Mechanical and Thermal Loading of a Composite Gun Barrel

Kyle Koppenhoefer*, Jeff Crompton, Sergei Yushanov
AltaSim Technologies, Columbus, OH, USA
*Corresponding author: kyle@altasimtechnologies.com

Introduction

Carbon fiber composites offer reduced weight and increased strength for a wide range of products. Integration of carbon fiber composites into gun barrels not only offers superior strength, rigidity and lighter weight compared to traditional barrels but can provide improved thermal response. By tailoring the properties of the carbon fiber and matrix material, composite barrels with thermal characteristics that rapidly dissipate heat can be developed. The resulting improvement in temperature distribution and increased dissipation of heat in the barrel can result in enhanced long-term performance.

To study the effect of combined thermal and pressure loading on the performance of a composite fiber reinforced barrel, a three-dimensional thermal-mechanical finite element model was developed using the Composite Material Module in COMSOL Multiphysics® Version 5.4. The barrel was modeled as a steel liner on top of which a multi-layer stack of carbon-reinforced composite material is wrapped.

Layerwise theory (LWT) formulation was used to model the mechanical response of the composite laminate; structural stability was analyzed using Tsai-Wu orthotropic failure criterion. Internal pressure loading and thermo-mechanical loading of the composite barrel was applied to study the effect of ply orientation and lay-up orientation of the composite layers on the interlaminar stresses and stress distribution along the barrel.

Model description

The composite barrel is shown schematically in Figure 1. The inside radius is $R_0 = 80\ mm$ and the length is $L_0 = 600\ mm$. The barrel consists of a stainless-steel liner and four layers of fiber-reinforced composite materials. The fiber reinforced composite improves the barrel performance. A circumferential ply improves the strength of the laminate structure; axial plies improve the structural rigidity. To increase the stiffness and strength of the barrel, the composite consists of sixteen layers, which includes four layers of 0°-lay, four layers of +45° -lay, four layers of –45° -lay and four layers of 90° -lay from the inside to the outside. The thickness of the metal liner and a single layer of the composite material are 25 mm and 0.25 mm respectively.

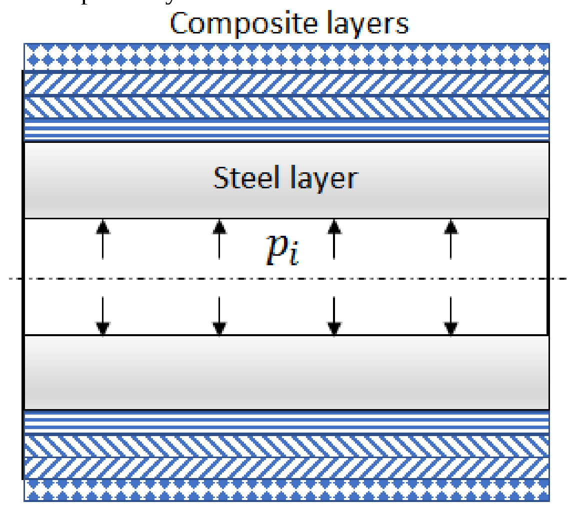

Figure 1. Schematic of composite barrel.

The composite is carbon/epoxy with a fiber volume fraction of $V_f = 0.6$. Thermomechanical properties of the composite material are given in Table 1. The composite laminate is a transversely isotropic material with an elasticity matrix calculated using elastic properties of the constituents via a procedure specified by Christensen [(1)].

Table 1. Thermomechanical properties of Carbon/Epoxy composite.

Fiber Young’s modulus, fiber direction	$E_1^{(f)} = 230\ GPa$
Matrix Young's modulus	$E^{(m)} = 4\ GPa$
Fiber Poisson's ratio	$\nu_{12}^{(f)} = 0.2$
Matrix Poisson's ratio	$\nu^{(m)} = 0.35$
Fiber CTE, fiber direction	$\alpha_1^{(f)} = -0.6 \cdot 10^{-6}$ 1/K
Fiber CTE, perpendicular to fiber direction	$\alpha_2^{(f)} = 8.5 \cdot 10^{-6}$ 1/K
Matrix CTE	$\alpha^{(m)} = 55 \cdot 10^{-6}$ 1/K
Lamina thermal conductivity, fiber direction	$k_1 = 6.2\ W/(m \cdot K)$

Lamina thermal conductivity, perpendicular to fiber direction	$k_2 = 0.5\ W/(m \cdot K)$

Numerical Model

The numerical model includes mechanical and thermal loading of the composite material barrel. An internal pressure is applied at the inner wall of the barrel, and thermal loads are produced by the propellant gas. The breech end of the barrel is fixed. A baseline analysis was performed for internal pressure $p_i = 200\ MPa$. Temperature at the inner diameter of the barrel varies linearly along the barrel length with $T_1 = 1{,}200$ ºF at the breech end and $T_2 = 800$ ºF at the muzzle end. Convection to ambient cools the outer diameter of the barrel.

COMSOL Model

The cylindrical geometry of the gun barrel is shown in Figure 2. It consists of the inner, gray domain that representing the stainless-steel liner. The composite laminate is represented by a single boundary (layered shell), shown in blue.

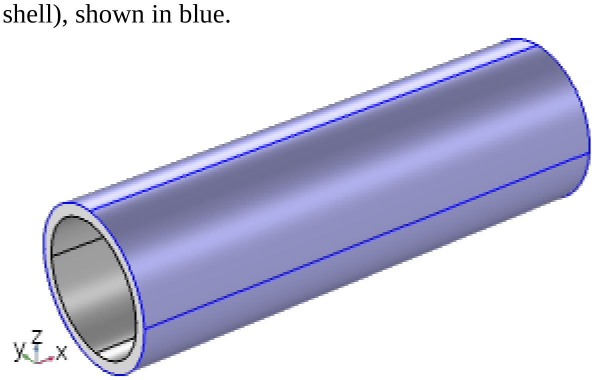

Figure 2. Gun barrel geometry.

The technique for defining material properties for a layered shell differs from other physics interfaces. In general, several different materials are used on the same boundary, so it would not be possible to use the assignment in a normal **Material** node under the **Component** level. Instead, a Global Material should be used. The composite ply properties are defined at the Global Materials node. Then laminate thermo-mechanical properties are specified using Global **Layered Material 1 (lmat1)** option, as shown in Figure 3. Laminate lay-up sequence can be visualized using Layer Stack Preview plot button, as shown in Figure 5.

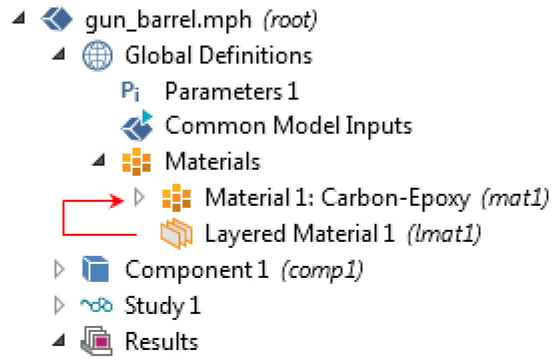

Figure 3. Laminate propertied definition using **Layered Material** option.

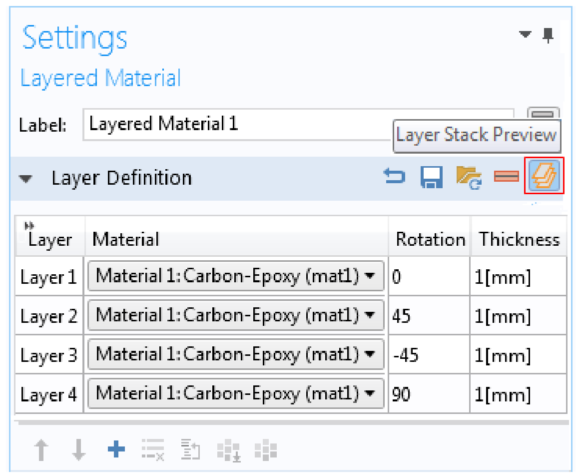

Figure 4. Layered material option references Global Carbon/Epoxy material and defines lay-up angles and layers thicknesses.

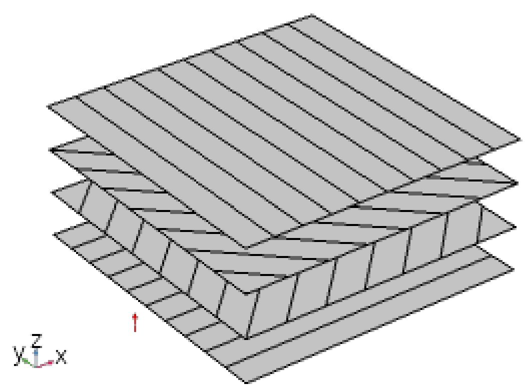

Figure 5. Layer stack preview plot showing a composite laminate having 4 layers with stacking sequence [0/45/-45/90]

Four physics interfaces are used to model the thermo-mechanical response of the gun barrel. A **Solid Mechanics (solid)** physics interface is used for stress analysis in the steel domain and a **Layered Shell (lshell)** physics is used for stress analysis in the composite layers. The **Heat Transfer in Solids (ht)** physics interface is used for thermal analysis in the steel domain and **Heat Transfer in Shells (htlsh)** physics is used for thermal analysis in composite layers. In addition, two Multiphysics interfaces were used to couple momentum balance equations to the energy balance equations. The **Thermal Expansion 1 (te1)** Multiphysics interface couples structural and thermal analysis for the steel domain. The **Thermal Expansion, Layered Shell 1 (tel1)** Multiphysics interface couples structural and thermal analysis for the composite layers domains. The overall physics tree of the COMSOL model is shown in Figure 6.

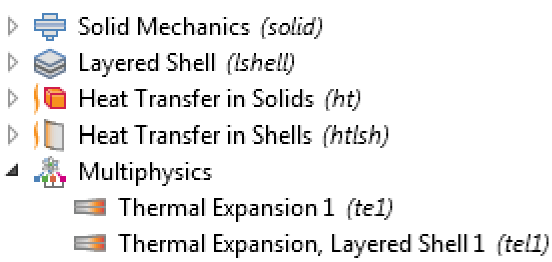

Figure 6. Physics interfaces used to calculate thermo-mechanical response of the gun barrel.

Note that both **Layered Shell (lshell)** and **Heat Transfer in Solids (ht)** physics are active on the exterior boundary (shown in blue in Figure 2) which is actually a layered material. The most fundamental way of attaching the layered material definition to the physical geometrical surfaces is through a **Layered material Link (llmat1)** node, which is added under the **Materials** node within a component, as shown in Figure 7. Note the Layered Material Link option allows us to attach the boundary to the laminate midplane, down side, up side, or at a user defined location. For our case of composite layers placed on a top of steel liner, the correct choice is to set position of the **Layered Material Link** to **Downside on boundary**.

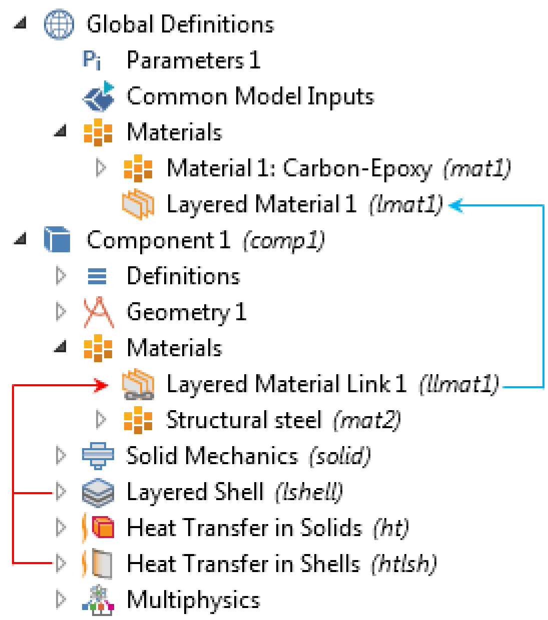

Figure 7. The **Layered Material Link (llmat1)** component node allows attaching multilayered material definition to the physical geometrical surfaces.

By default, the dependent variables of the **Solid Mechanics (solid)** physics interface are (u, v, w) and dependent variables of the **Layered Shell (lshell)** physics interface are $(u2, v2, w2)$. It means that these two physics interfaces are not coupled to each other and deformation of solid steel liner domain is not coupled to the deformation of laminate materials. Coupling can be achieved by imposing continuity of displacements along the solid/shell interface:

$$u_2|_{solid/shell} = u, \quad v_2|_{solid/shell} = v, \quad w_2|_{solid/shell} = w$$

A special boundary condition is available in the **Layered Shell (lshell)** physics interface to establish this continuity condition. The **Prescribed Displacement, Interface** boundary node, as shown in Figure 8, provides this capability. Setup details of this boundary condition are shown in Figure 9.

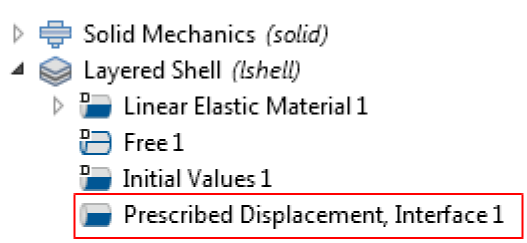

Figure 8. The **Prescribed Displacement, Interface** boundary node used to establish structural coupling between solid and shell domains.

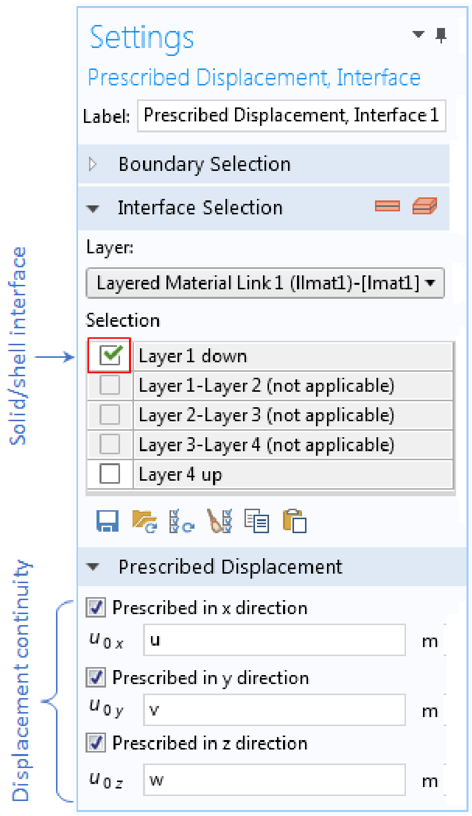

Figure 9. Details of **Prescribed Displacement, Interface** boundary condition setup to impose displacement continuity at the solid/shell interface.

Similarly, by default, the dependent variable of the **Heat Transfer in Solids (ht)** physics interface is T and dependent variables of the **Heat Transfer in Shells (htlsh)** physics interface is $T2$. It means that these two physics interfaces are not coupled to each other and heat flow in solid steel liner domain is not coupled to the heat flow in laminate materials. Coupling can be achieved by imposing continuity of temperature along the solid/shell interface:

$$T_2|_{solid/shell} = T$$

A special boundary condition is available in the **Heat Transfer in Shells (htlsh)** physics interface to establish this continuity condition. The **Temperature, Interface** boundary node, as shown in Figure 10, provides this continuity. Setup details of this boundary condition are shown in Figure 11.

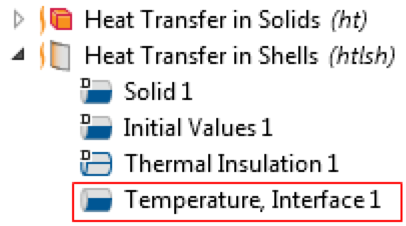

Figure 10. The **Temperature, Interface** boundary node used to establish thermal coupling between solid and shell domains.

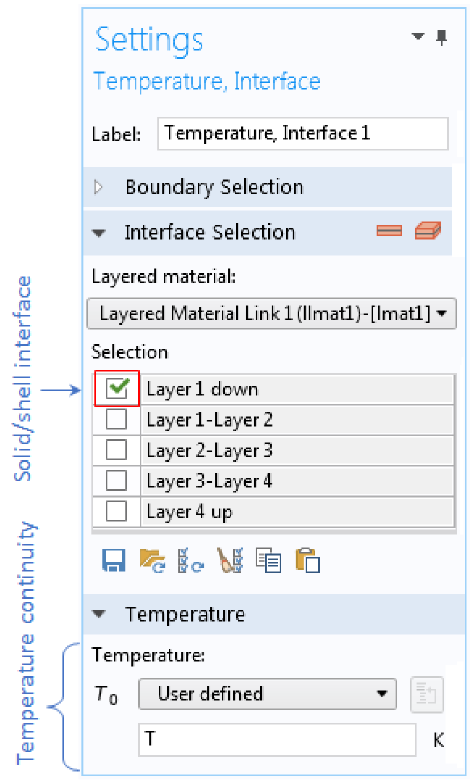

Figure 11. Details of **Temperature, Interface** boundary condition setup to impose temperature continuity at the solid/shell interface.

Convective cooling at the outer surface of laminate is imposed using a **Heat Flux, Interface** boundary node available in v. 5.4. This allows addition of heat flux on the exterior boundaries of the layered shell, as well as heat flux at the internal interfaces between layers. In our case, this boundary condition is used to impose convective cooling at the exterior boundary of the laminate, as shown in Figure 12.

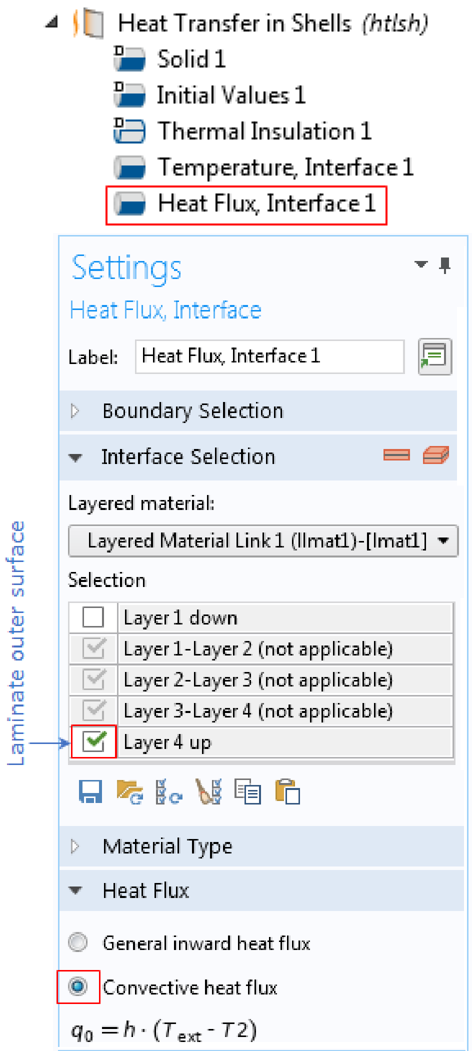

Figure 12. Boundary condition used to set up convection cooling at the exterior boundary of the laminate.

The remaining boundary conditions are standard boundaries of the solid domain. The **Temperature** boundary node is used to impose linearly varying temperature at the interior boundaries of steel liner for the **Heat Transfer in Solids (ht)** physics. The Boundary Load node is used to impose internal pressure for the **Solid Mechanics (solid)** physics. Finally, a **Rigid Motion Suppression** domain node is used to eliminate rigid body motion of the steel liner solid domain.

Simulation Results

Temperature distributions in steel domain and composite laminate are shown in Figure 13. Convective cooling decreases the temperature of laminate.

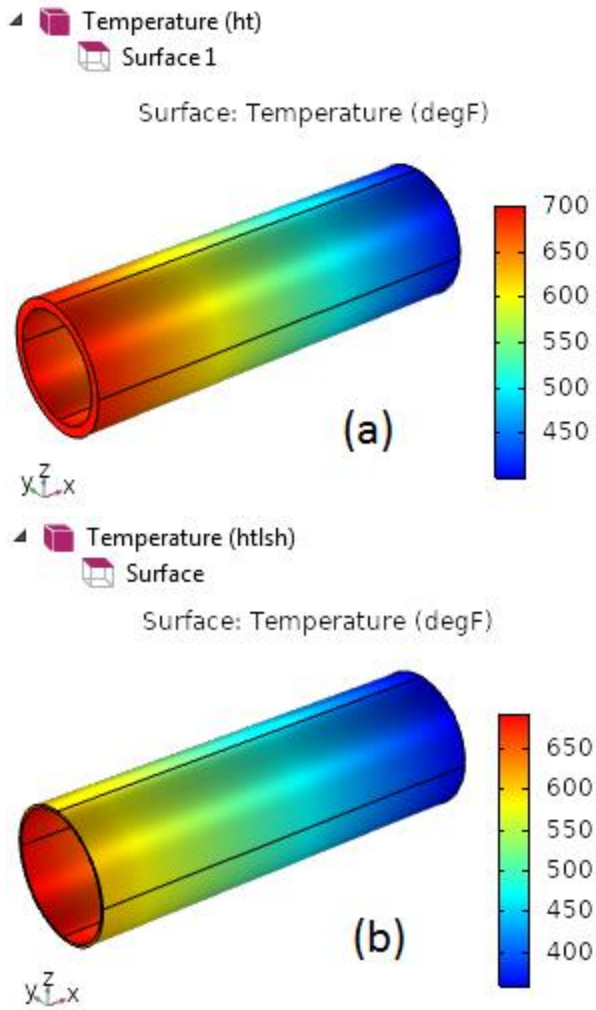

Figure 13. Temperature distribution in steel liner (a) and in laminate (b).

Distribution of stress through the thickness of laminate at the breech end is shown in Figure 14.

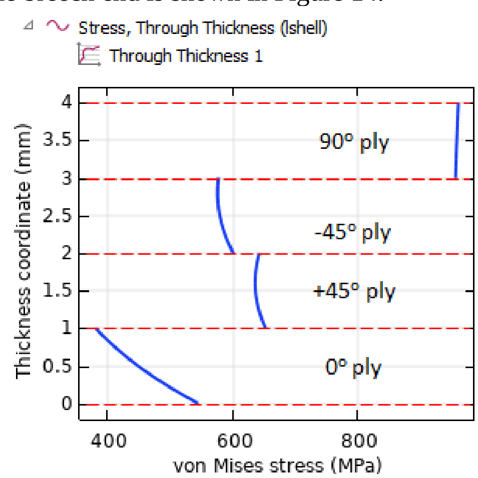

Figure 14. Stress distribution through laminate thickness.

Failure Analysis

In composite laminates, it is common to have different failure modes. Thus, it becomes essential to perform various kinds of failure analysis for composite laminates. Both the laminate theories allow the computation of failure indices or safety factors based on the following criteria. In COMSOL Multiphysics laminate theories allows computation of failure indices or safety factors based on the various criteria. For orthotropic laminate the appropriate failure criterion is Tsai-Wu Orthotropic. It can be activated by adding a **Safety** sub-node to the **Linear Elastic Material** node in the **Layered Shell (lshell)** physics, as shown in Figure 15. Orthotropic strength properties of the Carbon/Epoxy composite are given in Table 2.

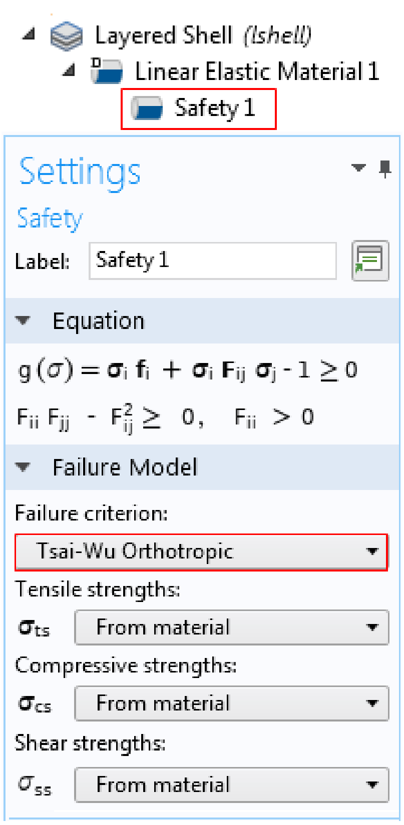

Figure 15. The **Safety** sub-node used to perform failure analysis of composite laminate.

Table 2. Strength properties of the Carbon/Epoxy material.

Tensile strength, longitudinal	500 MPa
Tensile strength, transverse	5 MPa
Compressive strength, longitudinal	350 MPa
Compressive strength, transverse	75 MPa
Shear strength	35 MPa

In addition to the standard result presentation tools, COMSOL Multiphysics offers specialized methods for evaluating and plotting results in composite laminates. Even though the composite laminate is modeled as a surface (2D) geometry, we can visualize the results on geometry with finite thickness using a **Layered Material Slice** plot. This post-processing capability is used in Figure 16 to plot Tsai-Wu safety factor (predefined variable lshell.lemm1.sf1.s_f) for 0° ply and +45° ply. A value of safety factor less than 1 suggests the Tsai-Wu Orthotropic failure criterion is satisfied, such that the +45° ply fails near the breech side.

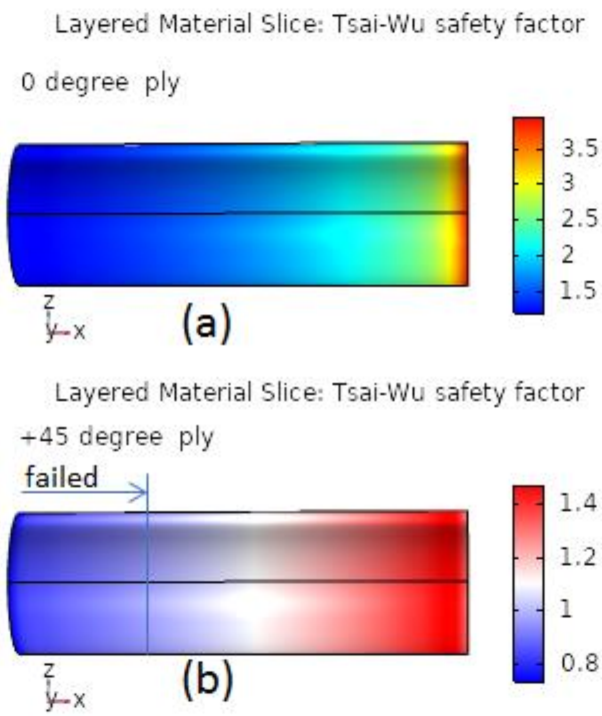

Figure 16. The Safety sub-node used to perform failure analysis of composite laminate.

Conclusions

The computational model developed for this work simulates the composite gun barrel under thermal and mechanical loadings. Failure criterion for orthotropic material is used to predict ply failure near the breech end.

References

1. Christensen, R. M., *Mechanics of Composite Materials*, Wiley, New York, 1979.