat source (Heating Collar) reaches a maximum of $\mathbf{180.03^{\circ}C}$, while the end of the nut stack, which is exposed to $\mathbf{25^{\circ}C}$ convection, drops to a minimum of $\mathbf{173.83^{\circ}C}$. This is a $\mathbf{6.2^{\circ}C}$ **gradient** across the assembly.

2. **Nut-to-Nut Gradient: Figure 22,** clearly shows this gradient along the stack. The central nuts (red) are hotter than the end nuts (green/blue), which lose heat to the ambient air.

3. **Intra-Nut Gradient:** The visualizations confirm that even *within* a single nut, there is a clear spatial gradient from the hotter outer surface to the cooler inner bore.

This is the definitive proof that the Lumped Capacitance Model (LCM), which assumed a single temperature for a single nut, was a significant simplification.

b. Validation of Heat Flow Path (Heat Flux)

The second objective was to validate the physics of the model by visualizing the heat paths. A **Thermal > Total Heat Flux** result was inserted, scoped to all bodies.

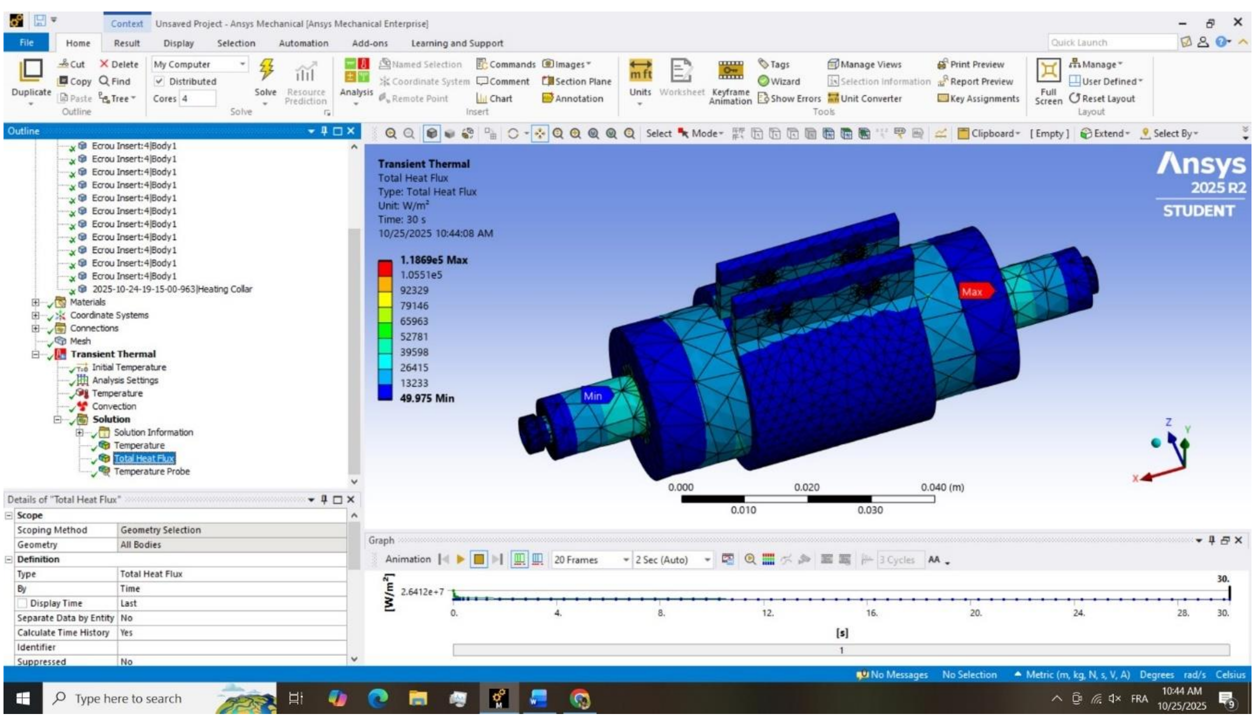

Figure 23: Total Heat Flux of the Full Assembly (t=30s).

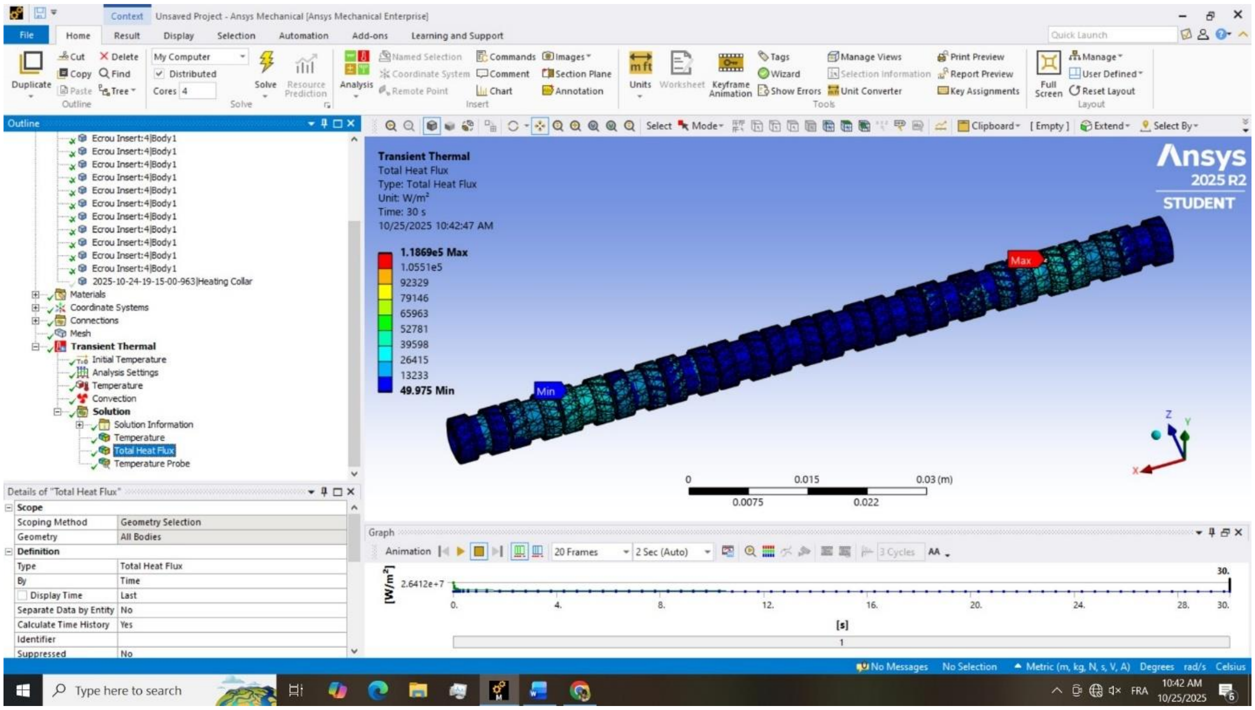

Figure 24: Total Heat Flux of the Nut Stack (Isolated View), highlighting nut-to-nut conduction.

Key Findings (Model Validated):

1. **Primary Heat Path: Figure 23** shows the highest heat flux at the interfaces between the heating collar and the adapting cylinder, confirming this is the primary path for heat entering the system.

2. **Secondary Heat Path: Figure 24** provides a critical insight that was impossible to model with the LCM. The heat flux vectors are clearly visible *between* the faces of adjacent nuts. This confirms that **nut-to-nut conduction** is a significant secondary heat path, pre-heating nuts before they even reach the center of the thermal zone. This validates the decision to model the full stack.

c. Determination of the Numerical Cycle Time

This is the most critical quantitative result of the entire simulation. The goal is to extract the new, high-fidelity heating curve.

- **Action:** A **Probe > Temperature** was inserted. As shown in **Figure 25**, this probe was scoped to the *average temperature* of a **central nut** in the stack. This nut is the most representative, as it is shielded from the heat-loss "end effects."

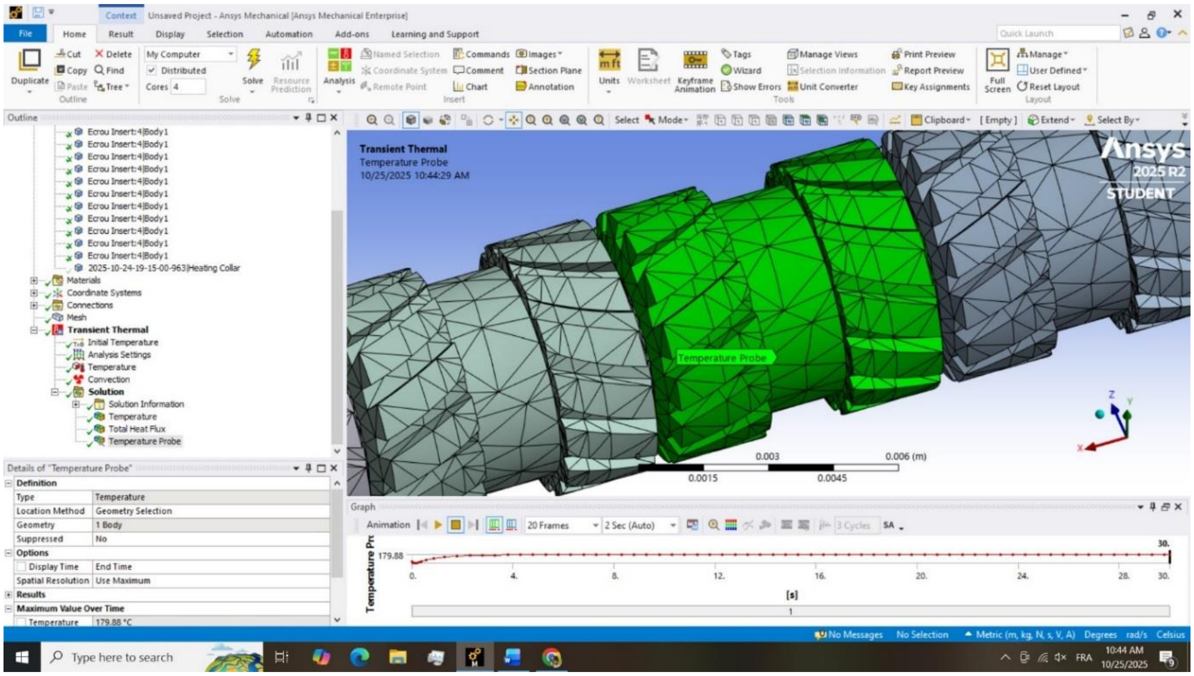

Figure 25: Temperature Probe, scoped to a representative central nut.

- **Result:** ANSYS automatically generated the graph in **Figure 26**, plotting the average temperature of this nut.

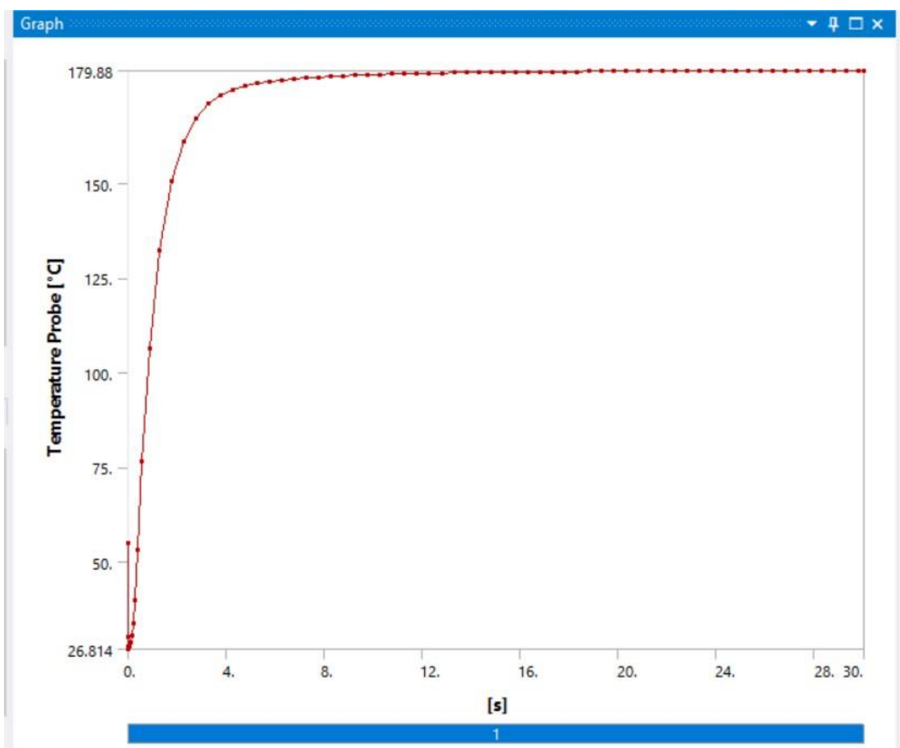

Figure 26: The ANSYS-generated Heating Curve for the central nut.

Key Findings:

This graph is the **numerically validated heating curve**. From this data, we can now extract the precise, high-fidelity time required to reach our operational thresholds:

- **Time to reach $150^{\circ}C$** : $t \approx 2.8$ s
- **Time to reach $170^{\circ C}$** (*optimal insertion temp*): $t \approx 4.0$ s
- **Time to Stabilization:** The nut reaches its final equilibrium temperature of ≈ 179.88 C after approximately **12-14 seconds**.

V. Comparative Analysis

This final chapter synthesizes the entire research exploration, from the initial theoretical approximation (LCM) to the high-fidelity numerical validation (ANSYS).

The results from the Lumped Capacitance Model (LCM) are now directly confronted with the validated data extracted from the full-stack ANSYS simulation.

This comparison will:

1. Quantify the error of the initial analytical model.
2. Explain the physical sources of that discrepancy.
3. Establish the final, validated engineering parameters for the automated press.

A. Comparison of Theoretical Results and Numerical Simulations Using MATLAB

The most direct method for validating the two models is to superimpose their respective heating curves. The theoretical data (t_{LCM}) from the MATLAB and the numerical data (t_{ANSYS}) from the central nut's temperature probe were exported and plotted on a single graph.

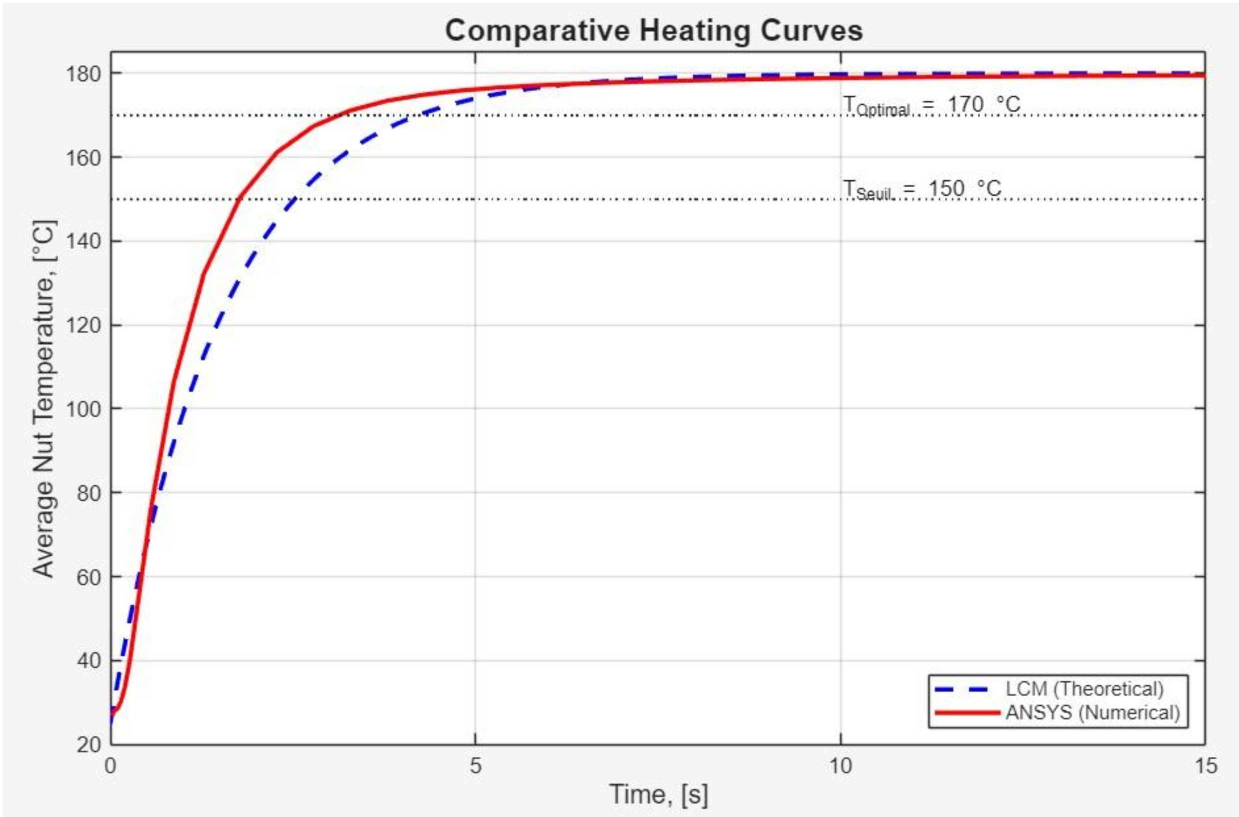

Figure 27: Comparative Heating Curves [Theoretical (LCM, dashed blue) vs. Numerical (ANSYS, solid red)]

The visual comparison in **Figure 27** is striking. It immediately reveals a discrepancy between the two models:

The **ANSYS (Numerical) curve rises substantially faster** than the **LCM (Theoretical) curve**. This indicates that the real-world heating process is far more efficient than the initial analytical model predicted, with the ANSYS model crossing both the 150°C and 170°C thresholds significantly earlier.

B. Quantitative Error Analysis

To quantify the discrepancy shown in the graph, the precise times required to reach the critical operational thresholds are extracted from the raw data (provided in the previous section) and presented in **Table 8.**

Temp. Target	LCM (Theoretical) [s]	ANSYS (Numerical) [s]	Discrepancy (Error of LCM)
$\mathbf{150^{\circ C}}$	2.30	$\mathbf{1.78}$	$\mathbf{+29.2}$\% (LCM overestimated time)
$\mathbf{170^{\circ C}}$	4.22	$\mathbf{3.28}$	$\mathbf{+28.7}$\% (LCM overestimated time)

Table 8: Quantitative Comparison of Theoretical vs. Numerical Model Results

The data confirms the visual trend: the analytical LCM model was **highly conservative**, overestimating the time required to reach the 170°C optimal temperature by **28.7%**.

C. Analysis of Discrepancies

The significant error in the LCM is attributed to two fundamental simplifications that were corrected in the ANSYS model:

1. **Massive Underestimation of Heat Transfer:** The LCM relied on an *estimated* $h_{\text{eff}} = 2000\ \text{W/m}^2 \cdot \text{K}$. The ANSYS results prove that the true, combined heat transfer from **conduction (air gap) + radiation (surface-to-surface)** across the grooved geometry is **far more effective** than this estimate.

2. **Ignored Secondary Heat Path:** This is the most critical finding. The LCM was a 1D model of a *single, isolated nut*. The ANSYS simulation (validated by Figure 7.K) included the full **nut stack**. This proved that **axial nut-to-nut conduction** is a major secondary heat path. The central nut (which we probed) receives heat not only *radially* from the cylinder but also *axially* from its neighboring nuts, significantly accelerating its heating.

Conclusion: The LCM was flawed because it modeled only one (poorly estimated) heat path for one body. The high-fidelity ANSYS model proved that the real-world heating is an accelerated, 3D phenomenon involving *multiple* heat paths. This finding directly leads to a revised and optimized cycle time.

VI. Conclusion and Engineering Contribution

This research successfully fulfilled its primary objective: replacing the uncertainty of the initial theoretical thermal model with a **precise, validated foundation** for the automated hot-insertion press design.

A. Definitive Findings

1. **Quantifiable LCM Error:** The theoretical LCM model was proven to be highly conservative, overestimating the required heating time by $\mathbf{28.7\%}$.
2. **Source of Discrepancy:** The error was definitively traced to the LCM's failure to account for the **3D heat transfer phenomena** identified in the ANSYS analysis: the highly efficient heat transfer across the complex air gap (conduction and radiation) and, critically, the **axial heat conduction (nut-to-nut)**, which significantly accelerates the heating of the central nuts in the stack.

B. Engineering Contribution: Simplified Control Strategy

The core engineering contribution of this work is the ability to simplify the process control based on the validated data:

- **Process Cadence:** The target insertion cycle time is between **15 and 30 seconds**.
- **Validated Heating Time:** The validated time required to reach $\mathbf{170^{\circ C}}$ is only $\mathbf{3.28}$ **seconds**.

Since $\mathbf{3.28\ s \ll 15\ s}$, the heating process is **no longer the bottleneck**. This allows the control logic (PLC) to abandon the "time-based heating" routine in favor of a simpler, more robust, and highly reliable strategy:

1. **Guaranteed Heating:** The heating element is simply kept **ON** as long as the machine is operating. The nuts will reach and maintain their equilibrium temperature well before the press is ready for insertion.
2. **Thermal Damage Control:** The primary safety concern shifts entirely to limiting the **source temperature** ($\boldsymbol{T_\infty}$) itself. By setting $\boldsymbol{T_\infty} = \mathbf{180^{\circ C}}$, we ensure the nut stabilizes just

under the critical degradation temperature of the polycarbonate, thus eliminating the risk of thermal damage through excessive time exposure.

The **validated time of 3.28 seconds** confirms that the thermal system is fully capable of meeting the required production cadence.

VII. Recommendations and Future Work

This investigation successfully validated the *thermal* aspect of the process. The following steps are recommended to complete the full engineering picture and optimize the machine's performance.

A. Proposed Experimental Validation Protocol

The final step in any simulation is physical validation. It is recommended that a real-world test be conducted on the final automated press using:

1. **High-speed thermal imaging (IR camera):** To visually confirm the temperature evolution of the nuts in the stack.
2. **Embedded thermocouples:** To provide physical probe data that can be directly compared to the $\boldsymbol{t}_{\text{ANSYS}}$ curve.

B. Future Investigation: Optimizing the Thermomechanical Process

This report successfully answered: "How long does it take to heat the nut?" This validated thermal data now serves as the **Initial Condition** for the next, more critical phase of the investigation, which must answer: **"What happens *during* the physical insertion?"**

This future study should focus on the following key unanswered questions:

1. What is the true Optimal Temperature $\left(T_{\text{opt}}\right)$?

We used $170^{\circ C}$ as a target, but is this the true optimum? A higher temperature might reduce insertion force but risks polymer degradation (discoloration, weakening). A lower temperature might require excessive force, inducing high stress in the part.

2. What is the required Insertion Force (F_{insert})?

What pneumatic force is required to insert the nut at that optimal temperature? This force dictates the sizing of the press actuator and the mechanical design of the frame.

3. What is the final Joint Quality?

What is the resulting pull-out force and torque resistance of the cooled joint? This is the ultimate metric of success.

C. Required Simulation Elements and Physics

To answer these questions, a **Coupled Thermal-Structural (Thermomechanical) Analysis** is required. This simulation must include:

- **Viscoplastic Material Model:** The Polycarbonate cannot be modeled as a simple solid. It must be defined as a **viscoplastic material**, where its stiffness and yield strength are highly dependent on both **Temperature** and **Strain Rate** (the speed of the insertion).
- **Large Deformation:** The simulation must be configured to handle the large-scale physical displacement of the plastic as it melts and flows around the nut's grooves.
- **Frictional Heating:** The model must include **contact friction** between the nut and plastic, which generates *additional heat* during the high-speed insertion.
- **Residual Stress (Cooling):** The simulation must model the cooling phase *after* insertion to predict how the plastic shrinks onto the nut. This predicts the final "locked-in" stresses, which are critical for long-term crack prevention.

D. Required Skills and Software

This level of simulation is a significant step up in complexity.

- **Software:** The **ANSYS Mechanical** (or Workbench) platform is the ideal tool, as it contains all the necessary non-linear, coupled-field solvers.
- **Skills:** This requires advanced expertise in:
    1. **Non-Linear Material Science:** Finding, interpreting, and implementing complex viscoplastic material models for thermoplastics.
    2. **Advanced FEA:** Deep knowledge of non-linear contact (frictional, large deformation) and transient (time-dependent) solver setup.

3. **Data Interpretation:** Skill in analyzing complex results like plastic strain, stress concentrations, and material flow.

VIII. References

1. Incropera, F. P., DeWitt, D. P., Bergman, T. L., & Lavine, A. S. (2013). Fundamentals of Heat and Mass Transfer (7th ed.). John Wiley & Sons.

(Foundational source for the Lumped Capacitance Model (LCM), Biot Number derivation, and general principles of transient conduction.)

2. NASRALAH, R. (2025). Automation (PLC/HMI), Mechanical Arm and Pneumatic Press Design, and IT Maintenance (Stage PFA). [Rapport Interne Non Publié]. SMCV et ENSET.

(Primary source for the problem statement, initial design goals, and original theoretical LCM hypotheses.)

3. Polycarbonate Resin Technical Data.

(Source for the critical operational temperatures of the Polycarbonate, including the Softening Threshold.)

4. ANSYS, Inc. (2025). ANSYS Mechanical APDL Thermal Analysis Guide. Release 2025 R2.

(Official documentation detailing the numerical implementation of the Transient Thermal Solver, Thermal Contact Conductance (TCC), and Surface-to-Surface Radiation used to model the air gap physics.)