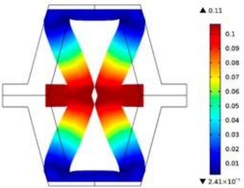
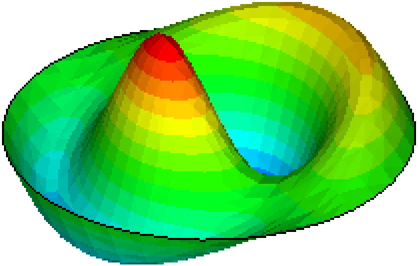


Multiphysics
Mechanics of Solid
Heat Transfer - Thermal Stresses

Tomasz Stręk
Institute of Applied Mechanics, Poznan University of Technology
ul. Jana Pawla II 24, 60-965 Poznan, Poland

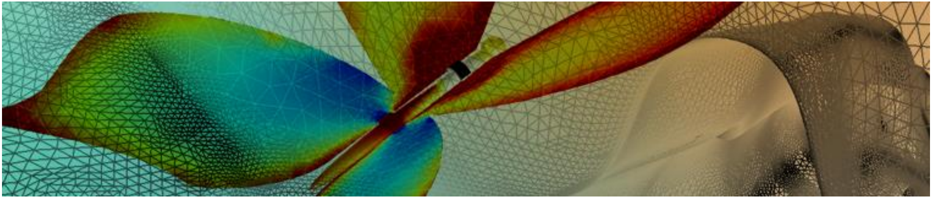
DATE: 2020.05.12



Allan F. Bower

Applied Mechanics of Solids

http://solidmechanics.org/index.html



Introduction to Structural Mechanics

https://www.comsol.com/multiphysics/introduction-to-structural-mechanics

Thermal Expansion and Thermal Stresses

https://www.comsol.com/multiphysics/thermal-expansion-and-thermal-stresses

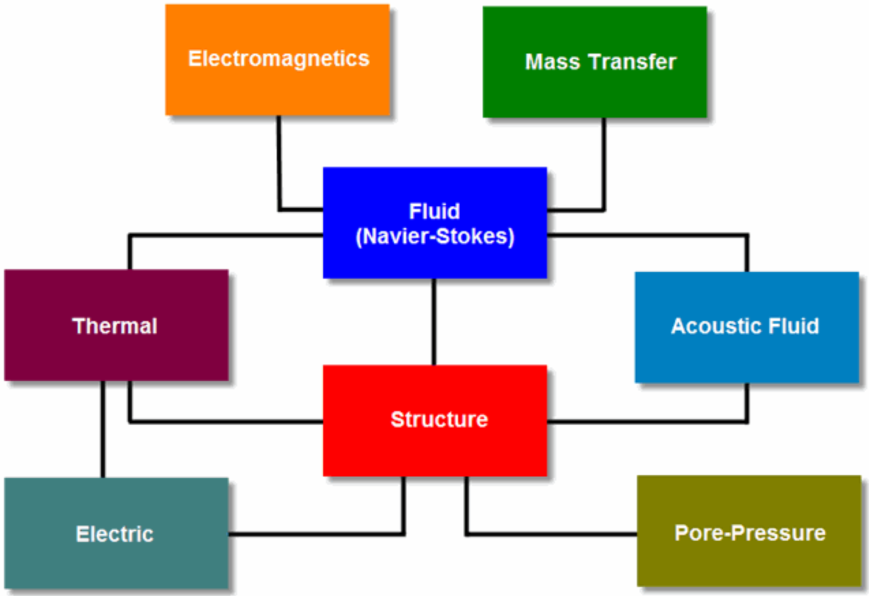
WHAT IS MULTIPHYSICS?

Multiphysics *mul·ti·phys·ics [mul-ti-fiz-iks] noun*

1. Coupled physical phenomena in computer simulation.
2. The study of multiple interacting physical properties.

Understanding Physics

We can describe what happens in the world using sets of physical laws. Since the 1940s, we have been using computers to understand physical phenomena. Originally, computing resources were scarce, so physical effects were observed in isolation. But, as we know, physics do not occur in isolation in the real world.



Electromagnetics
Mass Transfer
Fluid
(Navier-Stokes)
Thermal
Acoustic Fluid
Structure
Electric
Pore-Pressure

HEAT TRANSFER MODELLING

The fundamental law governing all heat transfer is the first law of thermodynamics, commonly referred to as the principle of conservation of energy.

Pierwsza zasada termodynamiki – jedno z podstawowych praw termodynamiki, jest sformułowaniem zasady zachowania energii dla układów termodynamicznych. Zasada stanowi podsumowanie równoważności ciepła i pracy oraz stałości energii układu izolowanego.

Dla układu zamkniętego (nie wymienia masy z otoczeniem, może wymieniać energię) zasadę można sformułować w postaci:

Zmiana energii wewnętrznej układu zamkniętego jest równa energii, która przepływa przez jego granice na sposób ciepła i pracy

$$\Delta U = Q + W$$

gdzie: ΔU – zmiana energii wewnętrznej układu,
Q – energia przekazana do układu jako ciepło,
W – praca wykonana na układzie.

W powyższym sformułowaniu przyjmuje się konwencję, że gdy:

- W> 0 – do układu przepływa energia na sposób pracy,
- W<0 – układ traci energię na sposób pracy,
- Q>0 – do układu przepływa energia na sposób ciepła,
- Q<0 – układ traci energię na sposób ciepła.

W przypadku układu termodynamicznie izolowanego układ nie wymienia energii z otoczeniem na sposób pracy $W = 0$ ani na sposób ciepła $Q = 0$ wówczas: $\Delta U = 0$.

However, internal energy, U, is a rather inconvenient quantity to measure and use in simulations. Therefore, the basic law is usually rewritten in terms of the temperature, T.

For a fluid, the resulting heat equation is:

$$\rho C_p \left(\frac{\partial T}{\partial t} + (\mathbf{v}\cdot\nabla)T\right) + \nabla\cdot\mathbf{q} = \boldsymbol{\tau}\cdot\boldsymbol{\varepsilon} - \frac{T}{\rho}\frac{\partial\rho}{\partial T}\bigg|_{\boldsymbol{p}} \left(\frac{\partial p}{\partial t} + (\mathbf{v}\cdot\nabla)p\right) + Q,$$

where ρ is the density (SI unit: kg/m^3), C_p is the specific heat capacity at constant pressure (SI unit: J/(kg·K)), T is the absolute temperature (SI unit: K), $\mathbf{v}$ is the velocity vector (SI unit: m/s), $\mathbf{q}$ is the heat flux by conduction (SI unit: W/m^2), p is the pressure (SI unit: Pa), $\boldsymbol{\tau}$ is the viscous stress tensor (SI unit: Pa), $\boldsymbol{\varepsilon}$ is the strain-rate tensor (SI unit: 1/s), Q contains heat sources other than viscous dissipation (SI unit: W/m^3).

The equation also assumes that mass is always conserved, which means that the density and the velocity must be related through:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0.$$

The Heat Transfer interfaces use Fourier's law of heat conduction, which states that the conductive heat flux, **q**, is proportional to the temperature gradient:

$$\mathbf{q} = -k\nabla T$$

where k is the thermal conductivity (SI unit: W/(m·K)). In a solid, the thermal conductivity can be anisotropic (that is, it has different values in different directions).

Then k becomes a tensor

$$k = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix}.$$

The second term on the right-hand side of Equation XXX represents viscous dissipation in the fluid. An analogous term arises from the internal viscous damping of a solid.

The operation ":" is a contraction and can in this case be written on the following form: $\mathbf{a}:\mathbf{b} = \sum_n \sum_m a_{nm}\, b_{nm}$.

The third term represents pressure work and is the result of heating under adiabatic compression as well as some thermoacoustic effects. It is generally small for low Mach number flows.

Inserting Equations, reordering the terms and ignoring viscous dissipation and pressure work put the heat equation into a more familiar form:

$$\rho C_p \left(\frac{\partial T}{\partial t} + (\mathbf{v} \cdot \nabla) T \right) + \nabla \cdot (-k \nabla T) = Q.$$

If the velocity is set to zero, the equation governing purely conductive heat transfer in solid is obtained:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = Q.$$

Heat transfer in porous material

Volume average

$$k_{eff} = \Theta_p k_p + (1 - \Theta_p) k_f + k_{disp}$$

Θ_p is volume fraction, k_p – thermal conductivity of porous matrix, k_f – thermal conductivity of fluid, k_{disp} - thermal conductivity for thermal dispersion.

Reciprocal average

$$\frac{1}{k_{eff} - k_{disp}} = \frac{\Theta_p}{k_p} + \frac{1 - \Theta_p}{k_f}$$

Power law

$$k_{eff} = \left(k_p\right)^{\Theta_p}\left(k_f\right)^{1-\Theta_p} + k_{disp}$$

Types of boundary conditions

The heat equation accepts two basic types of boundary conditions: specified temperature and specified heat flux.

The first type of boundary conditions that we can have would be the **prescribed temperature** boundary conditions, also called **Dirichlet conditions**.

The next type of boundary conditions are **prescribed inward heat flux**, also called **Neumann conditions**.

The specified boundary condition is of constraint type and prescribes the temperature on a boundary:

$$T = T_0 \text{ on } \partial\Omega,$$

while the latter specifies the inward heat flux

$$-\mathbf{n} \cdot \mathbf{q} = q_{\mathbf{0}} \text{ on } \partial\Omega,$$

where $\mathbf{q}$ is the conductive heat flux vector (SI unit: W/m2), $\mathbf{n}$ is the normal vector on the boundary and $q_{\mathbf{0}}$ is the inward heat flux (SI unit: W/m2), normal to the boundary.

The inward heat flux, $q_{\mathbf{0}}$, is often a sum of contributions from different heat transfer processes (for example, radiation and convection). The special case $q_{\mathbf{0}} = 0$ is called thermal insulation.

A common type of heat flux boundary conditions is one for which

$$q_{\mathbf{0}} = h\left(T_{inf} - T\right),$$

where T_{inf} is the temperature far away from the modeled domain and the heat transfer coefficient, h, represents all the physics occurring between the boundary and "far away." **It can include almost anything, but the most common situation is that h represents the effect of an exterior fluid cooling or heating the surface of a solid, a phenomenon often referred to as convective cooling or heating.**

This is third type of boundary conditions and use **Newton's law of cooling** and are sometimes called **Robins conditions**. These are usually used when the bar is in a moving fluid and note we can consider air to be a fluid for this purpose.

Heat Transfer Fundamentals

What Is Heat Transfer?

From the kitchen toaster to the latest high-performance microprocessor, heat is ubiquitous and of great importance in the engineering world. To optimize thermal performance and reduce costs, engineers and researchers are making use of finite element analysis. Because most material properties are temperature-dependent, the effects of heat enter many other disciplines and drive the requirement for multiphysics modeling.

For instance, both the toaster and the microchip contain electrical conductors that generate thermal energy as electric current passes through them. As these conductors release thermal energy, system temperature increases as does that of the conductors. If the electric conductivity is temperature dependent, it changes accordingly and, in turn, affects the electric field in the conductor. Other examples of multiphysics couplings that involve heat transfer are thermal stresses, thermal-fluid convection, and induction heating.

Heat transfer is defined as the movement of energy due to a temperature difference. It is characterized by the following three mechanisms:

- *Conduction* is heat transfer by diffusion in a stationary medium due to a temperature gradient. The medium can be a solid or a fluid.
- *Convective heat transfer* is when heat is transported by a fluid motion. (Engineers sometimes uses convection to refer to heat transfer between either a hot surface and a cold moving fluid or a cold surface and a hot moving fluid.)
- *Radiation* is heat transfer via electromagnetic waves between two surfaces (A and B) with different temperatures T_{A} and T_{B}, provided that Surface A is visible to an infinitesimally small observer on Surface B.

The Heat Equation

The mathematical model for heat transfer by conduction is the *heat equation*:

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k \nabla T) = Q$$

Quickly review the variables and quantities in this equation:

- T is the temperature.
- ρ is the density.
- C_p is the *heat capacity* at constant pressure.
- k is *thermal conductivity.*
- Q is a *heat source* or a *heat sink*.

For a steady-state model, temperature does not change with time, and the first term containing ρ and C vanishes.

If the thermal conductivity is anisotropic, k becomes the thermal conductivity tensor:

$$k = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix}$$

Convection and Conduction application mode as:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \nabla \cdot T = \nabla \cdot (k \nabla T) + Q$$

where **u** is the velocity field. This field can either be provided as a mathematical expression of the independent variables or calculated by a coupling to the velocity field from an application mode such as Incompressible Navier-Stokes.

For transport through conduction and convection, a heat flux vector can be approximated by

$$\mathbf{q} = -k \nabla T + \rho C_p T \mathbf{u}$$

where **q** is the heat flux vector. If the heat transfer is by conduction only, **q** is instead determined by

$$\mathbf{q} = -k \nabla T$$

Note: *Heat capacity* here refers to the quantity that represents the amount of heat required to change one unit of mass of a substance by one degree. It has units of energy per mass per degree (J/(kg·K) in SI units). This quantity is also called *specific heat* or *specific heat capacity.*

Boundary Condition Types

The available boundary conditions are:

BOUNDARY CONDITION	DESCRIPTION
$\mathbf{n} \cdot (k\nabla T) = q_0 + h(T_{\mathrm{inf}} - T) + C_{\mathrm{const}}(T_{\mathrm{amb}}^4 - T^4)$	Heat flux
$\mathbf{n} \cdot (k\nabla T) = 0$	Insulation or symmetry
$T = T_0$	Prescribed temperature
$T = 0$	Zero temperature
$\mathbf{n}_1 \cdot (k_1\nabla T_1) = \frac{k}{d}(T_2 - T_1)$ $\mathbf{n}_2 \cdot (k_2\nabla T_2) = \frac{k}{d}(T_1 - T_2)$	Thin thermally resistive layer (pair boundaries only)

Examples of Heat Transfer Models

The following heat transfer benchmark examples show how to model heat transfer using:

- Steady-state and transient analysis
- Temperature, heat flux, convective cooling, and radiation boundary conditions
- Thermal conductivity as a function of temperature

1D Steady-State Heat Transfer with Radiation

The first example shows a 1D steady-state thermal analysis including radiation to a prescribed ambient temperature.

Model Definition

This 1D model has a domain of length 0.1 m. The left end is kept at 1000 K, and at the right end there is radiation to 300 K. For the radiation, the model properties are:

- The emissivity, ε, is 0.98.
- The Stefan-Boltzmann constant, σ, is $5.67 \cdot 10^{-8}$ W/(m^2·K^4).

In the domain, use the following material property:

- The thermal conductivity is 55.563 W/(m·K).

Results

The following plot shows the temperature as a function of position:

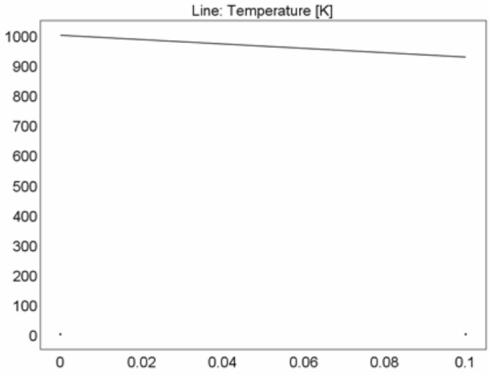


Figure 7-4: Temperature as a function of position.

The benchmark result for the right end is a temperature of 927.0 K. The COMSOL Multiphysics model, using a default mesh with 15 elements, gives a temperature at the end as 926.97 K, which is the exact benchmark value to four significant digits.

2D Steady-State Heat Transfer with Convection

This example shows a 2D steady-state thermal analysis including convection to a prescribed external (ambient) temperature.

Model Definition

This model domain is 0.6 m-by-1.0 m. For the boundary conditions:

- The left boundary is insulated.
- The lower boundary is kept at 100 °C.
- The upper and right boundaries are convecting to 0 °C with a heat transfer coefficient of 750 $W/(m^2 \cdot °C)$.

In the domain use the following material property:

- The thermal conductivity is 52 $W/(m \cdot °C)$.

Results

The following plot shows the temperature as a function of position:

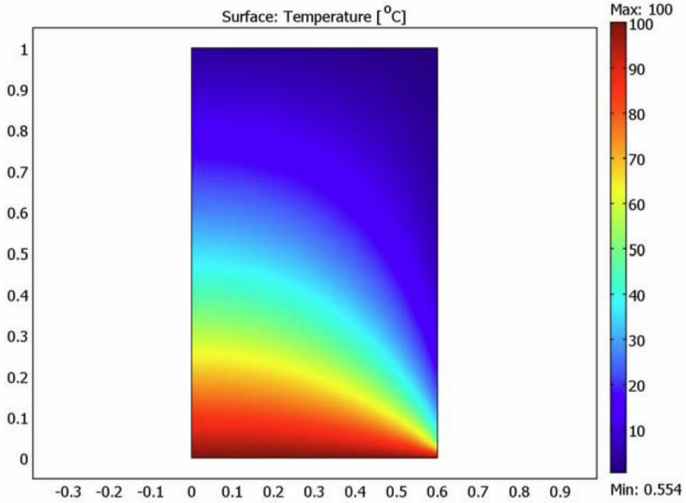


Figure 7-5: Temperature distribution resulting from convection to a prescribed external temperature.

The benchmark result for the target location (x = 0.6 m and y = 0.2 m) is a temperature of 18.25 °C. The COMSOL Multiphysics model, using a default mesh with 556 elements, gives a temperature of 18.28 °C. Successive uniform refinements show temperatures of 18.26 °C and 18.25 °C, converging toward the benchmark result.

2D Axisymmetric Transient Heat Transfer

This example shows an axisymmetric transient thermal analysis with a step change to 1000 °C at time 0.

Model Definition

This model domain is 0.3 m-by-0.4 m. For the boundary conditions, assume the following:

- The left boundary is the symmetry axis.
- The other boundaries have a temperature of 1000 °C. The entire domain is at 0 °C at the start, which represents a step change in temperature at the boundaries.

In the domain use the following material properties:

- The density, ρ, is 7850 kg/m^3
- The heat capacity is 460 J/(kg·°C)
- The thermal conductivity is 52 W/(m·°C)

Results

The following plot shows the temperature as a function of position after 190 seconds:

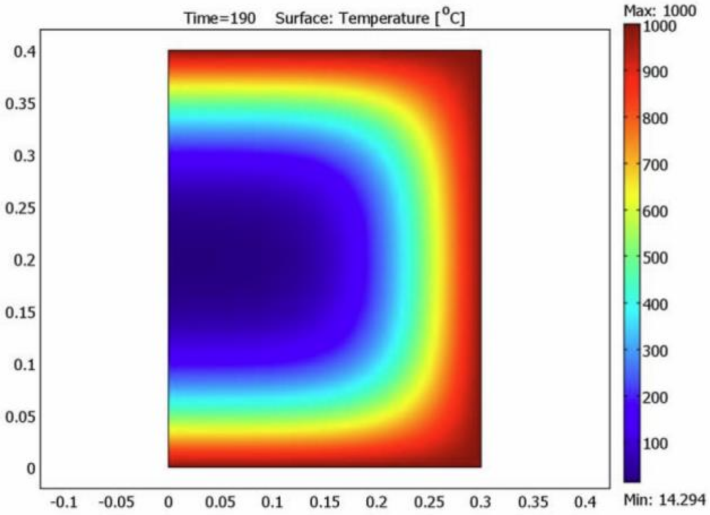


Figure 7-6: Temperature distribution after 190 seconds.

The benchmark result for the target location ($r = 0.1$ m and $z = 0.3$ m) is a temperature of 186.5 °C. The COMSOL Multiphysics model, using a default mesh with about 720 elements, gives a temperature of roughly 186.4 °C.

As an additional postprocessing step, map the axisymmetric solution to 3D using an extrusion coupling variable to show the solution for the entire cylinder (see Figure 7-7).

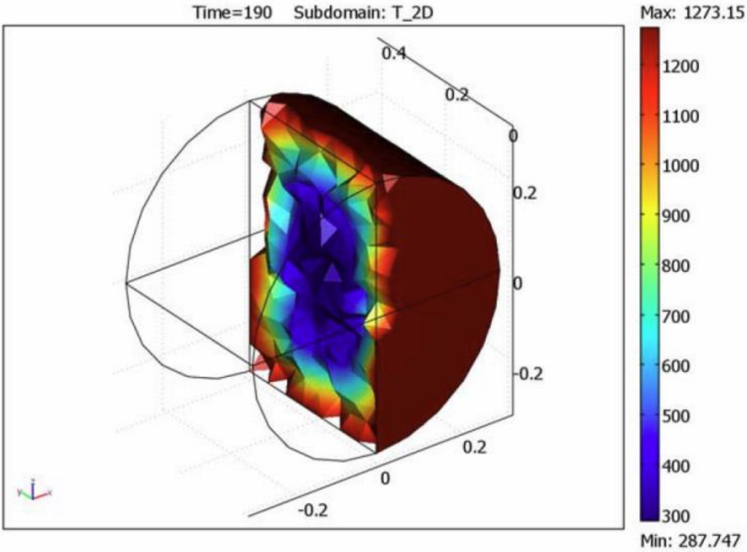


Figure 7-7: Postprocessing of the temperature in the full 3D geometry.

THERMAL STRESSES

Thermal stress is stress created by any change in temperature to a material. These stresses can lead to fracture or plastic deformation depending on the other variables of heating, which include material types and constraints. Temperature gradients, thermal expansion or contraction and thermal shocks are things that can lead to thermal stress. This type of stress is highly dependent on the thermal expansion coefficient which varies from material to material. In general the larger the temperature change, the higher the level of stress that can occur.

Temperature gradients

When a material is rapidly heated or cooled, the surface and internal temperature will have a difference in temperature. Quick heating or cooling causes localized areas of thermal expansion or contraction, this localized movement of material causes thermal stresses.

Imagine heating a cylinder, first the surface rises in temperature and the center remains the same initial temperature. After some time the center of the cylinder will reach the same temperature as the surface. During the heat up the surface is relatively hotter and will expand more than the center. **An example of this is dental fillings** can cause thermal stress in a person's mouth. Sometimes dentist use dental fillings with different thermal expansion coefficients than tooth enamel, the fillings will expand faster than the enamel and cause pain in a person's mouth.

Thermal expansion or contraction

Example of deformation induced by thermal stress on the rails

Material will expand or contract depending on the material's thermal expansion coefficient. As long as the material is free to move, the material can expand or contract freely without generating stresses. Once this material is attached to a rigid body at one end, thermal stresses can be created. This stress, σ, is calculated by multiplying the change in temperature, material's thermal expansion coefficient and material's Young's modulus:

$$\sigma = E\alpha\left(T_0 - T_f\right) = E\alpha\Delta T \ .$$

where E is young's modulus, α is thermal expansion coefficient, T_0 is temperature original, and T_f is the final temperature.

As the temperature increases the stress will be in compression due to the constraints, this is when T_f is greater than T_0. The opposite happens which cooling, when T_f is less than T_0, the stress will be in tension. A welding example involves heating and cooling of metal which is a combination of thermal expansion, contraction, and temperature gradients. After a full cycle of heating and cooling, the metals left with residual stress around the weld.

Thermal shock

This is a combination of a large temperature gradient in addition to rapid change in temperature on brittle materials. The change in temperature causes stresses on the surface that are in tension, which encourages crack formation and propagation. Ceramics materials are usually susceptible to thermal shock. An example is when glass is heated up to a high temperature and then quickly quenched in cold water. As the temperature of the glass falls rapidly, stresses are induced and causes fractures in the body of the glass which can be seen as cracks or even shattering in some cases.

Thermal Expansion

As the temperature changes, most materials react by a change of volume. For a constrained structure, the stresses that evolve with even moderate temperature changes can be considerable. The volume change can be is represented a thermal strain $\boldsymbol{\varepsilon}_{\mathrm{th}}$, which produces stress-free deformations. For a linear elastic material, the constitutive law is

$$\mathbf{S} = \mathbf{S}_0 + \mathbf{C} : (\boldsymbol{\varepsilon} \quad - \boldsymbol{\varepsilon}_0 - \boldsymbol{\varepsilon}_{\mathrm{th}})$$

The Thermal Stress interface combines a Solid Mechanics interface with a Heat Transfer. The coupling occurs on the domain level, where the temperature from the Heat Transfer interface acts as a thermal load for the Solid Mechanics interface, causing thermal expansion.

The thermal strain, $\boldsymbol{\varepsilon}_{\mathrm{th}}$, depends on the coefficient of thermal expansion (CTE) α, the temperature T, and the strain-free reference temperature T_{ref} as

$$\boldsymbol{\varepsilon}_{\mathrm{th}} = \alpha\left(T - T_{ref}\right).$$