Thermo-Structural Analysis of Bimetallic Strip
Final Project for AE 420/ME 471: Finite Element Analysis

Aaron Havens, Akhilesh Somani, Aravind Ramakrishnan, Kamalpreet Singh
University of Illinois at Urbana-Champaign, IL, USA

August 31, 2020

Abstract

Bimetallic strips have several useful engineering applications in thermo-mechanical systems such as thermometers and circuit breakers. These structures are especially useful because their function results purely from design of material properties, yielding simple and reliable actuation. However, consisting of two layered materials of various complex geometries, these structures can be difficult to design and model for such applications requiring a certain precision. In this manuscript we formulated a finite element method problem to model a simple bimetallic cantilever structure using 8-node isoparametric elements. Using a Gaussian quadrature integration we can compute the resulting deformation under a uniform temperature field. This general isoparametric formulation can be reasonably extended to more complicated geometries. We compare the resulting solution fields (temperature, stress, displacement) to a commercial finite element software *ABAQUS*. Additionally, we perform a convergence study to optimize the number of elements in the model. We observe that there is less than 3.5% discrepancy in displacement of the free end of the cantilever when comparing our method and the ABAQUS solution, however we observe the desired qualitative behavior in the structure deflection.

1 Introduction

A bimetallic strip consist of two metal plates layered and riveted together at both ends (depicted below as figure 1). If the metals are chosen as so to have different thermal expansion coefficients, the strip will deflect when undergoing heating due to the geometric constraint of the riveted ends. Bimetallic strips can be used in a number of engineering applications serving as thermometer or a thermally dependent mechanical elements such as circuit breakers (figure 2) and MEMS applications [4]. Accurately modeling the thermo-mechanical response to these bimetallic strip structures is crucial to their engineering design and application, without expensive and time-consuming empirical design iterations. Using this accurate model, one could directly optimize the design for a specific engineering use case and ensure the the element properties stay in an expected range during nominal conditions, especially in such safety critical applications. In this manuscript we formulate and evaluate a finite element method (FEM) approach to model the thermal response of a simple bimetallic strip composed of steel and aluminum. In parallel, we model the same system using a commercial FEM modeling software *ABAQUS* [5] to compare the quality of our FEM approach. We would like to emphasize that although only study a simple cantilever case in this manuscript, in principle, this general iso-parametric FEM formulation may be extended to other geometries with similar boundary conditions.

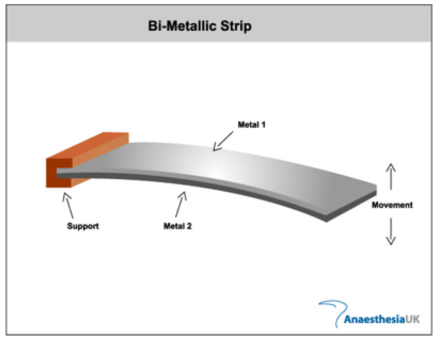

Figure 1: A cantilever bi-metallic strip [3].

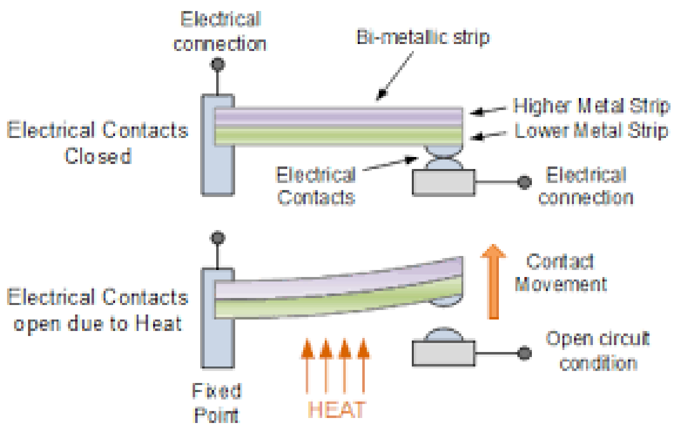

Figure 2: The bimetallic strip cantilever mechanism can serve as a circuit breaker in an electrical circuit [2]. These thermal actuation mechanism based on only material properties is inherently simple and reliable.

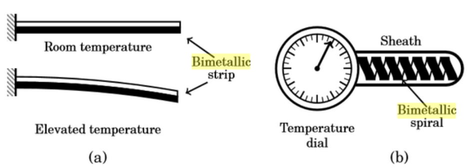

Figure 3: Schematics of Bimetallic Strip Thermometers (a) Cantilever type, (b) Spiral Type [1]

2 Formulation

In this project, authors will model the bimetallic strip present in an Iron Box. The main function of the bimetallic strip is to maintain the temperature inside the box. It is achieved due to the bending of the strip as it breaks the circuit. By adjusting the screw, different temperature levels can be maintained inside the box. Essentially, adjusting screw restricts the deflection of bimetallic strip thus requiring different temperatures to cause the deflection. Therefore, modelling bimetallic strip deflections for various temperatures is useful in calibrating the adjusting screw.

The bimetallic strip is meshed using 3-D brick elements and nodal displacements are found using code written in MATLAB. The system is then modeled in ABAQUS, a commercial finite element software, and the corresponding solutions given by the code and by ABAQUS software are compared.

2.1 Potential function ODE

Potential energy function is given by

$$\Pi = \frac{1}{2}\int_V < d > [B]^T[E][B]\{d\}dV - \int_V < u > \{F\}dV - \int_{\delta_T} < u > \{T\}dS$$

2.2 FEM problem formulation

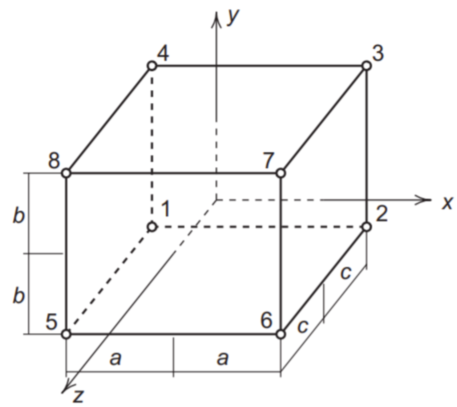

Figure 4: An 8-node brick element is used for the formulation.

Shape functions of isoparametric cubic element:

$$N_1 = \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta), \qquad N_5 = \frac{1}{8}(1-\xi)(1-\eta)(1+\zeta)$$

$$N_2 = \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta), \qquad N_6 = \frac{1}{8}(1+\xi)(1-\eta)(1+\zeta)$$

$$N_3 = \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta), \qquad N_7 = \frac{1}{8}(1+\xi)(1+\eta)(1+\zeta)$$

$$N_4 = \frac{1}{8}(1-\xi)(1+\eta)(1-\zeta), \qquad N_8 = \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta)$$

The isoparametric element is an 8-node element, hence it has 24 degrees of freedom:

$$\langle d \rangle = \Big\langle u_x^1 \quad u_y^1 \quad u_z^1 \quad u_x^2 \quad u_y^2 \quad u_z^2 \quad ..... \quad u_x^8 \quad u_y^8 \quad u_z^8 \Big\rangle$$

$$\begin{Bmatrix} \tilde{u}_x(x,y,z) \\ \tilde{u}_y(x,y,z) \\ \tilde{u}_z(x,y,z) \end{Bmatrix} = [N]_{3\times 24}\{d\}_{24\times 1}$$

$$[N] = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & ..... & N_8 & 0 & 0 \\ 0 & N_1 & 0 & 0 & N_2 & 0 & ..... & 0 & N_8 & 0 \\ 0 & 0 & N_1 & 0 & 0 & N_2 & ..... & 0 & 0 & N_8 \end{bmatrix}$$

$$\text{Elastic tensor : } [E] = \begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix}$$

$$\text{where} \quad \lambda = \frac{\nu E}{(1+\nu)(1-2\nu)} \quad \text{and} \quad \mu = \frac{E}{2(1+\nu)}$$

$$\{\epsilon\} = \begin{Bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \end{bmatrix} \begin{Bmatrix} u_x \\ u_y \\ u_z \end{Bmatrix} = [\partial]\{u\}$$

The stress and strain in an element may be interpolated using the nodal displacements and differentiated shape functions.

$$[B] = [\partial][N], \quad <\tilde{\epsilon}>=< d > [B]^T, \quad \{\tilde{\sigma}\} = [E]\{\tilde{\epsilon}\} = [E][B]\{d\}$$

Using 8-point Gaussian quadrature for 3-dimensional case,

$$\int_{-1}^{1}\int_{-1}^{1}\int_{-1}^{1} f(\xi,\eta,\zeta)d\xi d\eta d\zeta = \sum_{i=1}^{8} w_i f(\xi_i,\eta_i,\zeta_i)$$

Here, $w_i = 1$ for $i = 1, 2, ..., 8$ and $\xi_i = \pm\frac{1}{\sqrt{3}}$

The Jacobian matrix for the coordinate transformation $(\xi,\eta,\zeta) \to (x,y,z)$ is given by:

$$[J] = \begin{bmatrix} \dfrac{\partial x}{\partial \xi} & \dfrac{\partial y}{\partial \xi} & \dfrac{\partial z}{\partial \xi} \\ \dfrac{\partial x}{\partial \eta} & \dfrac{\partial y}{\partial \eta} & \dfrac{\partial z}{\partial \eta} \\ \dfrac{\partial x}{\partial \zeta} & \dfrac{\partial y}{\partial \zeta} & \dfrac{\partial z}{\partial \zeta} \end{bmatrix}$$

$$\begin{Bmatrix} \dfrac{\partial N_i}{\partial x} \\ \dfrac{\partial N_i}{\partial y} \\ \dfrac{\partial N_i}{\partial z} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \dfrac{\partial N_i}{\partial \xi} \\ \dfrac{\partial N_i}{\partial \eta} \\ \dfrac{\partial N_i}{\partial \zeta} \end{Bmatrix}$$

For this problem of uniform temperature rise in solids with physical constraints, the thermal strain is given by $\epsilon = -\alpha\Delta T$, and the thermal stress is given by $\sigma = -\alpha E \Delta T$ (compressive stress).

Here, α is the coefficient of linear thermal expansion of the material and ΔT is the temperature rise from a reference temperature.

$$[B_i] = \begin{bmatrix} \dfrac{\partial N_i}{\partial x} & 0 & 0 \\ 0 & \dfrac{\partial N_i}{\partial y} & 0 \\ 0 & 0 & \dfrac{\partial N_i}{\partial z} \\ \dfrac{\partial N_i}{\partial y} & \dfrac{\partial N_i}{\partial x} & 0 \\ \dfrac{\partial N_i}{\partial z} & 0 & \dfrac{\partial N_i}{\partial x} \\ 0 & \dfrac{\partial N_i}{\partial z} & \dfrac{\partial N_i}{\partial y} \end{bmatrix}, \quad [B] = \begin{bmatrix} B_1 & B_2 & B_3 & .... & B_8 \end{bmatrix}$$

We can rewrite the potential function Π using the elements $[B]$ and displacements $\{d\}$.

$$\Pi = \frac{1}{2}\int_V \langle\epsilon\rangle\{\sigma\}dV - \int_V \langle u\rangle\{F\}dV - \int_{\delta_T} \langle u\rangle\{T\}dS$$

$$\Pi = \frac{1}{2}\int_V ([B]\{d\} - \{\epsilon^t\})^T[E]([B]\{d\} - \{\epsilon^t\})dV - \int_V ([N]\{d\})^T\{F\}dV - \int_{\delta_T} ([N]\{d\})^T\{T\}dS$$

Nodal displacements $\langle d\rangle$ are found by minimizing Π:

$$\frac{d\Pi}{du_i} = 0$$

$$\int_V [B]^T[E][B]dV\{d\} - \int_V [B]^T[E]\{\epsilon^t\}dV - \int_V [N]^T\{F\}dV - \int_S [N]^T\{T\}dS = 0$$

Hence we obtain the linear system of equations:

$$[k]\{d\} = \{f\}$$

where,

$$[k] = \int_V [B]^T[E][B]dV$$

$$\{f\} = \int_V [N]^T \{F\} dV + \int_S [N]^T \{T\} dS + \int_V [B]^T [E] \{\epsilon^t\} dV$$

Here, $[k]$ is the element stiffness matrix, and $\{f\}$ is the local load vector. $\{\epsilon^t\}$ is the thermal strain, given by: $\{\epsilon^t\} = \left\{\alpha\Delta T \quad \alpha\Delta T \quad \alpha\Delta T \quad 0 \quad 0 \quad 0\right\}$

2.3 Geometry

In this project, a bimetallic strip made up of steel and aluminum is modeled. The length, width and depth of the bimetallic strip are 80 mm, 20 mm and 5 mm respectively. The material properties for steel and aluminum are shown in table 1.

Table 1: Material Properties

	Young's Modulus, E (GPa)	Poisson's Ratio, ν	Expansion Coefficient, α ($^\circ C^{-1}$)
Steel	210	0.3	1.1×10^{-5}
Aluminium	70	0.3	2.3×10^{-5}

The geometry is shown in Figure: 5. Steel is on the top side. To create this model, a cuboid is partitioned into two parts using a datum plane (to simulate the condition that the two materials are welded along the length).

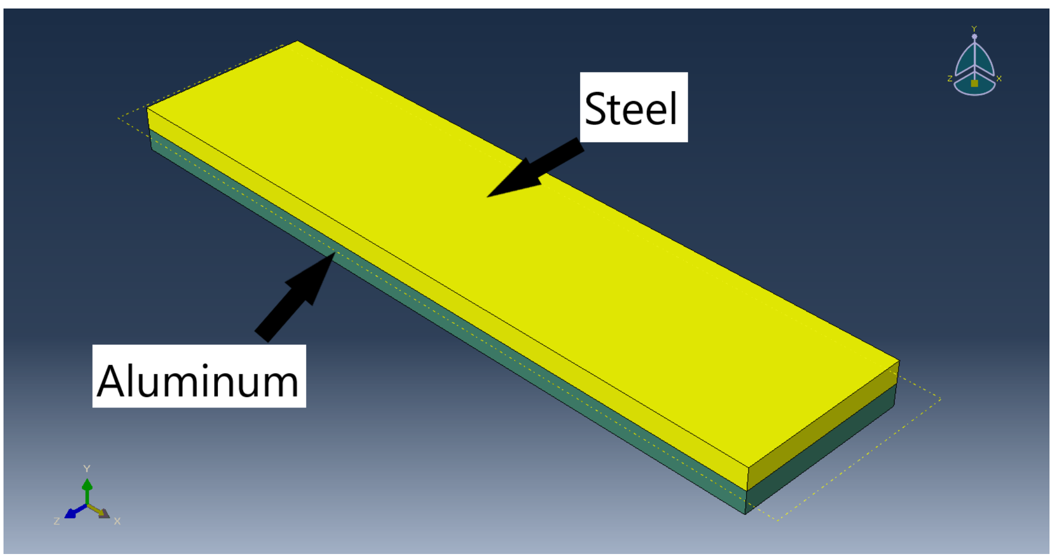

Figure 5: Modeling of Bimetallic Strip in ABAQUS

The meshing is shown in Figure: 6. 8-node cuboid shaped elements have been chosen for the analysis.

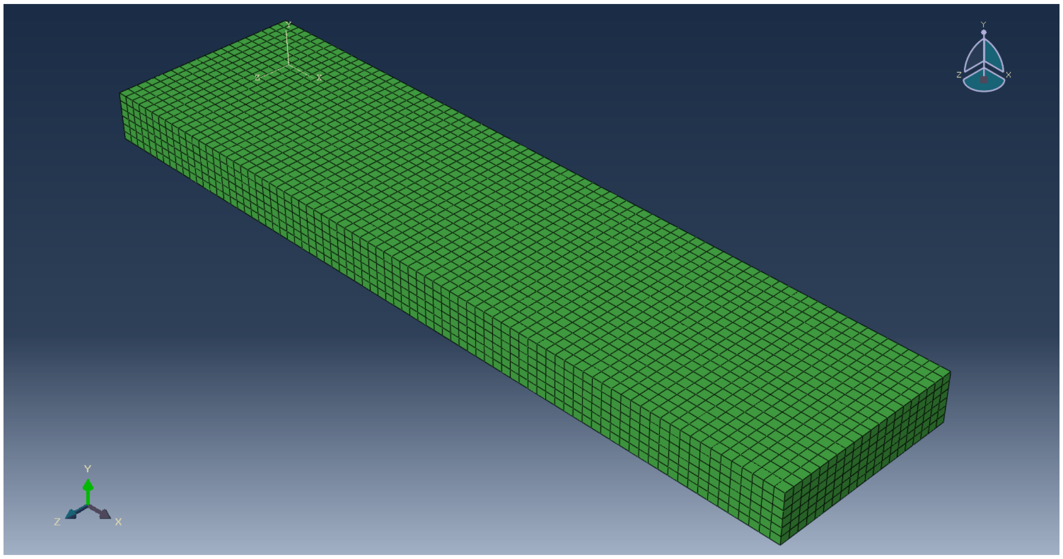

Figure 6: Mesh Structure of Bimetallic Strip in ABAQUS

2.4 Loading and Boundary Conditions

One end of the strip is fixed whereas the other is free. A uniform temperature field is applied along the entire strip. The initial temperature is taken as 25 $^{\circ}C$ as it is assumed as room temperature. In general, the temperature range inside the iron box is 155 - 195 $^{\circ}C$. Therefore, the change in the temperature range is 130 - 170 $^{\circ}C$. However, for this study, only one temperature difference has been considered.

- Temperature constant throughout volume
- $T(x, y, z) = 25$ initially throughout V (volume)
- $T(x, y, z) = 155$ finally throughout V (volume)
- Thin cantilever beam ($h << l \sim b$)
- $u_x = u_y = u_z = 0$ and $u'_x = u'_y = u'_z = 0$ on one of the faces of the strip.

3 Results and Discussions

3.1 Convergence study

A convergence study is performed to optimize the number of elements used in computations. Convergence study is performed and compared between ABAQUS and MATLAB. Two cases in ABAQUS are used for this study: C3D8 and C3D8R. C3D8 refers to the use of a cuboid with 8 ($2 \times 2 \times 2$) Gauss Quadrature integration points. C3D8R refers to reduced integration (where only one integration point is used in Gauss Quadrature). Figure 7 shows the trend of the variation of magnitude of maximum spatial displacement for the cases.

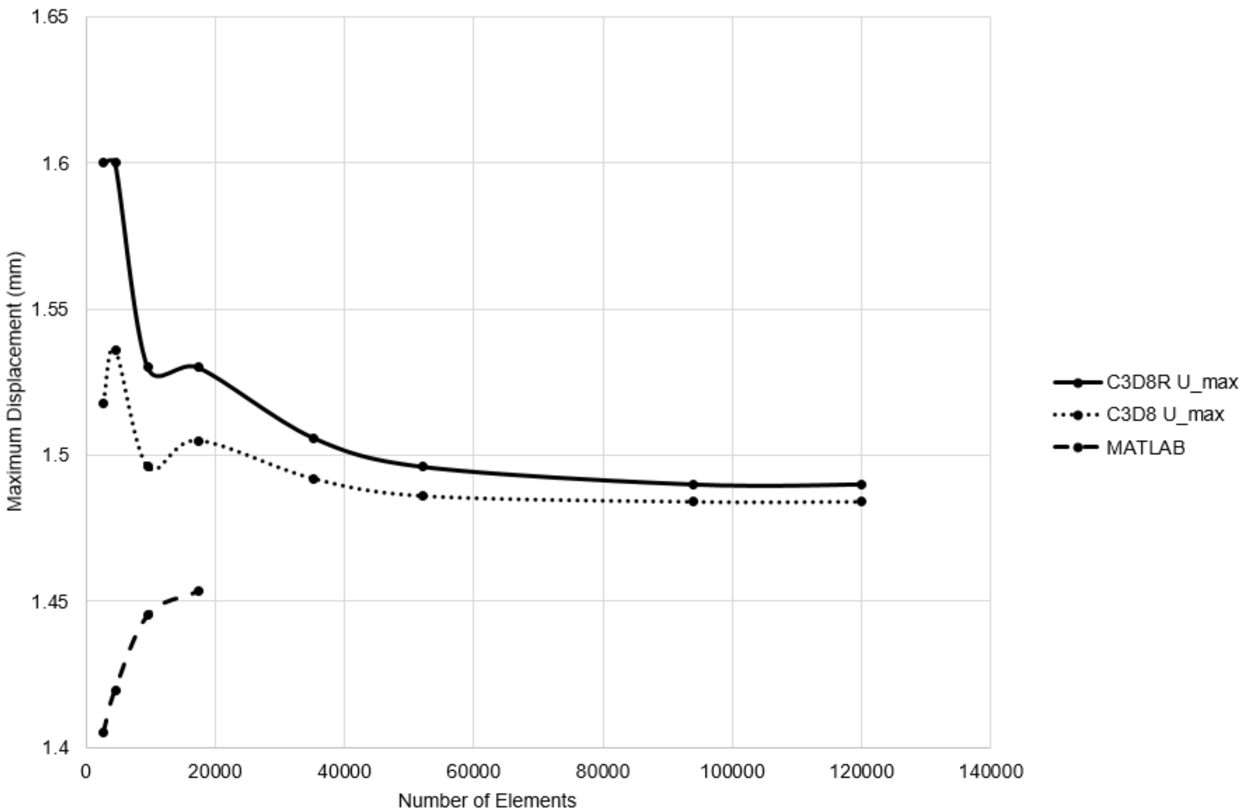

Figure 7: Plot of maximum displacement (U - magnitude) obtained using C3D8 integration rule and reduced C3D8 integration rule against number of elements (in ABAQUS)

Trends show that the value of U (magnitude) stabilizes after ~ 35000 elements. However, using this value of number of elements in ABAQUS means using the same value in MATLAB (to enable comparison between the FE code and ABAQUS). Using 35000 elements results in a lengthened computational time in MATLAB. Hence, to reduce this time (and still be fairly consistent with the results), a value of ~ 17000 elements is used in ABAQUS and MATLAB. To compare the results of ABAQUS and MATLAB, model of 9600 elements is used as it is closer to the stabilized value than 17000 elements. So, unless explicitly mentioned, the number of elements used in MATLAB and ABAQUS analysis are 9600 elements. It is also observed that the percentage difference between ABAQUS and MATLAB results are around 30% for model with lower number of elements (< 1000). As the number elements increases in the model the percentage error reduces to less than 3.5%.

Please note that the convergence study is performed only upto 17334 elements as the computational time greatly increases for more number of elements. However, the error with the ABAQUS results is also insignificant (lower than 3.5 percent). The percentage difference between ABAQUS and MATLAB results for different number of element is shown in the below table 2.

Table 2: Percentage Difference

Number of elements	U_{ABAQUS}(mm)	U_{MATLAB}(mm)	% error
968	1.832	1.303	28.9
2756	1.518	1.405	7.44
4556	1.536	1.420	7.57
9600	1.496	1.445	3.40
17334	1.505	1.453	3.43

To the authors' best knowledge, error is not found in the MATLAB code. Therefore, the discrepancies might be due to different techniques and considerations adopted by ABAQUS. ABAQUS might use a different technique to solve the model with lower number of elements whereas in the code a standard approach is used irrespective of number of elements. This can explain the higher difference for lower number of elements in the model. Also, it might be the way thermal loading is considered in ABAQUS. Thermal loading might be considered differently in ABAQUS than the conventional approach. To the authors' best knowledge consideration of thermal loading could not be identified in the original documentation of ABAQUS.

3.2 Post-Processing

The displacement along the Y-direction is of important consideration as the values are used for the calibration of adjusting screw. It is due to the different coefficient of linear expansion that the combined structure bends in Y-direction, and has significant U2 (Y-component of displacement field) compared to X and Z directions. This significant value of U2 is utilized in numerous applications like switches. Figure 8 shows the variation of U2 along the length of the strip.

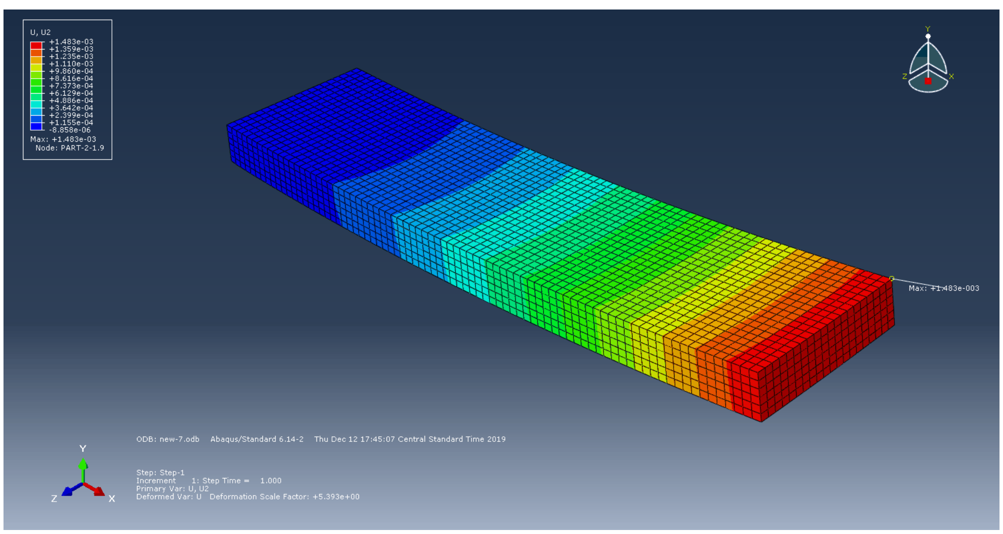

Figure 8: Variation of U2 (Displacement component along Y) in ABAQUS

Figure 9 shows the deformed structure vs the original structure. The structure bends in a particular direction depending on lower coefficient of thermal expansion (COTE). In this case, steel has a smaller COTE, hence it expands less in comparison of Aluminium.

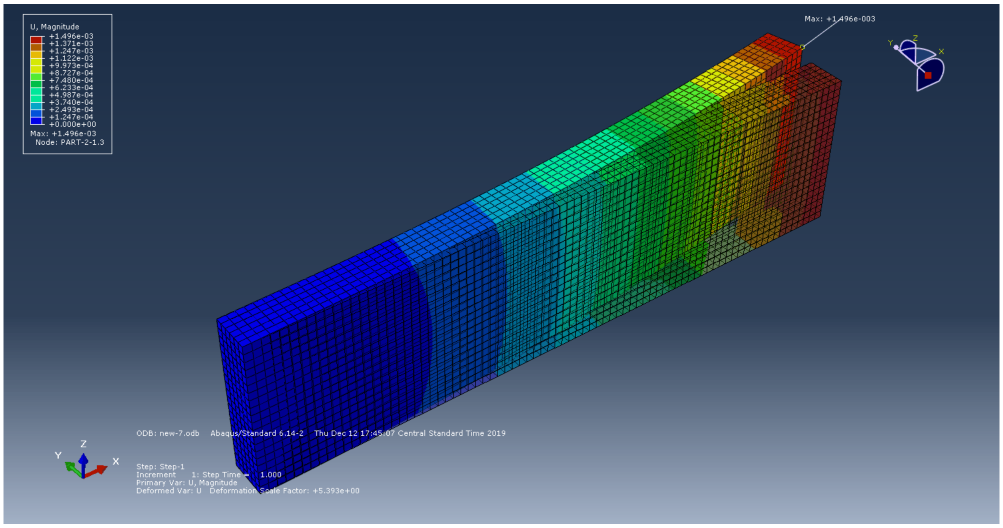

Figure 9: Original vs Deformed strucutre (U - magnitude) in ABAQUS

Figure 10 shows the deformed structure vs the original structure plotted using MATLAB. It can be seen from both figures that the values and deformed shape is similar.

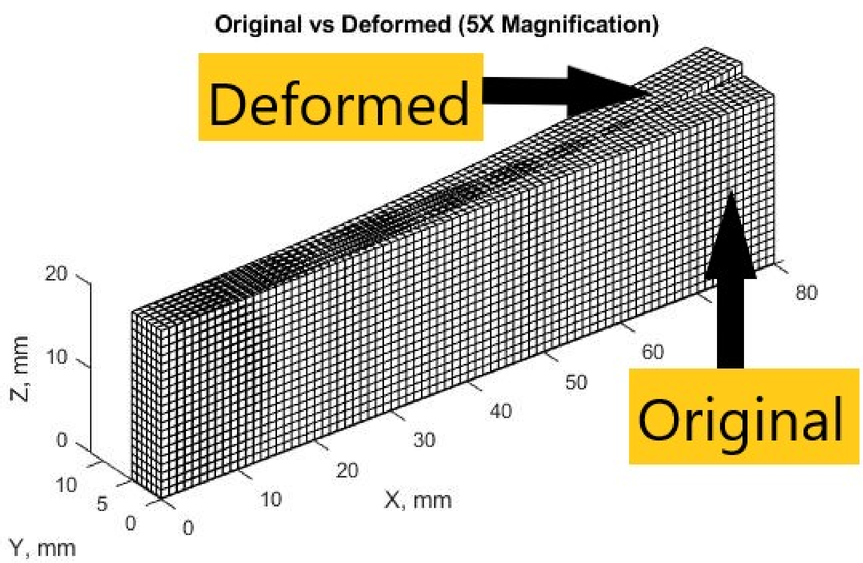

Figure 10: Original vs Deformed Structure (with 5 × magnification) in MATLAB

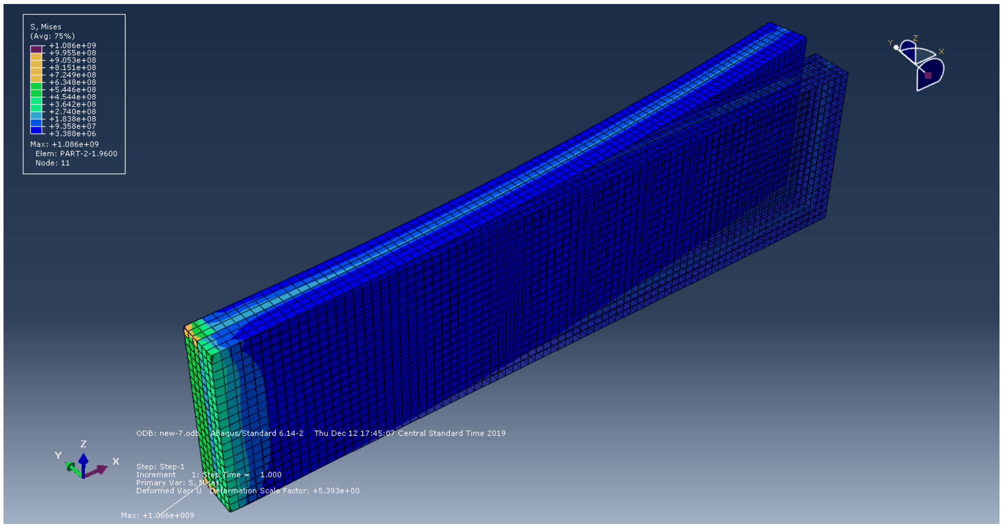

Figure 11: Von-Mises Stress in ABAQUS

Figure 11 shows the variation of Von-Mises stress in the entire structure. The stress is maximum at the face where the structure is fixed.

Table 3: Stress (Von-Mises) Comparison

	Absolute Maximum Stress (MPa),	Absolute Minimum Stress (MPa))
ABAQUS	1086	3.338
MATLAB	596.3	2.162

The absolute value of the maximum stress differs substantially in MATLAB and ABAQUS. The minimum value also differs by a substantial percentage. The exact reason could not be identified as Von-Mises stresses do not converge to a stable solution for both ABAQUS and MATLAB. The Von-Mises stresses for different number of elements is shown in the table 4,

Table 4: Von-Mises Stress

Number of elements	Von-Mises Stress$_{ABAQUS}$(MPa)	Von-Mises Stress$_{MATLAB}$(MPa)
968	910	418
2756	892	456
4556	992	521
9600	1086	596
17334	1239	701

4 Conclusions

- The deformed structures in both ABAQUS and MATLAB show the trend as expected from scientific intuition
- The displacement values begin to converge and the error becomes insignificant for the number of elements greater than or equal to 9600

- The values of displacements differ less than 3.5 percent in ABAQUS and MATLAB for 9600 elements
- For a temperature difference of 130 $^{\circ}C$, a maximum displacement of 1.496 mm occurs at the end face (face away from the fixed end). This displacement value can be used for applications like switches and circuit breakers
- The values of Von-Mises stresses in ABAQUS and MATLAB do not converge to a stable solution. Therefore, the reason for the difference in the values could not be identified accurately

References

[1] . Bimetallic strip image, 2018. [Online; accessed Dec 12, 2019].

[2] . Thermal circuit breaker, 2019. [Online; accessed Dec 12, 2019].

[3] CARGO. Introduction to bimetalic temperature sensors, 2013. [Online; accessed Dec 12, 2019].

[4] B. Zupancic M. Boltezar G. Cepon, B. Starc. Coupled thermo-structural analysis of a bimetallic strip using the absolute nodal coordinate formulation, 2017.

[5] Michael Smith. *ABAQUS/Standard User's Manual, Version 6.9.* Dassault Systèmes Simulia Corp, United States, 2009.